



Laser & Electronics

Manual CircuitCAM 5.0

English, Version 1.0

LPKF Laser & Electronics AG

Osteriede 7
D-30827 Garbsen

Telephone : ++ 49 - 51 31 - 70 95 - 0
Telefax : ++ 49 - 51 31 - 70 95 - 90
eMail : lpkf@lpkf.de
Homepage : <http://www.lpkf.de>

Copyright (c) 2004 LPKF AG

Distribution and reproduction of this manual and use of its content only permitted with written permission from LPKF AG. Modifications reserved. No liability is assumed for the information contained herein. In particular, we assume no liability for damages resulting from information given, information absent or erroneous information. All trademarks are registered by their owners.

About this manual

This manual is an introduction into the operation of *CircuitCAM* 5.0 for Windows™.

CircuitCAM is a combined Computer Aided Design (CAD) and Computer Aided Manufacturing (CAM) program.

You can use *CircuitCAM* to import, check and edit circuit board production data in various CAM formats, and then output them again into a CAM format (LMD). *CircuitCAM* is particularly useful in calculating the isolation channels between the conductor tracks in circuit board prototype production using LPKF circuit board plotters.

In addition to installation of the *CircuitCAM* software and a general introduction of the *CircuitCAM* user surface, it teaches you the basic features and concept of *CircuitCAM* on the basis of exercises.

This manual will not deal with the basics of Windows. If you are not yet familiar with the different Windows objects, please refer to the Windows documentation first.

In order to receive a better quality of the graphic representation in this tutorial, the representations may partly differ from the original displays on the screen as far as the colors are concerned.

It is the duty of the software user to check the observance of the following points. It must be ensured that

- **the software is only operated in accordance with its designated use and in conjunction with the system delivered;**
- **the complete manual is kept in a legible condition and is always at hand at the software's place of use;**
- **only authorized and sufficiently qualified personnel are entrusted with the handling of the software;**
- **the personnel has read and understood this manual and in particular the safety instructions contained herein.**

I. Conventions used in this manual

Bold text is used to emphasise important information.

Illustrations are numbered. Example: Fig. 5

- › Prompts for actions are identified with an arrow.

Italic sections are used to indicate the reactions consequent on an action.

Word printed in *italics* mark proper names.

Key inscriptions and menu terms are printed in **BOLD CAPITALS**.

II. Notes on the symbols used



Danger! This symbol is used to highlight danger to life or health.



Caution! This symbol is used to identify hazards which may cause damage.



Note: This symbol is used for notes intended to help you avoid faults in operation or to help you improve your procedures.

III. Legend

CircuitCAM : Software for data processing

BoardMaster : Machine control software

IV. Target Group

This Manual is intended for personnel familiar with the basic use of Windows as well as exporting of corresponding data from the CAD programs used (Eagle, Protel... etc.).

Table of contents

| | |
|--|-----------|
| 1.0 Installation | 9 |
| 1.1 System requirements | 9 |
| 1.2 Materials supplied | 9 |
| 1.3 Installation | 9 |
| 1.4 Directories created during installation | 10 |
| 1.5 User preferences | 11 |
| 1.6 General settings | 12 |
| 2.0 Getting started | 13 |
| 2.1 Starting CircuitCAM | 13 |
| 2.2 Using online help | 14 |
| 2.3 Show program information | 14 |
| 2.4 Wizard (Processing Assistant) | 15 |
| 2.4.1 Basic Wizard Functions | 16 |
| 2.5 CircuitCAM application window | 18 |
| 2.5.1 Graphic window | 19 |
| 2.5.2 List window | 20 |
| 2.5.3 Report window | 20 |
| 2.5.4 Toolbars | 21 |
| 2.5.5 Status bar | 22 |
| 2.6 Opening the "Tutor" sample circuit board | 23 |
| 2.7 Closing CircuitCAM | 23 |
| 3.0 Preface to the exercises | 24 |
| 3.1 "Tutor" sample circuit board | 24 |

| | |
|--|-----------|
| 3.2 Listing of the exercise files | 25 |
| 4.0 Exercise 1 | 26 |
| <hr/> | |
| 4.1 Create new CircuitCAM document | 26 |
| 4.2 Customizing grid, unit and coordinates display . . . | 27 |
| 4.3 Import production data | 28 |
| 4.3.1 Aperture list | 28 |
| 4.3.2 Tool list | 34 |
| 4.3.3 Gerber data for solder and component sides | 36 |
| 4.3.4 Excellon data | 39 |
| 4.3.5 Importing the data using the import assignment . | 41 |
| 5.0 Exercise 2 | 42 |
| <hr/> | |
| 5.1 Checking production data | 42 |
| 5.1.1 Working with the view functions | 42 |
| 5.1.2 Working in the layer list window | 43 |
| 5.2 Preparing production data for prototyping | 46 |
| 5.2.1 Deleting graphical objects outside the circuit board | 46 |
| 5.2.2 Moving the circuit board outline to a separate layer | 50 |
| 5.3 Direct Coordinate Entry | 52 |
| 5.3.1 Placing text objects on separate layers | 52 |
| 5.3.2 Isolating with the default job | 53 |
| 5.3.3 Preparing for Production Data Export | 54 |
| 5.3.3.1 Checking The Tool Lists | 55 |
| 5.3.3.2 Export production data | 58 |
| 6.0 Exercise 3 | 60 |
| <hr/> | |
| 6.1 Using contour routing | 60 |
| 6.1.1 Modifiable Milling Direction and Sequence | 64 |

| | |
|---|-----------|
| 6.2 Create rubout area | 65 |
| 7.0 Exercise 4 | 70 |
| 7.1 Customize isolation job. | 70 |
| 7.1.1 CircuitCAM - Lite | 70 |
| 7.1.2 CircuitCAM - PCB | 74 |
| 7.1.3 Insulating Frames (PCBs only) | 78 |
| 7.1.4 Color Assignment for Insulating Tools | 82 |
| 8.0 Exercise 5 | 85 |
| 8.1 Creating a CircuitCAM template file | 85 |
| 8.2 Move layer | 87 |
| 9.0 Exercise 6 | 89 |
| 9.1 Using import assignments | 89 |
| 9.2 Rotate layer | 92 |
| 10.0 Appendix. | 97 |
| 10.1 Gerber Format. | 97 |
| 10.1.1 Use of Gerber Format | 97 |
| 10.1.2 Structure of a Photoplotter | 98 |
| 10.1.3 Gerber Format Syntax | 99 |
| 10.1.3.1 Drawing | 99 |
| 10.1.3.2 Flashing | 100 |
| 10.1.4 Aperture List | 101 |
| 10.1.5 Units, Format and suppressed Zeros. | 102 |
| 10.1.6 Extended Gerber | 103 |
| 10.2 Predefined Layer | 105 |
| 10.3 Importing DXF files | 106 |
| 10.4 Text Function | 111 |
| 10.5 Generating Ground Planes. | 113 |

| | |
|---|------------|
| 10.6 Design Rule Check. | 117 |
| 10.7 CircuitCAM template file. | 120 |
| 10.7.1 Creating your own template file | 121 |
| 10.7.2 Saving an aperture list in the template file | 122 |
| 10.7.3 Saving a modified isolation job. | 124 |
| 10.7.4 Saving a new layer. | 125 |
| 10.7.5 Saving the parameters | 125 |
| 10.8 Prepared isolation jobs. | 126 |
| 10.9 Creating Drilling/Milling Tools | 127 |
| 10.9.1 Definition of New Drilling Tool | 128 |
| 10.9.2 Definition of New Milling Tool. | 132 |
| 10.10 Export Jobs | 136 |
| 10.11 Batch Function (PCB Version Only) | 139 |
| 10.12 Processing AL-front panels (PCB version). . . . | 145 |
| 10.12.1 Prepare front plate outline data | 145 |
| 10.12.2 Preparing cutting and drilling data | 146 |
| 10.12.3 Setting text data | 149 |
| 10.13 Exporting and saving | 150 |
| 10.14 Preparing data for fiducial recognition | 150 |
| 10.15 Producing step-and-repeats (Only PCB Version). . . | 151 |
| 10.16 CircuitCAM 5.0 Hotkeys | 153 |
| 10.17 Updapping Translation File | 155 |
| 10.17.1 Structure of Translation Files | 156 |
| 10.17.2 Tips for Adapting Translation File. | 162 |
| 11.0 Index | 167 |

1.0 Installation

1.1 System requirements

The minimum system requirements for an installation of *CircuitCAM* are:

IBM PC or compatible system with

- Pentium-II processor (Pentium or equivalent processor recommended)
- Min. 256 MB (512 MB recommended)
- VGA colour monitor (recommended resolution 1024 x 768 pixels)
- Microsoft Windows 2000, Windows XP operating system
- (4x speed) CD-ROM drive

CircuitCAM requires approximately 20 MB free hard disk space for its installation.

1.2 Materials supplied

The materials supplied with *CircuitCAM* include:

- 1x CD-ROM
- this manual
- License card

1.3 Installation

As soon as you have inserted the CD in the CD-Rom drive of your PC, and provided the autostart-function of this drive is activated, then the CD-browser will start automatically. Its menu guidance will lead you through the installation of your LPKF programs.

If the autostart function is not activated on your PC, then proceed as follows:

- › Select **RUN...** from the Windows **START** menu.
- › Then click **BROWSE**.
- › Select the *CircuitCAM* installation directory and double click "Setup.exe".

The installation routine for CircuitCAM starts running.

- › Then follow the instructions in the installation program:



Note: Clicking on CANCEL allows you to close the dialog box and cancel the installation.

1.4 Directories created during installation

By default, the installation program creates a "LPKF 50" subdirectory in your program directory. Programs such as *CircuitCAM* and *BoardMaster* are saved here.

The following subdirectories are created in the "LPKF 50" directory for *CircuitCAM*:

- **CCAM** contains the program files for *CircuitCAM*..
- **Data** contains the data files for *CircuitCAM*. This is the default directory in which all newly created data are saved.
- **Ape_Templates** contains the translation files for the import of aperture and tool lists.
- **New_Templates** contains the *CircuitCAM* template files, listed in the New submenu of the *CircuitCAM* File menu.

You can customize the directories for the conversion and template files to your requirements as described in the following section see „User preferences“ on page 11.

1.5 User preferences

On completion of installation, the **CircuitCAM GENERAL SETTINGS** dialog box is displayed; here you can personalize your *CircuitCAM* software:

Figure 1:

› Enter the data that you require in this dialog field.



Note: The Serial Number and the Enabling Number can be found on the license card. You may call up this **GENERAL SETTINGS** dialog box in *CircuitCAM* at any time by selecting the General... line from the **CONFIGURATION** menu.

Changing the language can be done at any time using the **USER** tab. Please take note that *CircuitCAM* must be restarted afterwards in order for the change to take effect.

Clicking on the **DIRECTORIES** tab allows you customize the *CircuitCAM* directories for the translation and template files to your requirements:

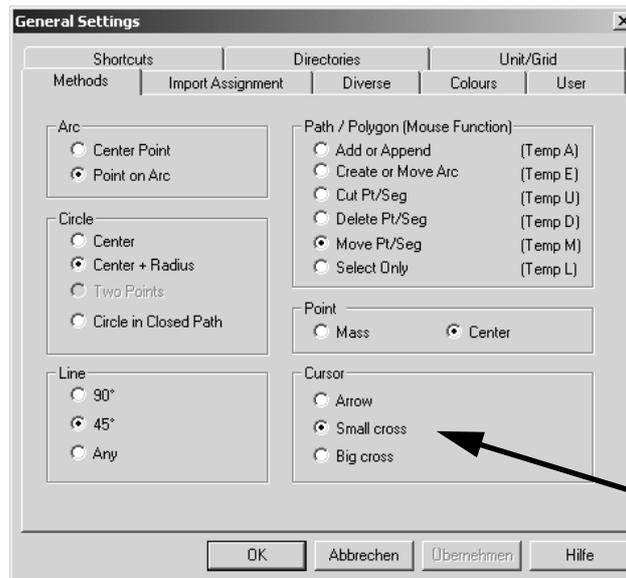
Figure 2:

This settings will be saved in the Windows registry.

1.6 General settings

In the *CircuitCAM* version 5.0 it is possible to select the new setting **BIG CROSS**. Go to the **CONFIGURATION / GENERAL...** menu and click on **METHODS**. Under **CURSOR** you find the new setting **BIG CROSS** (see figure 3) which makes appear a marking cross that extends over the whole screen. This new cross-hair cursor enables the easy modification or creation of layout data.

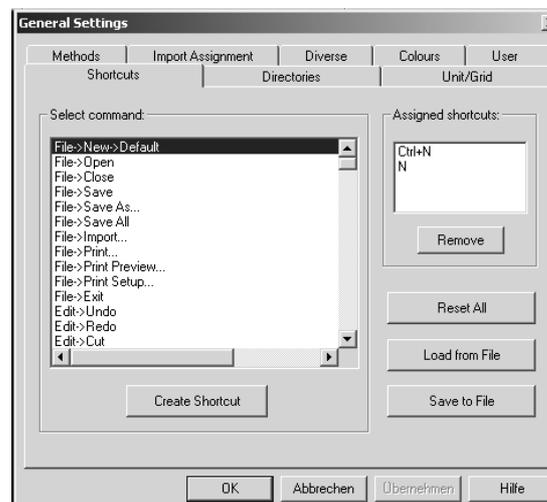
Figure 3: Big cross



Configurable Shortcuts

A key combination can be assigned to any program function of *CircuitCAM*. In order to add new key combinations or modify existing ones, click on the tab **SHORTCUTS** under **CONFIGURATION / GENERAL...**. The different function keys can be used to create new shortcuts and to modify or delete existing ones (see figure 4).

Figure 4: Short Cuts



2.0 Getting started

In this chapter, we would like to give you a first look at *CircuitCAM* and familiarize you with the user interface of *CircuitCAM*. We will show you how you can customize *CircuitCAM* to your requirements and how to use the online help when you require more information on a particular topic.



Note: This tutorial uses the default settings suggested during the installation for the name of the program group and the directories created.

2.1 Starting CircuitCAM

Once installation of *CircuitCAM* is complete, a new LPKF50 program group will be found under **PROGRAMS** in the Windows 2000/XP **START** Menu; it will contain the following icons:

Figure 5:

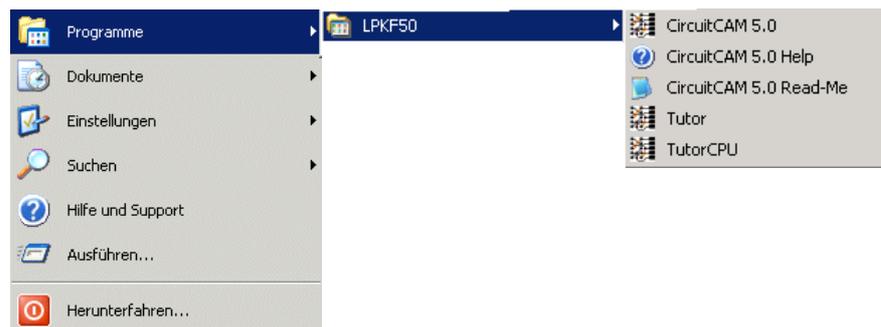


Note: Double-click on the **READ ME** icon to read the latest information on *CircuitCAM* not available when this tutorial was printed.

How to start *CircuitCAM*:

Select the *CircuitCAM* 5.0 shortcut from the LPKF 50 program group in the **START** Menu.

Figure 6:



2.2 Using online help

CircuitCAM has a context sensitive online help system. To help you quickly find the topic or concept you are looking for, the online help system has a table of contents, an index and a search function; in addition you can obtain help on the *CircuitCAM* function that is currently active.

How to call up the Online help topics:

- › Select **HELP TOPICS** from the **HELP** menu.

Figure 7:



How to obtain help on the current function:

- › Press function key **F1**.

How to obtain help on any function:

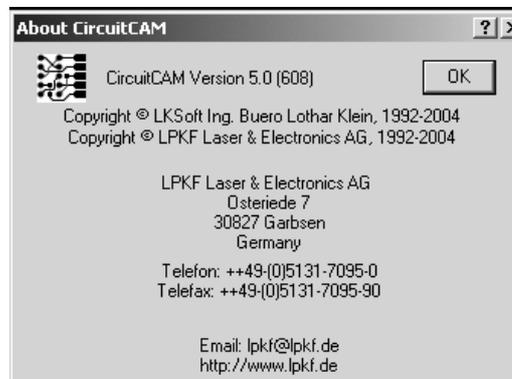
- › Click on  in the **DEFAULT** toolbar.
- › Select an item from the menu bar or click on a button in a toolbar to obtain more detailed information on the function selected.

Please refer to your Windows documentation for information regarding the use of online-help.

2.3 Show program information

You may display the *CircuitCAM* version number by selecting **ABOUT CIRCUITCAM...** from the **HELP** menu.

Figure 8:



This dialog box also contains information of how you may contact us.

2.4 Wizard (Processing Assistant)

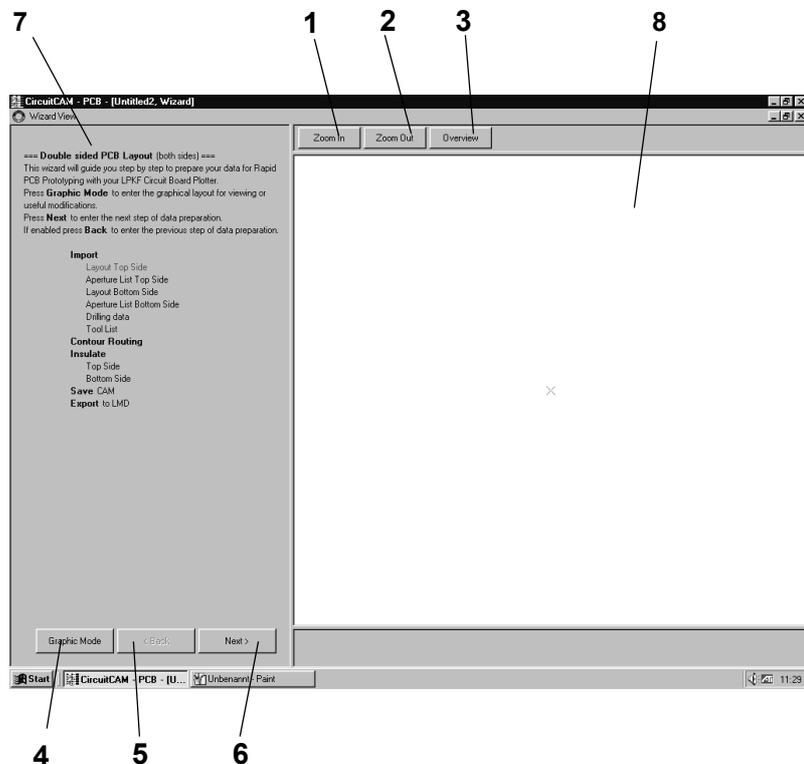
The “Wizard” supports you in processing data and leads you through the entire processing routine with appropriate instructions. This is intended to facilitate the work particularly of those who do not work regularly with *CircuitCAM* or for new users who want to familiarize themselves with the program quickly and reliably.



Note: It is necessary to observe that the Wizard only supports standard, extended Gerber and Excellon data during import.

The Wizard is opened by clicking on the  Wizard button.

Figure 9: Import Wizard



- | | |
|--|--|
| <p>1- Zoom in: Clicking this button increases the size of the data displayed (8).</p> <p>2- Zoom out: Clicking this button reduces the size of the data display (8).</p> <p>3- Overview: Clicking this button displays the entire overview in the data display (8).</p> <p>4- Graphic mode: Clicking this button switches off the import Wizard.</p> | <p>5- Back: Clicking this button goes back one step in the processing sequence.</p> <p>6- Next: Clicking this button jumps forward to the next operating step.</p> <p>7- Report window: This window shows instructions, descriptions and aids on the corresponding operations.</p> <p>8- This window shows the data in the current processing phase.</p> |
|--|--|



Note: In order to work in the normal graphic mode, the Import Wizard can be switched off by clicking **GRAPHIC MODE**. If you want to change to the Wizard while processing in graphic mode, click the  button.

Changeover is possible continuously at any time, whereby the Wizard is automatically updated to the current operating step.

2.4.1 Basic Wizard Functions

Instructions or requests required to perform the appropriate (next) operation are given in the left report window (see figure 9) for each operation. In comparison to Version 4.0, the precise procedure has not changed. Detailed instructions and explanations on the operating steps are given in the appropriate chapters in this Manual. As a matter of principle, the following operations are supported:

Figure 10:Supported functions



Importing top side layout data:

Here, the layout data (for the top side) for the CAD software used is imported into *CircuitCAM*, in order to process it.

Importing top side aperture list:

The aperture list is an ASCII file containing the required apertures with corresponding definition, e.g. aperture number, size and shape. The Gerber data can only be imported graphically correct if the appropriate aperture lists are used.

Importing bottom side layout data:

Here the layout data (bottom side) for the CAD software used is imported into *CircuitCAM* in order to process it.

Importing bottom side aperture list:

Some CAD software uses a separate aperture list for this layer, which must be imported in the same manner as the list for the top side.

Importing drilling data:

The drilling data (Excellon data) is also defined in ASCII files in which the coordinates for the holes to be drilled in the circuit board are defined together with the associated tools.

Importing tool list:

A tool list is comparable to an aperture list; it contains the tools required with corresponding definition, e.g. tool number, type and diameter of the drill.

Contour routing:

If you want to machine the circuit board with a contour router, instead of cutting it with circuit board shears, you can use the contour generator to calculate the routing contour.

Insulating top side:

Here, the entire operation for cutting the circuit board contours on the top side are supported (insulation cutting).

Insulating bottom side:

Here, the entire operation for cutting the circuit board contours on the bottom side are supported (insulation cutting).

Save (*.CAM):

When you have completed all steps for processing your data, the file can be saved with the extension *.CAM.

Exporting to LMD format:

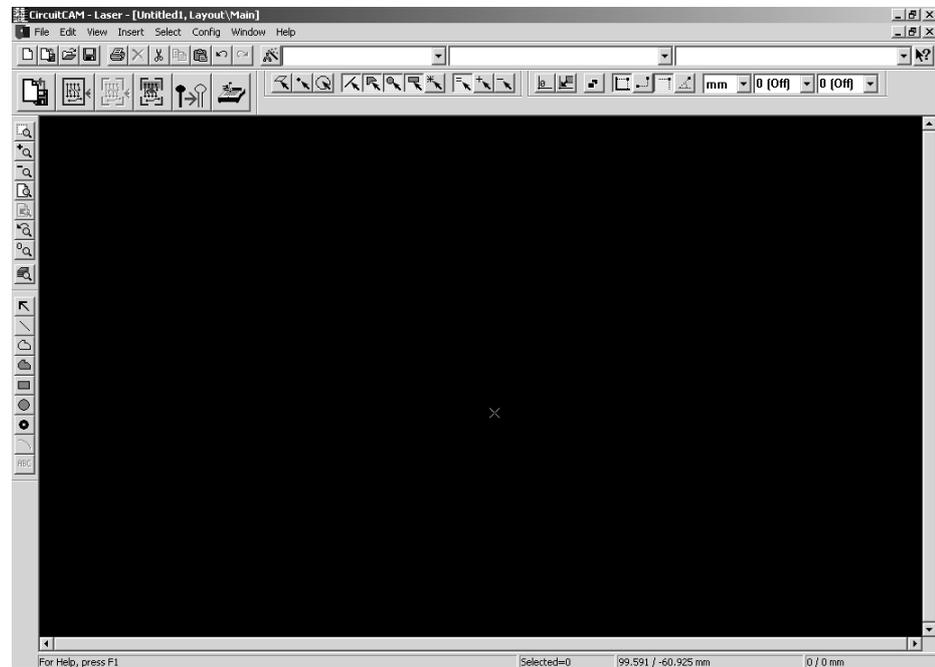
The completed CAM file can be exported in the LMD format which can be processed with the *BoardMaster* software to control the contour router accordingly

As a matter of principle, Standard Gerber, Extended Gerber, Excellon as well aperture lists and tool lists are supported by the for processing the data format.

2.5 CircuitCAM application window

CircuitCAM's working area, menus, toolbars and status bar look like many other Windows application programs to make it easier for you to begin using and working with the program.

Figure 11:



CircuitCAM has three types of document window:

GRAPHIC WINDOWS, LIST WINDOWS and REPORT WINDOWS..

When you start the program, the **GRAPHIC WINDOW** for the "empty" specification file is opened in full-screen mode.



Note: You can open a number of document windows in *CircuitCAM* at the same time. The **WINDOW** menu gives you access to functions for arranging the various document windows and for switching between the document windows.

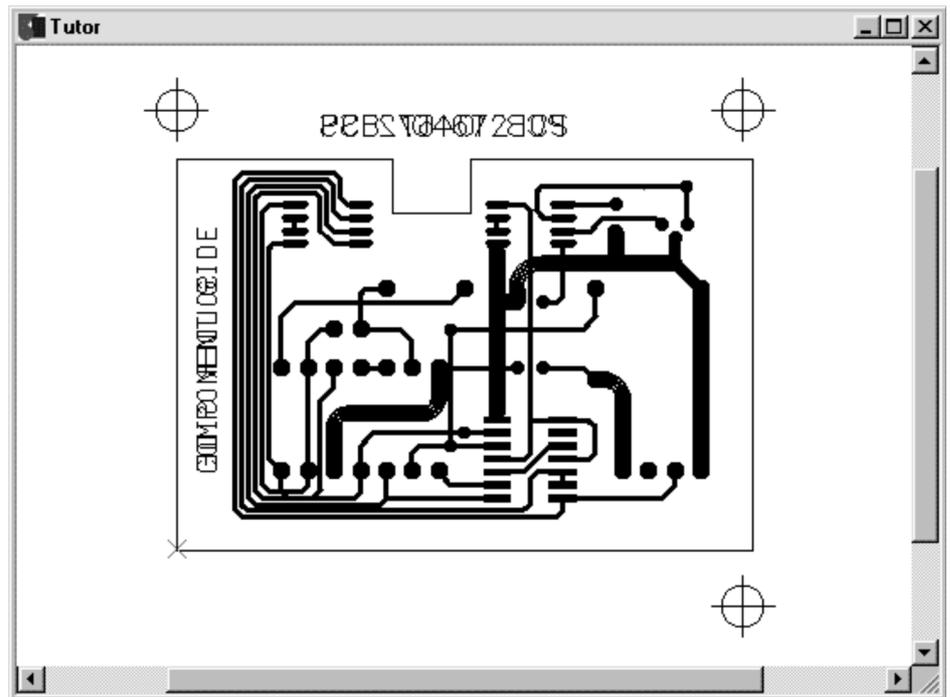
2.5.1 Graphic window

The production data imported for your circuit board are illustrated graphically in the graphic window. This allows you to check that all the data meet the desired requirements.



Note: The view is always of the upper side of the circuit board, so the lower side of the board is illustrated as a mirror image.

Figure 12:

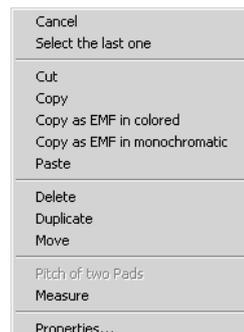


In addition, the graphic window gives you access to functions for editing the production data.



Note: When the mouse pointer is in the graphic window, clicking the right mouse button will access a context menu for editing selected objects:

Figure 13:

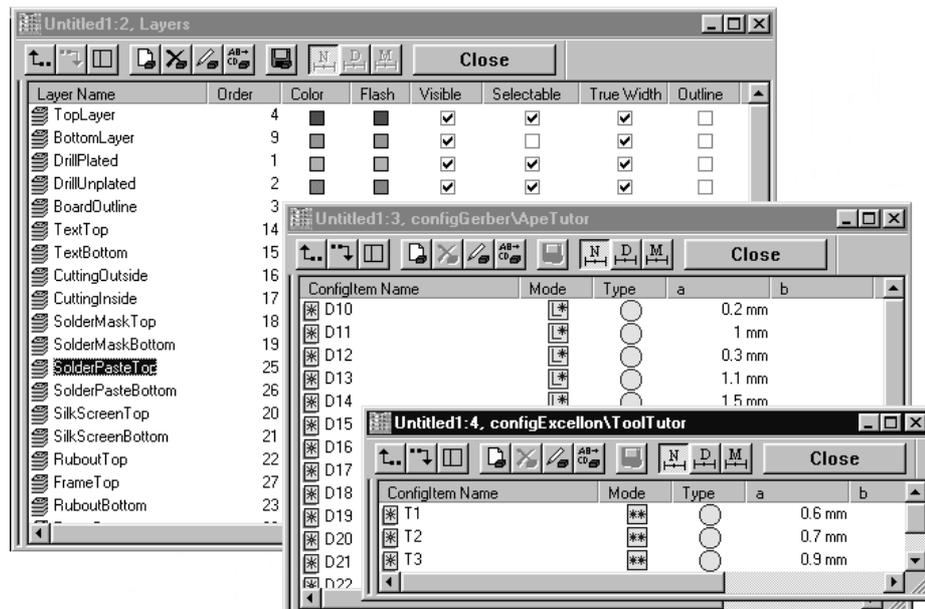


2.5.2 List window

The list window displays the apertures and tools imported and the layers in tabular form with their corresponding definitions. You are able to edit, delete or add new entries.

The opening of following windows will be explained later (see „Aperture list“ on page 28).

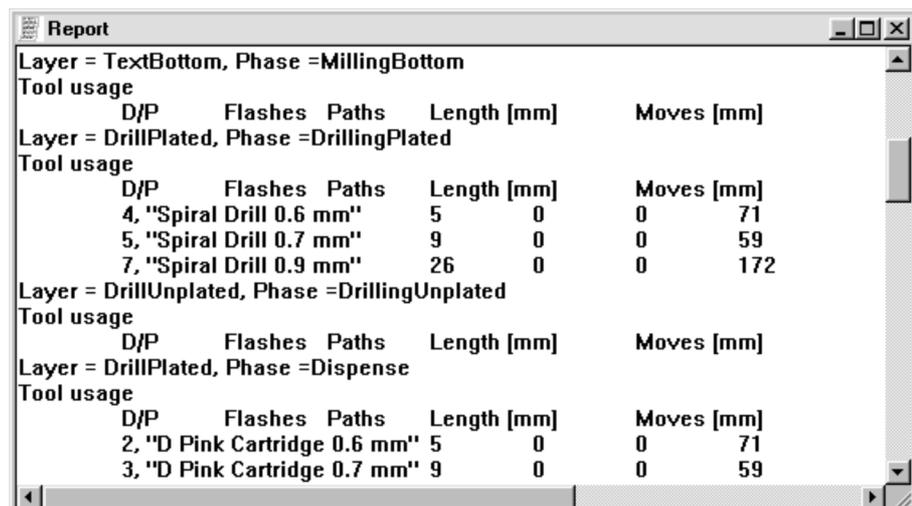
Figure 14:



2.5.3 Report window

The report window shows a log of functions such as export of the production data, and displays any error messages.

Figure 15:



| Report | | | | | |
|--|---------------------------|-------|-------------|------------|-----|
| Layer = TextBottom, Phase =MillingBottom | | | | | |
| Tool usage | | | | | |
| D/P | Flashes | Paths | Length [mm] | Moves [mm] | |
| Layer = DrillPlated, Phase =DrillingPlated | | | | | |
| Tool usage | | | | | |
| D/P | Flashes | Paths | Length [mm] | Moves [mm] | |
| 4, | "Spiral Drill 0.6 mm" | 5 | 0 | 0 | 71 |
| 5, | "Spiral Drill 0.7 mm" | 9 | 0 | 0 | 59 |
| 7, | "Spiral Drill 0.9 mm" | 26 | 0 | 0 | 172 |
| Layer = DrillUnplated, Phase =DrillingUnplated | | | | | |
| Tool usage | | | | | |
| D/P | Flashes | Paths | Length [mm] | Moves [mm] | |
| Layer = DrillPlated, Phase =Dispense | | | | | |
| Tool usage | | | | | |
| D/P | Flashes | Paths | Length [mm] | Moves [mm] | |
| 2, | "D Pink Cartridge 0.6 mm" | 5 | 0 | 0 | 71 |
| 3, | "D Pink Cartridge 0.7 mm" | 9 | 0 | 0 | 59 |

2.5.4 Toolbars

CircuitCAM has a number of toolbars which group together associated functions:

DEFAULT

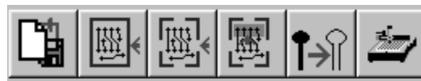
Figure 16:



Standard functions such as open, save, print and list boxes for layers, tool lists and aperture/tool

FRONT TO END

Figure 17:



Functions to the generation of milling- and drilling data: importing, contour routing, break-out-tabs insert, Insulate all Layer, exporting LMD - file.

View

Figure 18:



Functions for zooming in/zooming out in the graphical view and for setting the layer properties.

Grid+unit

Figure 19:



Functions for sliding layers on top of each other and for displaying the unit, buttons to define the origin of coordinates and of the coordinate system, as well as lists for unit and grid.

Select

Figure 20:



Functions for marking graphical objects

Graphics

Figure 21:



Functions for adding graphical objects

How to show or hide a toolbar:

- › Select the toolbar you wish to display or hide from the **Toolbars** submenu under the **View** menu.

How to move a toolbar:

- › Place the mouse pointer on the area between the buttons on the toolbar or on the title bar of a floating toolbar.
- › Drag the toolbar to a new position with the left mouse button pressed. The toolbar will automatically be docked at the edge of the window if you drag the toolbar to the edge of the application window.

2.5.5 Status bar

The status bar at the bottom of the application window displays information regarding the object or function selected.

Figure 22:



The status bar is divided into four information areas:

- Textual information
- Number of objects selected
- Current coordinates of the cursor
- Coordinates of the anchor point ✖

You can also show or display the status bar by selecting the **STATUS BAR** entry from the **TOOLBARS** submenu of the **VIEW** menu.

2.6 Opening the "Tutor" sample circuit board

Production data for two sample circuit boards are supplied with *CircuitCAM*. We recommend that you use the "Tutor" sample circuit board when you try out the *CircuitCAM* functions for yourself.

How to open the "Tutor" sample circuit board:

- › Select the entry **OPEN** in the menu **FILE**.
- › Select the directory C:\LPKF50\Data.
- › Mark the file **TUTOR** in the interactive box **OPEN**.
- › Click **OPEN** to open the file "**TUTOR.CAM**" or click **QUIT** to discontinue the procedure.



Note: If you have not started *CircuitCAM* you can open the sample circuit board together with *CircuitCAM* by double-clicking **TUTOR** symbol in the program group **LPKF40**. The same applies for the sample circuit board "TUTORCPU".

2.7 Closing CircuitCAM

How to exit *CircuitCAM*:

1. using the keyboard:

- › Press **ALT + F4**.

2. using the mouse:

- › Select Exit from the File menu
or click on  in the top right-hand corner of the application window.

3.0 Preface to the exercises

In the following sections you will become familiar with the basic functions of *CircuitCAM* by working through exercises.

In the first exercise, you will load production data and prepare it for circuit board prototyping. This exercise is somewhat more extensive than the remaining exercises, as it covers the entire sequence of operations required to generate production data suitable for passing on to LPKF *BoardMaster*.

The remaining exercises show you how to correct production data which does not work together, perhaps because a layer has been rotated or mirrored in the export of production data from the circuit board CAD program used or because there is an offset between the various types of production data. In addition, these sections will introduce you to functions in *CircuitCAM* which will help you, so far as is possible, in the preparation of production data for prototyping.

3.1 "Tutor" sample circuit board

The circuit board you will be using in the following exercises is a double-sided PCB with conventional and SMD components. The SMD components are located on the solder side (the bottom side) of the circuit board:

Figure 23:

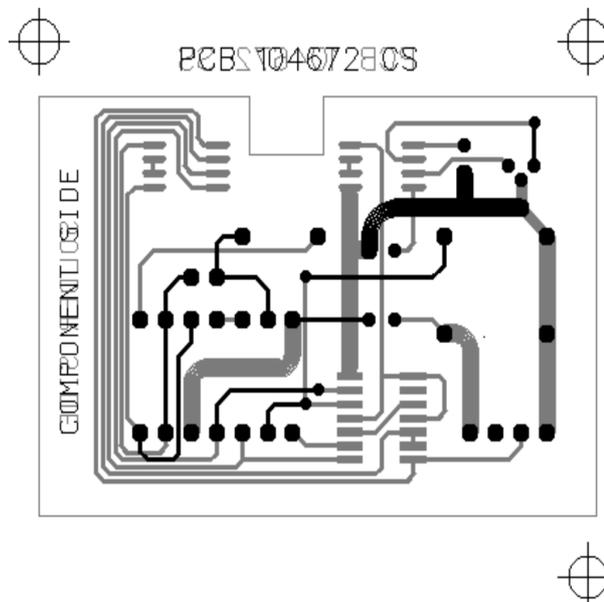
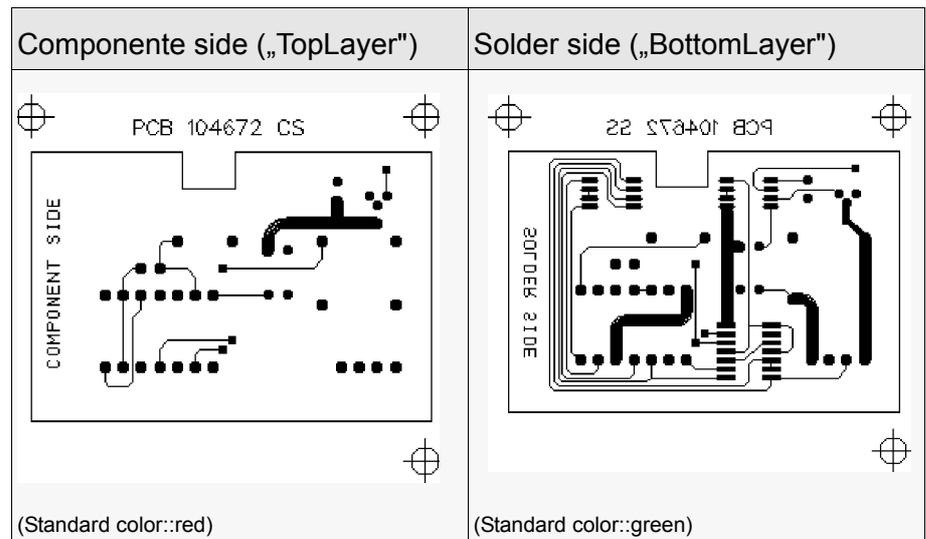


Figure 24:



3.2 Listing of the exercise files

You will work through the exercises using prepared production data for the "**Tutor**" sample circuit board which were copied to the "**Data**" *CircuitCAM* subdirectory during installation.

Production data for the "**Tutor**" sample circuit board:

| | |
|----------------|--|
| TUTORAPE.TXT | Translation file for the aperture lists (Ape_Templates subdirectory) |
| TUTORTOOLS.TXT | Translation file for the tool lists (Ape_Templates subdirectory) |
| TUTOR.CAM | CAM data, <i>CircuitCAM</i> version 5.0 format |
| TUTOR.WHL | Aperture list for Gerber data |
| TUTOR.NCT | Tool list for Excellon data |
| TUTOR1.CMP | Gerber data for the component side |
| TUTOR1.SLD | Gerber data for the solder side |
| TUTOR1.DRL | Excellon data |
| TUTOR2.CMP | Gerber data for the component side |
| TUTOR2.SLD | Gerber data for the solder side |
| TUTOR2.DRL | Excellon data with offset to the Gerber data |
| TUTOR3.SLD | Gerber data for the component side |
| TUTOR3.SLD | Gerber data for the solder side |
| TUTOR3.DRL | Excellon data rotated 90° against the Gerber data |



Note: The filenames and file extensions for the production data can be chosen freely and are not standardized.

4.0 Exercise 1

In this chapter, you learn the adjustment of **UNIT + SCREENS**, to open a new document, as well as importing of the CAD-production-data.

The sequence of operations in *CircuitCAM* may be divided up into the following three phases:

1. Import production data
2. Prepare production data for prototyping
3. Export production data

4.1 Create new CircuitCAM document

If you wish to edit production data in *CircuitCAM*, you must first open a *CircuitCAM* document into which the production data will be imported.

CircuitCAM offers two ways of creating a new document:

- You may use the document based on the template file "**Default.cat**" which is opened automatically in *CircuitCAM* when the program starts.
- You can create a new document by selecting a preconfigured template file from the **NEW** submenu under the **FILE** menu.

(You will be learning more about template files in exercise 5.)



Note: New documents are always named "**UNTITLED**" followed by a number. You may save these documents as a file by selecting **SAVE as** from the **FILE** menu.

In this exercise you will be importing the production data into the document based on the template file "**DEFAULT.CAT**" which is opened automatically in *CircuitCAM* when the program starts.

How to create a new *CircuitCAM* document:

- › Start the *CircuitCAM* program.
The document "**UNTITLED1**" will already be opened.
- › Select **SAVE as** from the **FILE** menu.
- › Enter the filename **TUTOR1** in the **FILENAME** text box of the **SAVE as** dialog box.
- › Click on **SAVE** to save the document as a file under the name "**TUTOR1.CAM**".



Note: By clicking on **CANCEL** or pressing **ESC** you can close the dialog box and cancel the function. This works in all dialog boxes.

The file extension **".CAM"** is automatically appended to the filename.

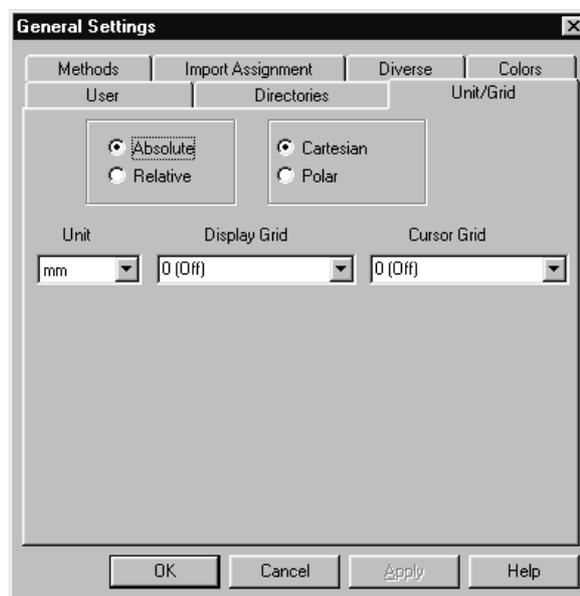
4.2 Customizing grid, unit and coordinates display

CircuitCAM has preference options for the grid, unit and coordinates display to assist you in accurate placement in the graphic window.

Up to now, you have been using the default settings for *CircuitCAM* while working in the graphic window. In this section we would like to show you how you can customize these settings to your requirements. In particular, it can be very useful to customize these preferences when you have significant amounts of work to do on graphical objects.

The functions for setting the grid, unit and coordinates display are grouped together in the **CONFIGURATION\GENERAL\UNIT/GRID** toolbar:

Figure 25:



The additional functions in the **GRID/UNIT** property sheet are:

Figure 26:

| | |
|--|---|
| | <p>Switching over the coordinate display in the status bar between ABSOLUTE COORDINATES and RELATIVE COORDINATES.</p> |
| | <p>Switching over the display format in the status bar between CARTESIAN COORDINATES and POLAR COORDINATES.</p> |

You can select the preferences in the three following list windows:



DISPLAY UNIT

All values will be displayed in the unit selected here.



DISPLAY GRID

Shows the dots in the grid selected here in the background.



CURSOR GRID

Allows placement in the grid selected here, also known as snap to grid.

4.3 Import production data

In the following steps, you will import into *CircuitCAM* the production data a circuit board CAD program generates for the production of the circuit board.

you can import the following file formats in CircuitCAM:

| | |
|----------------------------|----------------------------|
| - Standard Gerber | - HPGL (Only PCB Version) |
| - Extended Gerber | - DXF |
| - Excellon | - ODB++ (Only PCB Version) |
| - Sieb & Meyer | |
| - Barco (Only PCB Version) | |

In the first exercise includes:

- the aperture and tool lists with the corresponding definitions
- the circuit board data for the component side in Gerber format
- the circuit board data for the solder side in Gerber format
- the drilling data in Excellon format

4.3.1 Aperture list

The aperture list is an ASCII file listing the apertures required with corresponding definition, e.g aperture number, type and size. The Gerber data can only be graphically correctly converted with the matching aperture list.

For more information see „Gerber Format“ on page 97.



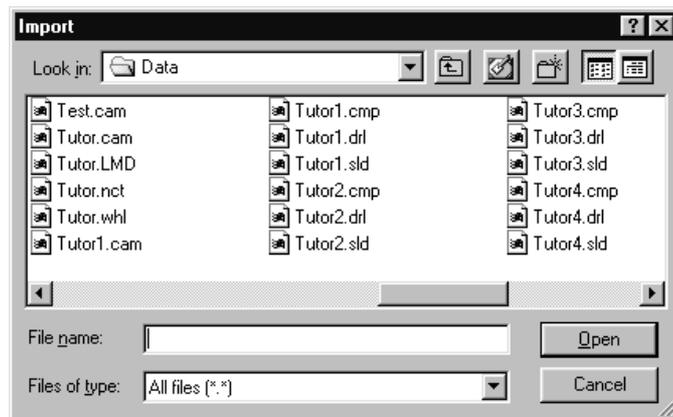
Note: An incorrect aperture list definition will be revealed by incorrect representation of the pads and conductor tracks in the graphic window, i.e these objects will be too small or too large in dimension, or displayed in the incorrect form.

How to import the aperture list:

- › Select **IMPORT...** from the **FILE** menu or click on  in the **FRON TO END** toolbar.

*The **OPEN** dialog box is displayed:*

Figure 27:

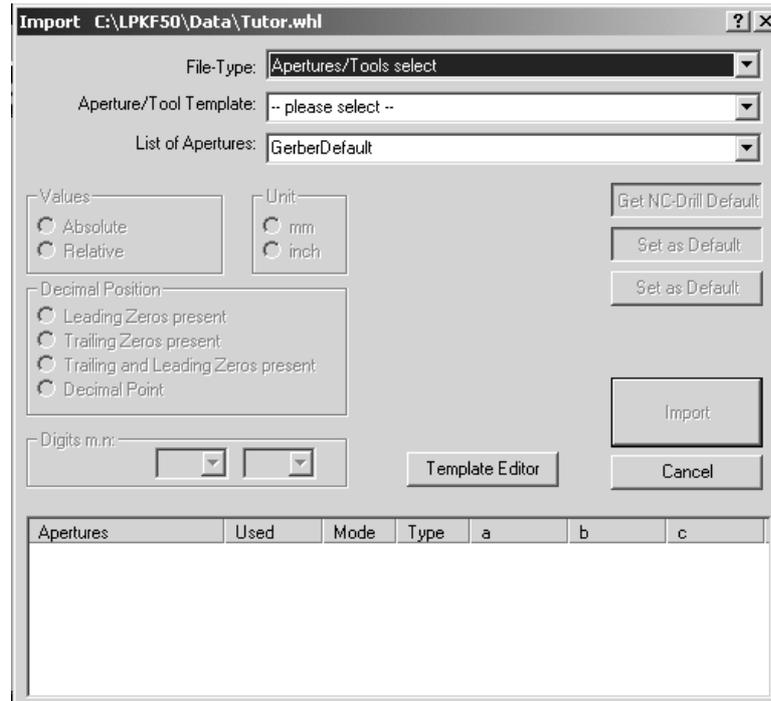


- › Select the file **TUTOR.WHL** and click on **OPEN** to import the aperture list.

*The **IMPORT** dialog box, the header which shows the name of the file you have selected, will be displayed.*

CircuitCAM automatically detects that this is an aperture list or a tool list (Apertures/Tools select). The first time the two fields Apertures/Tool template and Aperture list are empty. Make your entries here.

Figure 28:

**FILE-TYPE:**

Here if not detected automatically, you must specify the type of the file being imported so that *CircuitCAM* can interpret the data correctly.

APERTURE/TOOL TEMPLATE:

Appears if the file type **APERTURES/TOOLS** is selected.

Here you should select a translation file for your production data.

CircuitCAM needs this translation file to convert the data correctly as the aperture and tool lists used by circuit board CAD programs are not standardized.

More information on the topic of translation files may be found in the appendix, page 78 and following).

APERTURE LIST / TOOL LIST:

Here you should enter a name for the aperture list which will be used within *CircuitCAM*.

LAYER:

Appears if the file type Gerber or Excellon is selected. Here you should specify the layer to which the production data is to be assigned.

Template Editor

By clicking the button **TEMPLATE EDITOR** an additional program for simple creation of new translation files for apertures and tool lists can be opened.

- › Select the **TUTORAPE1.TXT** translation file from the Aperture/Tool Template list box

Figure 29:



- › Enter „**APETUTOR1**“ as the name for the aperture list in the listbox labeled **APERTURE LIST**.

Figure 30:



All the settings necessary for importing the aperture list are now completed. If the translation file has been correctly selected, i.e. it matches the aperture list to be imported 100%, the result will be displayed in the list box in the lower part of the Import dialog box.

Figure 31:

| Apertures | Used | Mode | Type | a | b | c |
|-----------|------|------|------|--------|---|---|
| * D10 | ✓ | * | ○ | 0.2 mm | | |
| * D11 | ✓ | * | ○ | 1 mm | | |
| * D12 | ✓ | * | ○ | 0.3 mm | | |
| * D13 | ✓ | * | ○ | 1.1 mm | | |
| * D14 | ✓ | * | ○ | 1.5 mm | | |
| * D15 | ✓ | * | ○ | 1.3 mm | | |



Caution!!! The exercise is interrupted here. The following paragraphs are additional information.

If some apertures/tools are undefined due to an erroneous translation file, i.e. marked with a question mark in the list window, you can define the aperture/tools here manually (see figure below).

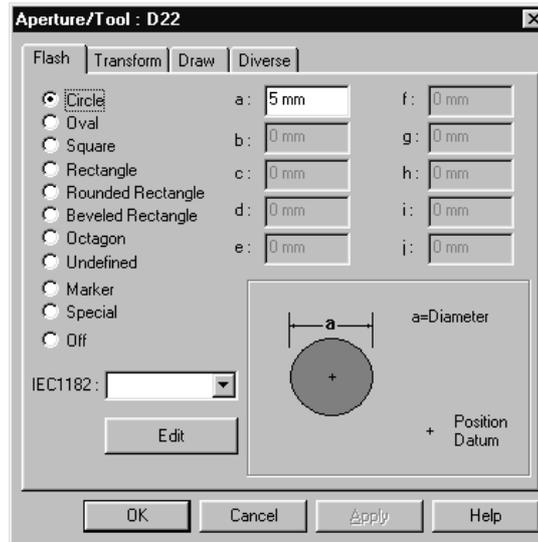
Figure 32: Undefined tools

| Apertures | Used | Mode | Type | a |
|-----------|------|------|------|---|
| * D10 | ✓ | ** | ? | |
| * D12 | ✓ | ** | ? | |
| * D13 | ✓ | ** | ? | |
| * D14 | ✓ | ** | ? | |
| * D15 | ✓ | ** | ? | |
| * D16 | ✓ | ** | ? | |

For this purpose, click one of the apertures/tools which are undefined (“?”) with the right mouse key.

The following window appears:

Figure 33: Aperture/tool



Note: The next 4 operations should be considered as examples: They are not part of the exercises in this manual. They are intended only to illustrate how an aperture can be defined manually and correctly (in this example, definition of a flash aperture):

- › Select the **CIRCLE** in the register **FLASH**
- › Enter the value 0.7 mm for **Parameter a**:
- › Select the index card **DRAW**.
- › Click the checkboxes **DERIVED FROM FLASH DEFINITION** and **AVAILABLE**.

With these settings, this aperture is defined as a pure flash aperture.



Note: Detailed information on the Aperture/Tool box is given in the Appendix.

Continue with this exercise:

If you wish also to use the translation file you have selected here for importing the aperture list for subsequent circuit boards, click on **SET AS DEFAULT** to save this setting as a default setting. The default aperture list name **GERBERDEFAULT** will be used in this case. This setting can then be used in further sessions by clicking on the Get **GERBER DEFAULT** button.

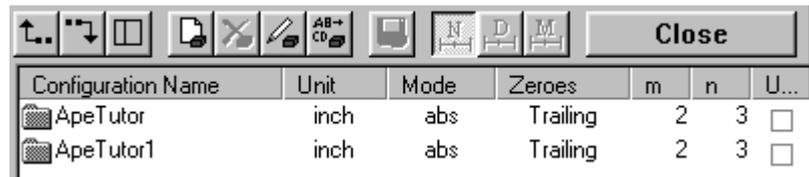
- › Click on **IMPORT** to import the aperture list.

You can show the imported aperture list in a list window.

- › Select the **GERBER APERTURE LIST...** line from the **FORMAT** submenu of the **CONFIGURATION** menu.

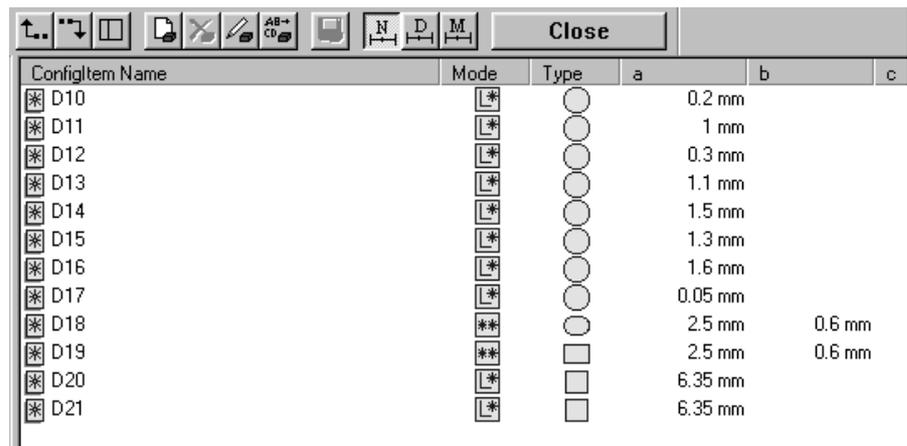
CircuitCAM shows the imported Gerber aperture lists in the list window:

Figure 34:



- › Double-click on the “**APETUTOR1**” entry to display the definitions for this aperture list:

Figure 35:



Note: You can check and edit the aperture definitions in the list window by clicking on the line of the aperture you wish to check or edit with the right mouse button.

It is not necessary to import the aperture list if the Gerber data are available in Extended Gerber format (274X, Gerber-X with mass parameter), as the Gerber data will already contain the aperture definitions.

You will find a full description of the various functions in the list window in the *CircuitCAM* online help.

- › Click on **CLOSE** to close the list window.

The aperture list has now been imported into CircuitCAM.



Note: Save your data to protect against data loss. For this purpose, select the **SAVE** in the menu **FILE**.

- › Select the name “Tutor1” and click **SAVE**.

4.3.2 Tool list

A tool list is similar to an aperture list, it lists the tools required with their definition, i.e. tool number, type and diameter of the drill.

The steps for importing the tool list are similar.

How to import the tool list:

- › Select **IMPORT...** from the **FILE** menu... or click on  in the **FRONT TO END** toolbar.

The **OPEN** dialog box is displayed.

- › Select the file “**TUTOR.NCT**“ and click on **OPEN** to import the tool list.

*The **IMPORT** dialog box, the header of which shows the name of the file you have selected, will be displayed.*

CircuitCAM detects automatically that this is an aperture list or a tool list (Apertures/Tools select). Initially, the two fields **APERTURES/TOOL TEMPLATE** and **APERTURE LIST** are empty. Make your entries here.

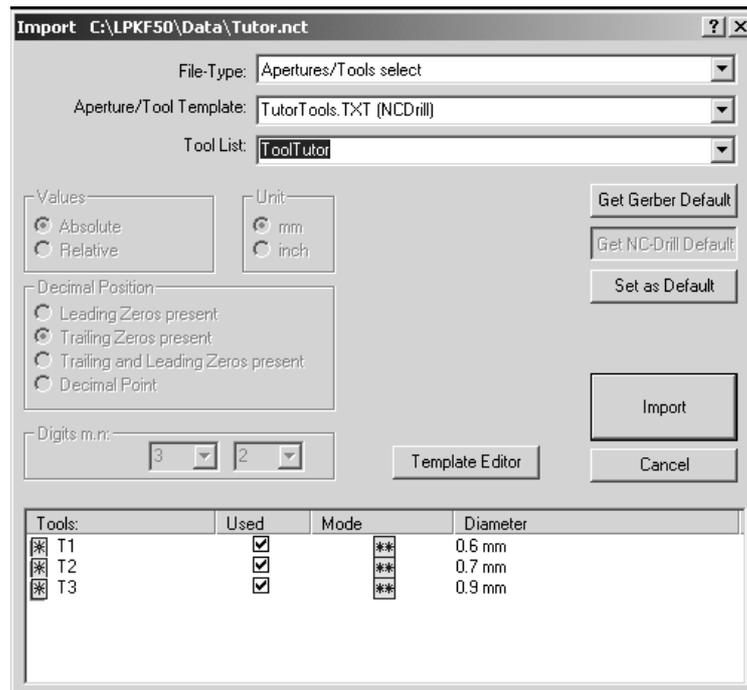
- › Select the “TutorTools.txt“ translation file from the **APERTURE/TOOL** translation list box.

Figure 36:



- › Enter “**TOOLTUTOR1**“ as the name for the tool list in the **TOOL LIST** edit field.

Figure 37:



All the settings necessary for importing the tool list are now completed. If the translation file has been correctly selected, i.e it matches the tool list to be imported 100%, the result will be displayed in the list box in the lower part of the **IMPORT** dialog box.

| Tools : | Used | Mode | Diameter |
|---------|------|------|----------|
| T1 | ✓ | *** | 0.6 mm |
| T2 | ✓ | *** | 0.7 mm |
| T3 | ✓ | *** | 0.9 mm |

- › If you wish also to use the translation file you have selected here for importing the tool list for subsequent circuit boards, click on **SET AS DEFAULT** to save this setting as a default setting.

The default tool list name **NCDRILLDEFAULT** will be used in this case. This setting can then be used in further sessions by clicking on the Get **NC-DRILL DEFAULT** button.

- › Click on **IMPORT** to import the tool list.

You can display the imported aperture list in a list window.

- › Select the **EXCELLON TOOL LIST...** line from the **FORMAT** submenu of the **CONFIGURATION** menu to do this.

CircuitCAM shows the imported Excellon tool lists in the list window:

Figure 38:

| Configuration Name | Unit | Mode | Zeros | m | n | U... |
|--------------------|------|------|----------|---|---|--------------------------|
| ToolTutor | mm | abs | trailing | 3 | 2 | <input type="checkbox"/> |
| ToolTutor1 | mm | abs | trailing | 3 | 2 | <input type="checkbox"/> |

- › Double-click on the “**TOOLTUTOR1**” entry to display the definitions for this tool list:

Figure 39:

| ConfigItem Name | Mode | Type | a | b |
|-----------------|------|---|--------|---|
| * T1 | ** |  | 0.6 mm | |
| * T2 | ** |  | 0.7 mm | |
| * T3 | ** |  | 0.9 mm | |

- › Click on **CLOSE** to close the function.

The tool list has now been imported into CircuitCAM.



Note: You will find a detailed description of individual functions within the list window in the CircuitCAM online help.



Save your data to protect against data loss. Do this by selecting SAVE from the FILE menu. The document will be saved under the filename “TUTOR1” already entered.

4.3.3 Gerber data for solder and component sides

Once you have imported and thus defined the aperture and tools into *CircuitCAM*, you are in a position to import the Gerber data for the solder and component sides.

How to import the Gerber data:

- › Select **IMPORT...** from the **FILE** menu... or click on  in the FRONT TO END toolbar.

The OPEN dialog box is displayed.



Note: The import function in CircuitCAM is capable of importing a number of files containing production data in a single process.

You can select multiple files by holding down the **CTRL** key as you make your selections.



You can select a range of files by selecting the first file in the range, holding down the **SHIFT** key and selecting the last file in the range. All files within this range will now be selected.

- › Hold down the **CTRL** key and select the files “**TUTOR1.SLD**“ and “**TUTOR1.CMP**“.
- › Click on **OPEN** to import the Gerber data.

The **IMPORT** dialog box is now displayed; here you define the file type of the file to be imported, and assign each of the files selected in sequence to an aperture list, tool list and layer.

Assignments for the Gerber file Tutor1.cmp (component side)



Note: Select the Gerber option under File-Type if *CircuitCAM* does not automatically interpret the file to be imported as a Gerber file.

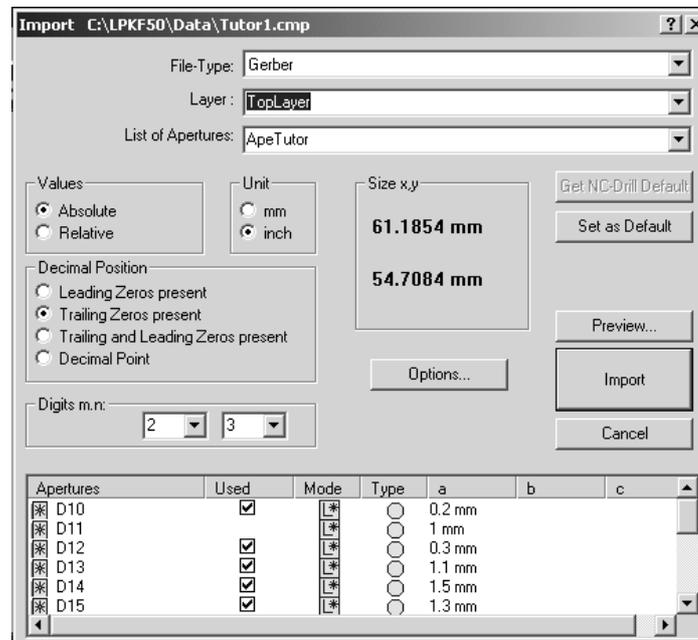
- › Assign the “**APETUTOR1**” aperture list previously defined to the Gerber data in the Aperture List list field by dropping down the list box with a click and selecting the “**APETUTOR1**” line:

Figure 40:



- › Select the name **TOPLAYER** from the **LAYER** list box to assign the Gerber data for the component side to this layer.

Figure 41:



- › Click on the button **SET AS DEFAULT** to store the current setting of the layer. By clicking on the button **DEFAULT** the preset default setting with the aperture list name Gerberdefault is used. Please take note that only one default setting can be used, i.e. the layer name must be newly allocated for a second Gerber file.

- › Click on **IMPORT** to import the file and to open the **IMPORT** dialog box for the import settings for the next file selected.

Data already imported will be shown in the background of the graphics window.

Assignments for the Gerber file Tutor1.sld (solder side):

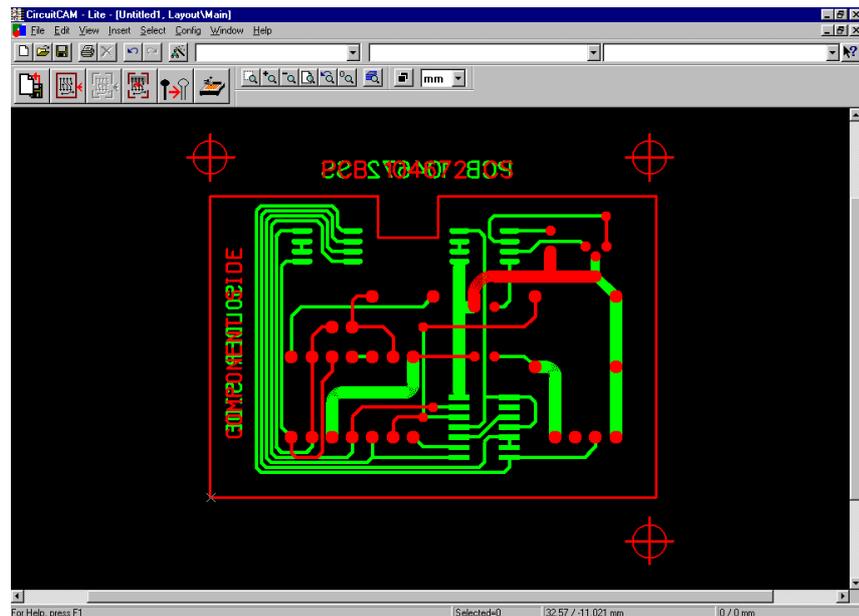


Note: Select Gerber from the File-Type combo box if *CircuitCAM* does not automatically interpret the file to be imported as a Gerber file.

- › Select in the list field **APERTURELIST** previously imported aperture list „**APETUTOR1**“.
- › Select in the listbox **LAYER** „**BOTTOMLAYER**“ to assign the gerber files for the bottom-layer.
- › Click on **IMPORT** to close the function.

The Gerber data for the solder and component sides have now been imported into CircuitCAM and will be displayed in the graphic window.

Figure 42:



Note: We recommend that you always assign the names **BOTTOMLAYER** and **TOPLAYER** to the layers for the solder and component sides, as *CircuitCAM* uses these layer names for the preset isolation job.

You can preview the data before it is imported by clicking on the **PREVIEW**- button to allow you to rectify incorrect settings before importing the data.

The aperture list selected is displayed in the lower section of the dialog box to give you the opportunity of detecting missing or incorrectly defined apertures in advance.

The dimensions of the data to be imported and the Gerber parameters (unit, values, decimal position and M.N) are also displayed.



Save your data to protect against data loss. Do this by selecting SAVE from the FILE menu. The document will be saved under the filename “TUTOR1” already entered.

4.3.4 Excellon data

The Excellon data are also in an ASCII file; this defines the drilling coordinates for the pads and plated-through holes on the circuit board with the associated tools.

How to import the Excellon data:

- › Select **IMPORT...** from the **FILE** menu, or click on  in the **FRONT-TO-END** toolbar.

The OPEN dialog box is displayed.

- › Select the name “**TUTOR1.DRL**“ and click on **OPEN** to import the Excellon-file.

The IMPORT dialog box is now displayed; here you assign each of the files selected an aperture/tool list and a layer.

Assignments for the Excellon file Tutor1.drl:



Note: Select Excellon from the file type list box if CircuitCAM does not automatically interpret the file to be imported as an Excellon file.

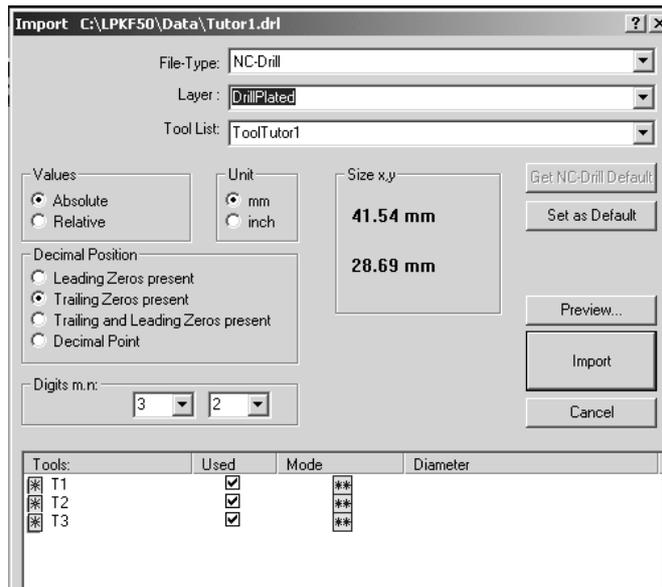
- › Assign the “**TOOLTUTOR1**“ tool list previously imported to the Excellon data in the **TOOL LIST** list field by dropping down the list box with a click and selecting the “**TOOLTUTOR1**“ line:

Figure 43:



- › Select the name **DRILLPLATED** from the **LAYER** list box to assign the Excellon data to this layer.

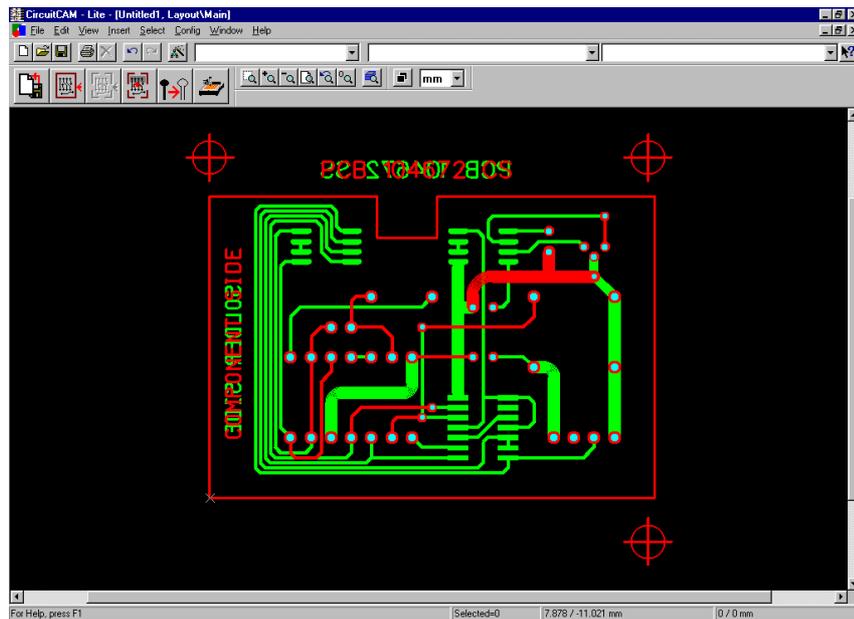
Figure 44:



› Click on **IMPORT** to import the file.

The Excellon data have now been imported into CircuitCAM and will be displayed in the graphic window.

Figure 45:



Note: You may import data into the existing data in *CircuitCAM* at any time. All production data (Gerber and Excellon) can be imported in one step.



Save your work to protect against data loss. Do this by selecting Save from the File menu. The document will be saved under the filename "Tutor1" already entered.

The DXF Data import will be explained in the appendix, because it's not component of this exercise.

4.3.5 Importing the data using the import assignment

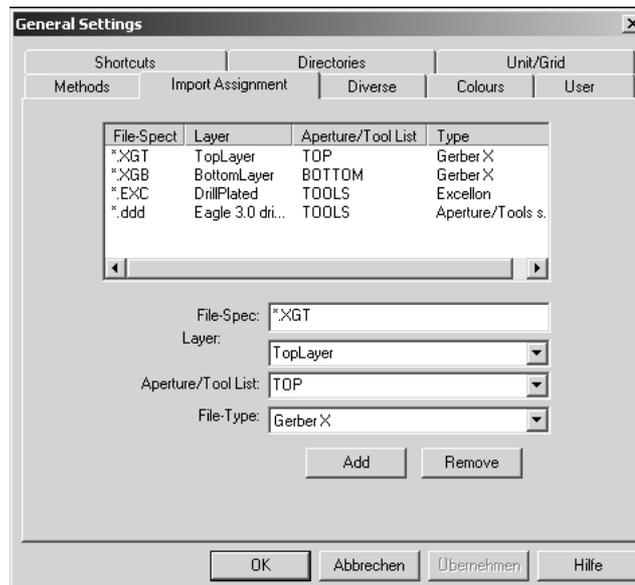
Importing data is significantly easier and quicker if you work with the aid of import assignment. Before importing, the parameters that were assigned manually previously in this exercise are predefined in a dialog box. This means that the data is displayed on the screen as soon as it has been selected for import.

- › Select “**TUTOR.CAT**“ from the **FILE-NEW** menu.

Import assignment for the Tutor files has already been defined in this template file.

- › Open the **GENERAL SETTINGS** dialog box from **CONFIGURATION-GENERAL...** and click on the **IMPORT Assignments** tab.

Figure 46:



All the settings required for importing the Tutor files quickly have been predefined here. Set your Import assignments in the same way for the data you wish to import and save it to your own template file. (See Exercise 6, Own template files).

- › Now repeat the import as described above. But select all the files to be imported (Tutor1.nct, Tutor1.whl, Tutor1.cmp, Tutor1.sld and Tutor1.drl) using the **CTRL** key.

*Once you have confirmed the selection with **OK**, the data will be imported and displayed on screen immediately, without you having to go to the **IMPORT** dialog box.*

5.0 Exercise 2

5.1 Checking production data

We recommend that you make a visual check to ensure that the imported data have been correctly interpreted by *CircuitCAM* once all the data required for prototyping the circuit board have been imported.



Note: You will find a listing of possible display errors, their causes and how to rectify them in the "Tips & tricks in customizing the translation file" section of the appendix on page 84.

The following section will describe the functions required to change the view in *CircuitCAM* to allow you to complete a visual check on the imported data. These functions have an important role to play in CAD programs in making it possible for you to work with the graphical objects with precision.

5.1.1 Working with the view functions

CircuitCAM has a View menu and a View toolbar. These group together the functions you need to adapt the scale or the selection of the graphical representation to your requirements.

There are also a number of keyboard functions for quickly zooming and moving the selection. Some of this keyboard functions act to the current cursor position.

How to change the graphical view:



ZOOM WINDOW

Activate this button and drag with the left mouse button pressed to select an area you wish to see in enlarged form.



ZOOM IN

on this button to enlarge the view by a factor of 2.
Keyboard: **PAGE UP**



ZOOM OUT

Click on this button to reduce the view by a factor of 2.
Keyboard: **PAGE DOWN**



OVERVIEW

Click on this button to display all graphical objects in the workspace.
Keyboard: **HOME**



PREVIOUS

Click on this button to return to the previous view.
Keyboard: **INSERT**



REFRESH

Click on this button to have the graphical objects redrawn.

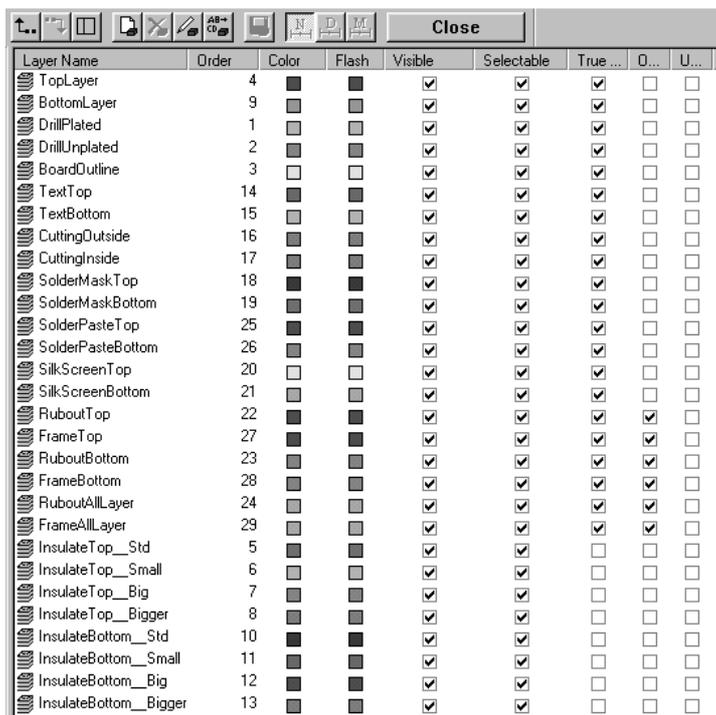
Use the **CURSOR KEYS** on the keyboard and the scrollbars to move the selection area.

Press **END** to center the selection area on the mouse position.

5.1.2 Working in the layer list window

The **VIEW** toolbar also includes the Layer button  which you can use to open the Layer list window. This list window shows all the existing layers with their current settings:

Figure 47:



| Layer Name | Order | Color | Flash | Visible | Selectable | True ... | O... | U... |
|-----------------------|-------|-------|-------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|--------------------------|
| TopLayer | 4 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BottomLayer | 9 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | 1 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillUnplated | 2 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BoardOutline | 3 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextTop | 14 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextBottom | 15 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| CuttingOutside | 16 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| CuttingInside | 17 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderMaskTop | 18 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderMaskBottom | 19 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderPasteTop | 25 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderPasteBottom | 26 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SilkScreenTop | 20 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SilkScreenBottom | 21 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| RuboutTop | 22 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| FrameTop | 27 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| RuboutBottom | 23 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| FrameBottom | 28 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| RuboutAllLayer | 24 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| FrameAllLayer | 29 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Std | 5 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Small | 6 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Big | 7 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Bigger | 8 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Std | 10 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Small | 11 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Big | 12 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Bigger | 13 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

The toolbar at the top of the list window also gives you access to these other functions:



CREATE NEW LAYER



DELETE SELECTED LAYER

 **EDIT SELECTED LAYER**

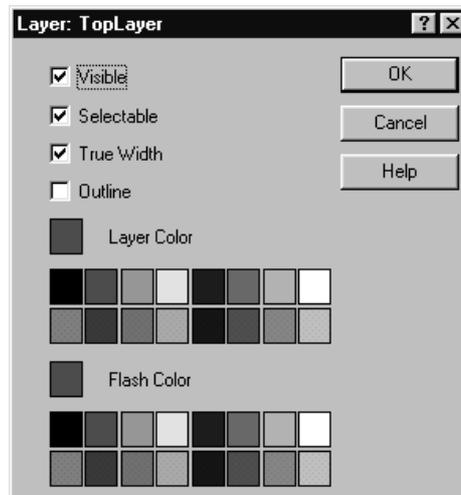
 **NAME LAYER**

 **SAVE**

You can change the setting for a layer by clicking on the relevant check box thereby either enabling it or disabling it .

Open the following dialog box by selecting a layer and clicking on .

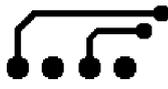
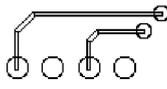
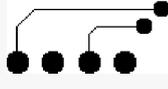
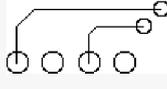
Figure 48:



Note: You can also call the LAYER dialog box by clicking on the line of the appropriate layer with the right mouse button.

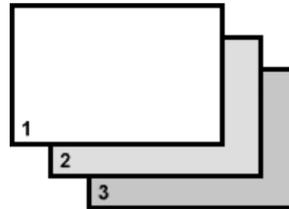
The following overview will clarify for you the effects of the **TRUE WIDTH** and **OUTLINE ILLUSTRATION** settings:

Figure 49:

| | True width <input type="checkbox"/> | Outline-illustration <input checked="" type="checkbox"/> |
|---|--|---|
| TRUE WIDTH <input checked="" type="checkbox"/> |  |  |
| TRUE WIDTH <input type="checkbox"/> |  |  |

The graphical objects in the individual layers are displayed in the graphic window as superimposed levels. The **ORDER COLUMN** in the **LAYER** list window indicates to you the level down from the top layer of the layer in the list.

Figure 50:



Remember that graphical objects in a lower level can be concealed by graphical objects in a higher level.

- › Click on the number of the layer which is to become the highest level in the **ORDER** column of the **LAYER** list window.

*The number 1 will be assigned to this layer in the **ORDER** column and the number of all other layers will be incremented by one, so that these layers are now one level deeper.*



Note: You specify the properties of the individual layers, but not the active layer, in the **LAYER list window.**

Select the active layer from the **LAYER** list box in the **DEFAULT** toolbar.

The active layer is the layer in which new graphical objects may be drawn, for example.

Make sure that you do not have an object selected when you change the active layer, as otherwise it will be moved to the newly selected layer.



Note: You will find a full description of the other functions in the list window in the *CircuitCAM* online help.

The new layers available in *CircuitCAM 5.0* - Insulate bottom (small, standard, big and bigger) - are layers for optimizing the routing process. A correspondingly narrow to extremely wide tool is assigned to the layers. For example during the insulate production phase, a tool as wide as possible is selected or requested automatically for large or wide routing surfaces. If the insulating strip to be cut is extremely small, a very small tool is selected. This selection is accomplished automatically by the software.

How to make a visual check on the imported data:

- › Click on  in the **VIEW** toolbar or press **HOME** to display all graphical objects within the workspace.
- › Click on the right mouse button and click now on **MEASURE** in the just opened menu.

- › Select a point in the graphic, from which the measurement should be started, e.g. the lower left corner of the circuit board frame.
- › Push the Mouse pointer to the endpoint of your measurement.
On the left in the status bar the removal is indicated as kartesic and polar coordinates.

Our sample circuit board has the dimensions 55.88 x 38.1mm.



Note: You may choose between millimeters, microns, inches and mils (1/1000 inch) as the units in *CircuitCAM*; the default is millimeter.

Change the unit using the **UNIT** list box on the **GRID+UNIT** toolbar or in **CONFIGURATION/GENERAL SETTING** under the **UNIT/GRID** tab.

- › Click **ESC** to finish the measurement.

Use the **VIEW** functions in *CircuitCAM* to zoom in on areas of the circuit board and to check the layers for cover.

5.2 Preparing production data for prototyping

In this section you will learn how to prepare the production data for the fabrication of a prototype circuit board. This involves the deletion of superfluous graphical objects, the addition of milling data and moving graphical objects onto other layers.

5.2.1 Deleting graphical objects outside the circuit board

The various layers often have registration marks outside the circuit board outline, as is the case with our sample circuit board. You might also think of other objects instead of the registration marks, such as labeling fields, company logos, etc.

However, only those objects inside the circuit board outline are of interest for the production of the prototype circuit board, and so all objects outside this can be deleted.

Objects must be marked before they can be deleted:

- You can mark individual objects by clicking in the immediate vicinity of the object to be marked with the left mouse key.
- To mark larger areas, hold the mouse key depressed and drag the mouse pointer diagonally across the surface to be marked.

Objects must first be selected before they can be deleted. *CircuitCAM* has a number of functions you can use to influence the way in which objects are selected to ensure that only the object desired is selected, and not other objects beneath, above or near the object. These functions may be found in the **SELECT** menu or on the **SELECT** toolbar:



SELECTING LAYER OBJECTS

Click on this button to select the entire active layer. The active layer is selected from the **LAYER** list box or by selecting a graphical object.



SELECTING APERTURES/TOOLS OBJECTS

Click on this button to select all graphical objects created with the active aperture or the active tool. The active aperture or tool list is selected from the **APERTURE/TOOL** list box or by selecting a graphical object.



SELECTING ALL OBJECTS

Click on this button to select all layers. (this function can also be activated by clicking on **ALL** in the main menu **SELECT**).

The following switches may be used to determine what kinds of graphical objects may or may not be selected:



LINE SELECTABLE

Lines may be selected if this button is enabled.



POLYGON SELECTABLE

Polygons may be selected if this button is enabled.



CIRCULAR SELECTABLE

Circles may be selected if this button is enabled.



RECTANGLE SELECTABLE

Rectangles may be selected if this button is enabled.



FLASH SELECTABLE

Pads (flashes) and drillings may be selected if this button is enabled.

All switches are enabled in the default setting (everything may be selected).

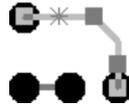


Note: You can change temporarily from the current selection mode to the Plus mode by holding down SHIFT and to the Minus mode by holding down CTRL. The original mode is re-enabled when the key is released.

You can remove the selection by clicking on  in the View toolbar.

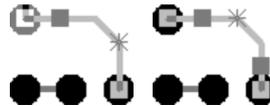
If just one object is selected, the handles for the selected object will be displayed:

Figure 51:



The handle closest to the cursor position when you clicked is displayed as a cross ✖ and serves as the anchor point. The selected object can therefore also have the following representations:

Figure 52:



The anchor point coordinates are displayed in the Status bar.



Note: Use the + or - keys to move the anchor point to the next handle on the selected object.

If a number of objects are selected, the selected objects are shown in a different color and the number of objects selected is shown in the **STATUS BAR**.



Note: Select PROPERTIES from the CONTEXT MENU (right mouse button) to view or modify the properties of a selected object.



Note: Select GENERAL ... from the CONFIGURATION menu to modify the methods for editing graphical objects to your requirements in the METHODS tab.

HOW TO DELETE OBJECTS OUTSIDE THE CIRCUIT BOARD:

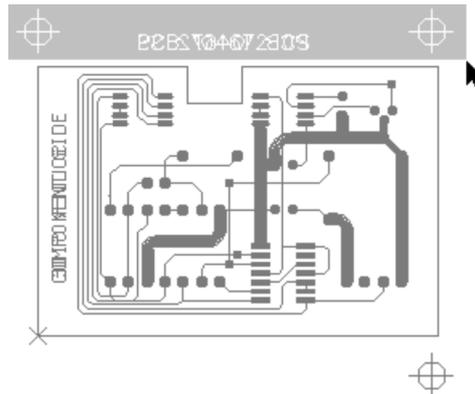
- › Make sure that all selection switches are enabled in the Select toolbar so that it is possible to mark all graphical objects:

Figure 53:



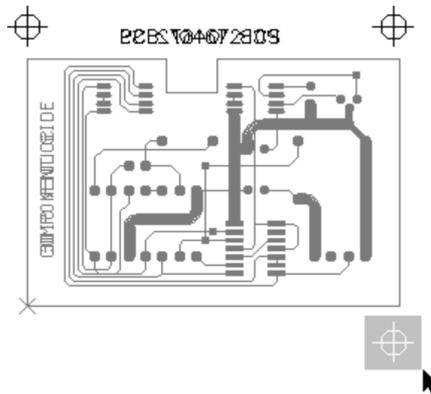
- › Click on  in the **SELECT** toolbar to enable the **NEW** selection mode.
- › Drag a box around the two upper registration marks using the left mouse button to select the text between them:

Figure 54:



- › Click on  in the **SELECT** toolbar to enable the **PLUS** selection mode.
- › Drag a box around the lower registration mark using the left mouse button to select them as well:

Figure 55:



All the objects outside the circuit board are now selected.

- › Click on  in the **DEFAULT** toolbar or press **DELETE** to delete the selected objects.

Tips&Tricks:

You can cancel an action by pressing **ESC**

Click on  in the Default toolbar or press **CTRL + Z** if you wish to undo one or more actions.

Click on  in the Default toolbar or press **CTRL + Y** if you wish to redo an action you have undone.

5.2.2 Moving the circuit board outline to a separate layer

In our "Tutor" sample circuit board, the circuit board outline is located on the solder side and on the component side and thus on an electrical layer.

It is sensible to place the circuit board outline on a separate mechanical layer to ensure that the circuit board outline is not milled along with the conductor tracks and pads, but that the circuit board outline is milled by itself.

In the operations described below, the circuit board outline on the component side ("TopLayer") is first selected and then moved to the "BoardOutline" layer. The circuit board outline on the solder side ("BottomLayer") will then be deleted.

How to select the circuit board outline on the component side:



Note: As the circuit board outline on our sample circuit board is located on two different layers, it is useful to assign the property not selectable to one of the two layers to make it easier to select the other layer.



Note: This procedure is always recommended when you wish to show a number of layers, but only wish to edit a specific layer.



Note: Where objects in different layers are located at the same position, *CircuitCAM* also allows you to toggle through these objects by clicking on them a number of times, i.e. a different object at this position will be selected with each new click. **Note:** during multiple click take care to the shape of the cursor to avoid modifications by mistake.

- › Reset the **SELECTABLE** property for the **BOTTOMLAYER** layer in the **LAYER** list window.

Figure 56:



| Layer Name | Order | Color | Flash | Visible | Selectable | True Width | Outline | Used |
|-------------|-------|-------|-------|-------------------------------------|-------------------------------------|-------------------------------------|--------------------------|--------------------------|
| TopLayer | 4 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BottomLayer | 9 | ■ | ■ | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | 1 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

This will make it easier for you subsequently to select the circuit board outline on the "**TOPLAYER**" layer in the graphic window.

- › Click on **CLOSE** to close the **LAYER** list window.
- › Click on the upper left-hand corner of the circuit board outline in the graphic window to select it.

Figure 57:

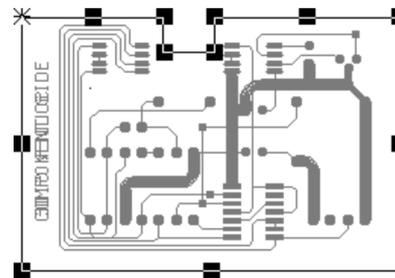


Note: You must always click on a handle of an object, i.e a corner point, to select it.

If you click on an object between two handles, you may select another object close by.

The circuit board outline on the component side (TopLayer) is selected and the solder side layer (BottomLayer) has the property **NOT SELECTABLE**:

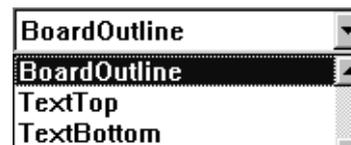
Figure 58:



How to move selected objects to a different layer:

- › Select **BOARDOUTLINE** from the **LAYER** list box on the **DEFAULT** toolbar to move the selected circuit board outline to this layer:

Figure 59:



- › Press **ESC** to deselect all graphic objects.

The circuit board outline on the solder side can now be deleted.

- › Set the **SELECTABLE** property for the **BOTTOMLAYER** layer in the **LAYER** list window.
- › Set the **NOT SELECTABLE** property for the **BOARDOUTLINE** layer in the **LAYER** list window.
- › Select the circuit board outline on the **BOTTOMLAYER** layer in the graphic window.
- › Press **DELETE** to delete the selected circuit board outline.
- › Press **HOME** to redraw the picture.

5.3 Direct Coordinate Entry

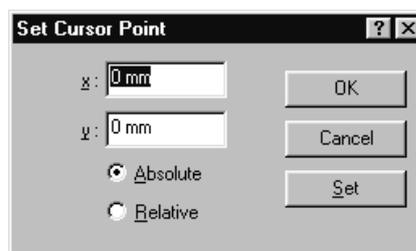
In *CircuitCAM* you can also activate mouse clicks by coordinate entry. Selecting a certain position as well as redrawing an object have been improved. A number of coordinates can be entered in sequence without terminating the function. This considerably accelerates generating front boards or RF layouts.

The following description is an example and is not part of the exercise. However, you can practice this function on any desired open file as follows:

- › Select a layer.
- › Select the **POLYGON** tool or **CLOSED LINE**.
- › Select **PROCESSING, SET CURSOR** in the main menu bar.

The following window appears:

Figure 60:



- › Make the following entries in the **SET CURSOR POINT** window:

x= 0 x= 10 x= 10 x= 0
y= 0 => **SET** y= 0 => **SET** y= 10 => **SET** y= 10 => **SET**

Confirm with **SET** after entering each coordinate (x and y) .

- › Then close the window by clicking **OK**.

5.3.1 Placing text objects on separate layers

It can also be useful to move text objects to a separate layer. As *CircuitCAM* does not distinguish between text objects and conductor tracks when these are located on the same layer, text objects would otherwise be isolated just like conductor tracks:

Text on electrical layer:

LPKF

Text on separate layer

LPKF

Since the procedure for moving text objects onto a separate layer is comparable with moving the circuit board outline to the **BOARDOUTLINE** layer, we will only describe the various operations in general terms here.

How to place text objects on a separate layer:

- › Select the text objects on the component side.
- › Select **TEXTTOP** from the **LAYER** list box on the **DEFAULT** toolbar to move the selected text objects to this layer:
- › Select the text objects on the solder side.
- › Select **TEXTBOTTOM** from the **LAYER** list box on the **DEFAULT** toolbar to move the selected text objects to this layer.
- › Press **ESC** to unselect all graphic objects



Note: You should also move the fastening holes from an electrical layer to a mechanical layer, e.g **DRILLUNPLATED** just like the circuit board outline and the text objects.



Save your work to protect against data loss.
Do this by selecting **SAVE AS...** from the **FILE** menu. Enter the name "TUTOR2" and press **OK**.
The document will be saved under the filename "TUTOR2" already entered.

5.3.2 Isolating with the default job

CircuitCAM offers default jobs which may be used for most circuit boards without modification of the isolation parameters to calculate the isolation channels. Later you will learn how you can customize the defaults for calculating the isolation channels individually.

The **FRONT TO END** toolbar groups together the functions required in the preparation of production data for passing on to *BoardMaster*, including the function for isolation using the default job:



ISOLATE COMPONENT SIDE AND SOLDER SIDE

How to calculate the isolation channels using the default settings:

- › Click on  in the **FRONT TO END** toolbar to calculate the isolation channels for the solder side and the component side in a single operation.

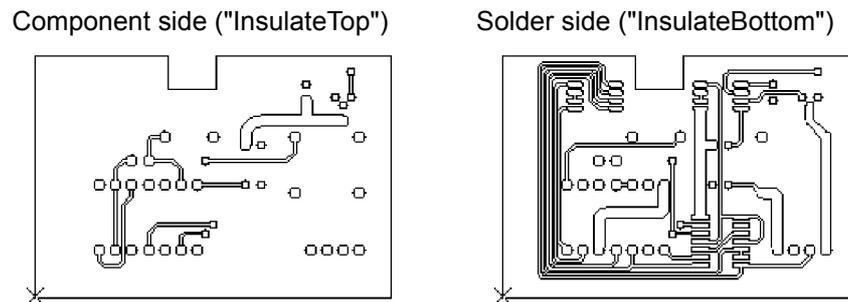


Note: Calculation of the isolation channels may take some time depending on the number of graphical objects and the parameters set such as the isolation grid. In addition, the processor performance and the amount of memory in your computer are important factors for the time requirement.

After calculation, the isolation channels are represented graphically. In this case, the cutting tracks are illustrated corresponding to the tools used in the colors assigned in each case (see „Color Assignment for Insulating Tools“ on page 82).

The isolation channels are illustrated graphically after calculation. The isolation channels on the component side are located in the **"INSULATETOP"** layer, those on the solder side on the **"INSULATEBOTTOM"** layer:

Figure 61:



Note: Save your work to protect against data loss. Do this by selecting **SAVE** from the **FILE** menu. The document will be saved under the filename **"TUTOR2"** already entered.

5.3.3 Preparing for Production Data Export

As of version 5.0 *CircuitCAM* also offers the possibility of checking the tool lists before exporting to ensure that an appropriate tool has been defined for all holes to be drilled and that different tools have not been defined for the same hole dimensions or routing areas. This offers the advantage that errors can be avoided for later tool assignments when the tool lists are made up. This checking and adapting work is required only when the standard lists and jobs have been changed. If you have not changed any of the standard lists, you can skip the next chapter "Checking the Tool Lists".

5.3.3.1 Checking The Tool Lists

Check the tool list as described below if required:

- › Click **CONFIGURATION**, then **FORMATS**, then **LPKF CIRCUIT BOARD PLOTTER TOOL LIST**.
- › Then double click **LPKF DRILLING TOOLS** in the list displayed.
The following window appears.

Figure 62: Tool list

| ConfigItem Name | Mode | Type | a | b | c | Used |
|-----------------------|------|------|--------|---|---|--------------------------|
| Spiral Drill 0.2 mm | ** | ○ | 0.2 mm | | | <input type="checkbox"/> |
| Spiral Drill 0.3 mm | ** | ○ | 0.3 mm | | | <input type="checkbox"/> |
| Spiral Drill 0.4 mm | ** | ○ | 0.4 mm | | | <input type="checkbox"/> |
| Spiral Drill 0.5 mm | ** | ○ | 0.5 mm | | | <input type="checkbox"/> |
| Spiral Drill 0.6 mm | ** | ○ | 0.6 mm | | | <input type="checkbox"/> |
| Spiral Drill 0.7 mm | ** | ○ | 0.7 mm | | | <input type="checkbox"/> |
| Spiral Drill 0.8 mm | ** | ○ | 0.8 mm | | | <input type="checkbox"/> |
| Spiral Drill 0.9 mm | ** | ○ | 0.9 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.0 mm | ** | ○ | 1 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.1 mm | ** | ○ | 1.1 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.2 mm | ** | ○ | 1.2 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.3 mm | ** | ○ | 1.3 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.4 mm | ** | ○ | 1.4 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.5 mm | ** | ○ | 1.5 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.6 mm | ** | ○ | 1.6 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.7 mm | ** | ○ | 1.7 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.8 mm | ** | ○ | 1.8 mm | | | <input type="checkbox"/> |
| Spiral Drill 1.9 mm | ** | ○ | 1.9 mm | | | <input type="checkbox"/> |
| Spiral Drill 2.0 mm | ** | ○ | 2 mm | | | <input type="checkbox"/> |
| Spiral Drill 2.1 mm | ** | ○ | 2.1 mm | | | <input type="checkbox"/> |
| Spiral Drill 2.2 mm | ** | ○ | 2.2 mm | | | <input type="checkbox"/> |
| Spiral Drill 2.3 mm | ** | ○ | 2.3 mm | | | <input type="checkbox"/> |
| Spiral Drill 2.4 mm | ** | ○ | 2.4 mm | | | <input type="checkbox"/> |
| Contour Router 2.0 mm | □ | ○ | 2 mm | → | ↩ | <input type="checkbox"/> |

- › It is only necessary to click the  button to display the list of drilling tools.

Here the drilling dimensions are displayed graphically in relation to one another on the right side of the tools (see figure below)

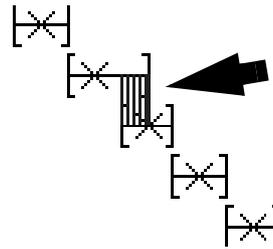
Figure 63: Drilling tool list

| ConfigItem Name | t | s | Drill gaps and overlaps | |
|-------------------------|---------|--------|-------------------------|-------------------------------------|
| Spiral Drill 0.2 mm | 0 mm | 0.2 mm | 0.25 mm | <input checked="" type="checkbox"/> |
| Spiral Drill 0.3 mm | 0.25 mm | 0.3 mm | 0.35 mm | <input type="checkbox"/> |
| Spiral Drill 0.4 mm | 0.35 mm | 0.4 mm | 0.45 mm | <input type="checkbox"/> |
| Spiral Drill 0.5 mm | 0.45 mm | 0.5 mm | 0.55 mm | <input type="checkbox"/> |
| Spiral Drill 0.6 mm | 0.55 mm | 0.6 mm | 0.65 mm | <input type="checkbox"/> |
| Spiral Drill 0.7 mm | 0.65 mm | 0.7 mm | 0.75 mm | <input type="checkbox"/> |
| Spiral Drill 0.8 mm | 0.75 mm | 0.8 mm | 0.85 mm | <input type="checkbox"/> |
| Spiral Drill 0.9 mm | 0.85 mm | 0.9 mm | 0.95 mm | <input type="checkbox"/> |
| Spiral Drill 1.0 mm | 0.95 mm | 1 mm | 1.05 mm | <input type="checkbox"/> |
| Spiral Drill 1.1 mm | 1.05 mm | 1.1 mm | 1.15 mm | <input type="checkbox"/> |
| Spiral Drill 1.2 mm | 1.15 mm | 1.2 mm | 1.25 mm | <input type="checkbox"/> |
| Spiral Drill 1.3 mm | 1.25 mm | 1.3 mm | 1.35 mm | <input type="checkbox"/> |
| Spiral Drill 1.4 mm | 1.35 mm | 1.4 mm | 1.45 mm | <input type="checkbox"/> |
| Spiral Drill 1.5 mm | 1.45 mm | 1.5 mm | 1.55 mm | <input type="checkbox"/> |
| Spiral Drill 1.6 mm | 1.55 mm | 1.6 mm | 1.65 mm | <input type="checkbox"/> |
| Spiral Drill 1.7 mm | 1.65 mm | 1.7 mm | 1.75 mm | <input type="checkbox"/> |
| Spiral Drill 1.8 mm | 1.75 mm | 1.8 mm | 1.85 mm | <input type="checkbox"/> |
| Spiral Drill 1.9 mm | 1.85 mm | 1.9 mm | 1.95 mm | <input type="checkbox"/> |
| Spiral Drill 2.0 mm | 1.95 mm | 2 mm | 2.05 mm | <input type="checkbox"/> |
| Spiral Drill 2.1 mm | 2.05 mm | 2.1 mm | 2.15 mm | <input type="checkbox"/> |
| Spiral Drill 2.2 mm | 2.15 mm | 2.2 mm | 2.25 mm | <input type="checkbox"/> |
| Spiral Drill 2.3 mm | 2.25 mm | 2.3 mm | 2.35 mm | <input type="checkbox"/> |
| Spiral Drill 2.4 mm | 2.35 mm | 2.4 mm | 2.45 mm | <input type="checkbox"/> |
| Contour Router 2.0 mm l | | | | <input type="checkbox"/> |

Gaps: Not available

If two or more drilling dimensions for various tools overlap, this is indicated graphically by accordingly shaded areas (see figure below).

Figure 64: Tool overlapping

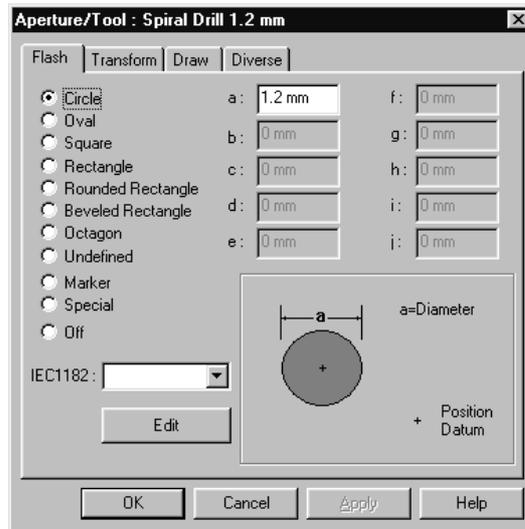


In this case the tool to be corrected should be changed so that the overlap is corrected or eliminated.

- › For this purpose open the settings window by clicking the appropriate tool in the list with the right mouse key.

The following illustration appears.

Figure 65: Flash

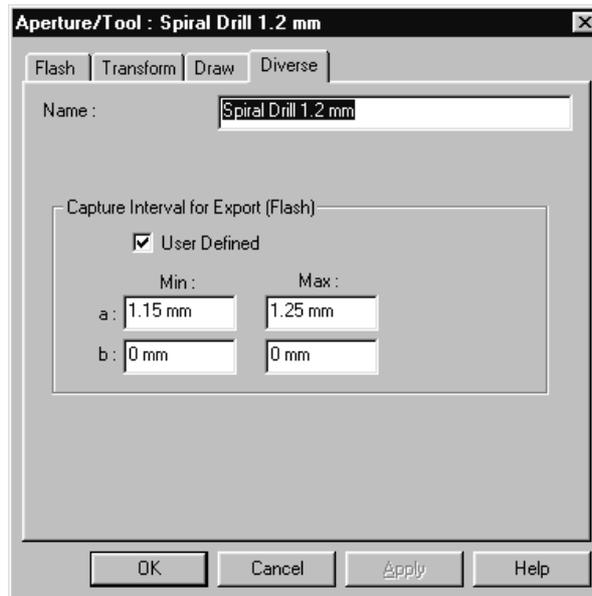


Here you can set the diameter of the tool as required.

› Click on **DIVERSE**.

The following illustration appears:

Figure 66: Diverse



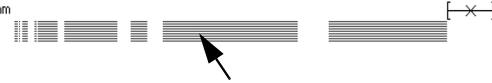
Here you can define the range of drill diameters to which the tool in question is to be assigned under **CAPTURE RANGE FOR EXPORT**. Any overlaps with other tools or gaps in the dimension settings, where no tool is assigned, can be corrected here by changing the values.

With the same procedure you can check or correct the milling tool. Here, however, it is necessary to click the **M** button after double clicking **LPKF MILLING TOOLS** (see beginning of chapter) to access the milling tool list. The catch areas for the milling tools are displayed accordingly. Overlaps are marked as described Figures 66 “Tool dimension overlapping” on Page 63, and gaps are marked according to the following illustration:

Figure 67: Maßfehler bei Fräswerkzeugen

| | 0.75 mm | 0.8 mm | 0.84 mm |
|---------------------------|---------|--------|---------|
| End Mill 0.8 mm (31 mil) | 0.75 mm | 0.8 mm | 0.84 mm |
| End Mill 1.0 mm (39 mil) | 0.95 mm | 1 mm | 1.05 mm |
| End Mill 2.0 mm (79 mil) | 1.9 mm | 2 mm | 2.1 mm |
| End Mill 3.0 mm (118 mil) | 2.85 mm | 3 mm | 3.15 mm |

Gaps:




Note: Gaps and overlaps are actually necessary and desirable for milling tools. For this reason, make all changes with extreme care!!

5.3.3.2 Export production data

Once the isolation channels have been calculated, the prepared production data can be exported to allow production of the prototype circuit board using the *BoardMaster* software and your prototyping system.

The LpkfMillDrill(LMD) binary data format was developed to optimize the link between *CircuitCAM* and *BoardMaster*. The production data from all the layers of a job can be captured to one file in the LMD format.

How to export production data in the LpkfMillDrill format:

- › Select **LPKF CIRCUIT BOARD PLOTTER** from the **EXPORT-LPKF** submenu of the **FILE** menu or click on  in the **FRONT TO END** toolbar.



Note: It is possible to select between 3 different jobs in the LPKF File Export Menu:

- LPKF Circuit Board Plotter:

This is the standard export job. As a matter of principle all applications can be handled with this job.

- LPKF Circuit Board Plotter Galv THP:

With this job the production phases for the *AutoContac* system are not output. This has the advantage, that superfluous error messages regarding the *AutoContac* system are avoided.

- LPKF Circuit Board Plotter No THP:

With this job no through-contacts of any type are planned. Some of the production phases are consolidated (e.g. all drilling phases ...). This facilitates more rational work (particularly with *ProtoMat 95/x*).

The file created will be saved to the working subdirectory ("Data") under the same name as the CircuitCAM file, but with the file extension ".LMD".

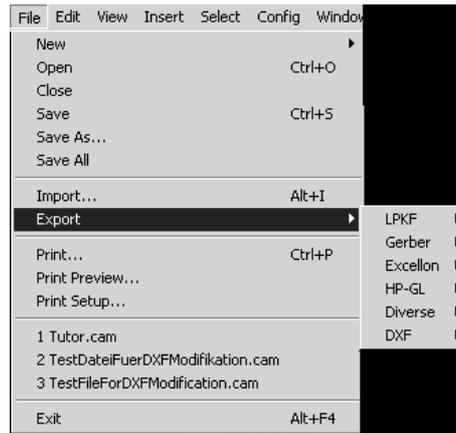
Following the export, the **REPORT WINDOW** displays information regarding the data exported, with any error messages.



Note: You can also export the production data in the most common CAD formats such as Gerber, Excellon and HP-GL (only PCB version) in addition to the LpkfMillDrill format.

- › Do this by selecting the appropriate line in the Export submenu of the File menu.

Figure 68:



To conclude the second exercise, you can now close the "TUTOR2" document by selecting **CLOSE** from the **FILE** menu.



Note: If the program *BoardMaster* should have started before the exporting the data, the exported lmd-file immediately is imported by BoardMaster and placed in the center of the material working area.

6.0 Exercise 3

In this chapter you learn, how to use the contour generator and the inserting of break-out-tabs. Furthermore the generating of surfaces by removing all the residual copper is shown.

6.1 Using contour routing

Should you wish to have the circuit board milled out by the circuit board plotter rather than having it cut to size by a circuit board guillotine you can use contour routing to calculate the milling contour.

Use the *CircuitCAM* file "**TUTOR1.CAM**" again for the following exercise:

- › Select **OPEN** from the **FILE** menu.
- › Select the "**TUTOR1**" file and click on **OPEN**.



Note: Once you have opened the file "**TUTOR1.CAM**" in **CircuitCAM**, you should save this document to a different file name to ensure that the original data remains available to you for further exercises.

- › Select **SAVE AS** from the **FILE** menu.
- › Enter "**TUTOR3**" as the filename and click on **SAVE**.

You may select either a selection or a layer as the source for the milling contour. A selection will be used as a source in this exercise, as the circuit board outline for this circuit board is located on the "**TOPLAYER**" and "**BOTTOMLAYER**" layers together with the conductor tracks and pad.

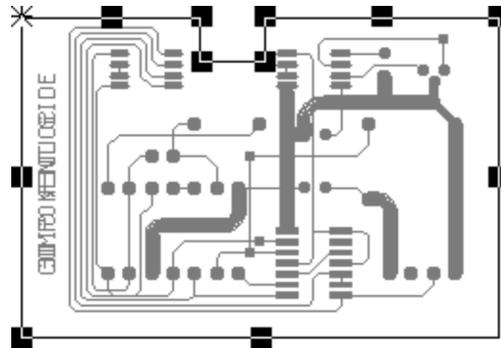


Note: If you are having the circuit board milled out by the circuit board plotter, it is no longer necessary to place the circuit board outline on a separate layer, as was the case in exercise 2.

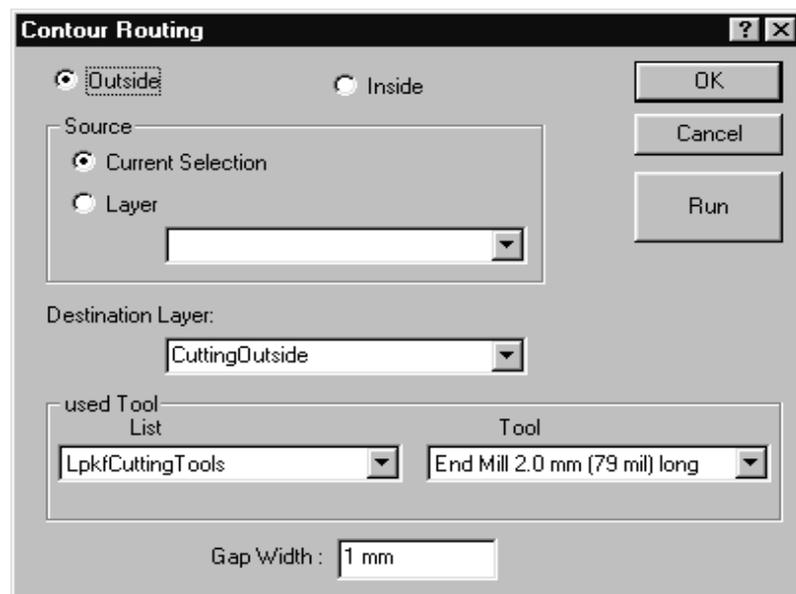
How to create a milling contour:

- › Select the circuit board outline in the **TOPLAYER** layer.

Figure 69:



- › Click on  in the **FRONT TO END** toolbar to open the **CONTOUR ROUTING** dialog box:

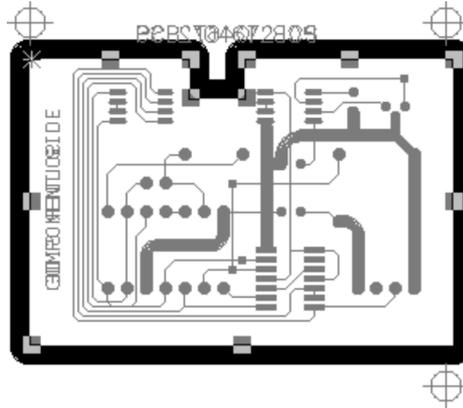


Note: If the circuit board outline for the circuit board is already placed on a separate layer, e.g. on the "BOARDOUTLINE" layer, you may use this layer as the source for calculating the milling contour by clicking on the **LAYER** radio button and selecting **BOARDOUTLINE** from the **LAYER** list box.

- › Click on **RUN** to generate the milling contour.

Once calculation is complete, the milling contour will be displayed in the graphic window:

Figure 70:



You can delete the circuit board outline from the **"TOPLAYER"** and **"BOTTOMLAYER"** since the milling contour has now been generated on the **"CUTTINGOUTSIDE"** layer:

- › Press **DELETE** to delete the circuit board outline, which is still selected, from the **"TOPLAYER"** layer.
- › Select the circuit board outline in the **"BOTTOMLAYER"** layer.
- › Press **DELETE** to delete this circuit board outline.



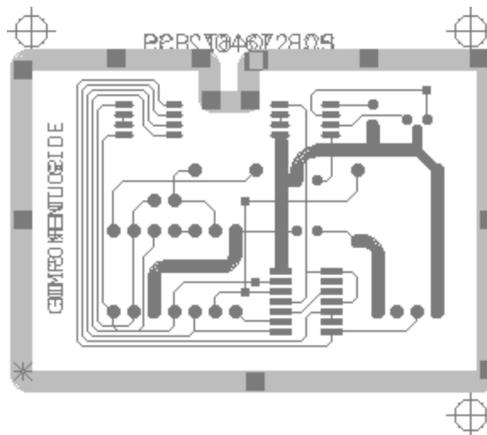
Caution! The circuit board would be completely milled out of the board material with the current contour and could be thrown about by the rotating milling head.

Caution! It may therefore be desirable to leave the circuit board connected to the board material by at least two break-out-tabs to prevent personal injury and damage to the circuit board plotter. The gap width is defined in the Contour Generator Dialog Box.

How to insert the break-out-tabs in the milling contour:

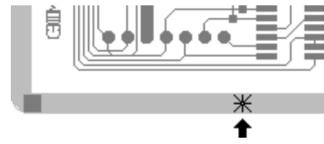
- › Select the contour generated on the **"CUTTINGOUTSIDE"** layer

Figure 71:



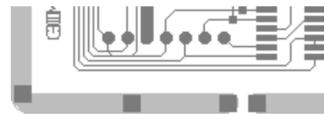
- › Use the **+** and **-** keys to move the anchor point* from handle to handle on the contour line.

Figure 72:



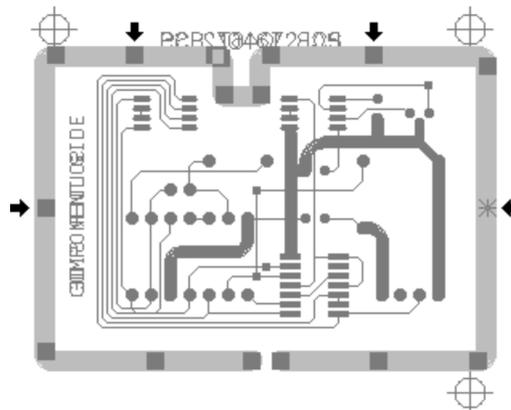
- › Press **CTRL + G** or click on  in the **FRONT TO END** toolbar to insert a tab at the anchor point position:

Figure 73:



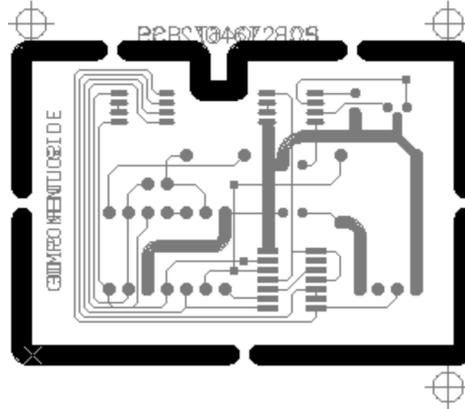
- › Repeat steps 2 and 3 to insert break-out-tabs at subsequent positions:

Figure 74:



You have now generated a milling contour in which the circuit board is connected to the board material by five break-out-tabs:

Figure 75:



Note: To save the current position of the exercise, select **SAVE** from the **FILE** menu. The document will be saved under the file name "TUTOR3" already entered.

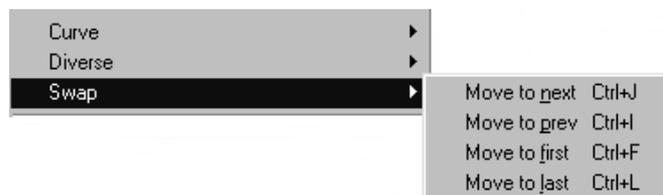
6.1.1 Modifiable Milling Direction and Sequence

In order to process interior and exterior contours optimally, you can define the milling sequence as well as milling direction. For this purpose, it is possible to consider your special requirements for comprehensive and/or irregular contours with numerous breakout tabs for various applications. To use this function proceed as follows:

- › Select a milling channel by clicking the left mouse button.
- › Select **EDIT, SWAP ...** in the main menu bar.

The following menu appears:

Figure 76: Swap



Here you can select whether the milling path is to be cut first or last.

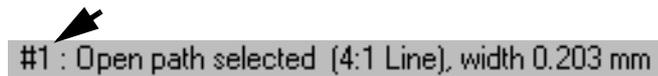
Move to first: This conductor is cut first.

Move to last: This conductor is cut last.

Move to next, next to previous: Moves the cutting path forward or back one step in the sequence.

At the bottom left of the screen, you can check the sequence in which the conductors will be cut (see figure below).

Figure 77:



#1 : Open path selected (4:1 Line), width 0.203 mm

If the lines are to be checked, each individual line must be clicked.

In order to check the cutting direction of a line, click on it with the left mouse key. The properties of the line are shown in the status display line (see figure 77). While the line is selected, you can also determine the direction by means of the + or - keys on the keyboard. If the marking moves along with the + key, then this is the actual cutting direction.

The optimum cutting direction for a cutting path depends on which side of the cut is to be the “good” side. With the function **EDIT - CURVE - REVERSE** you can change the cutting direction. When the cutting path is selected and the <+> key pressed, the anchor point moves in the cutting direction. However, observe the circuit board side. Data for the bottom layer is mirrored in *BoardMaster*, resulting in a reversal of the cutting direction as well.

6.2 Create rubout area

A rubout area is a zone in which not only the isolation channels around conductor tracks and pads are milled, but also all the residual copper between the conductor tracks and the pads are removed from the circuit board. This operation can be sensible for technical reasons (e.g. EMV) as no electrical potential is assigned to this residual copper.

You will find the functions for creating a rubout area in the **RUBOUT AREA** submenu of the **INSERT** and in the **FRONT TO END** toolbar:



RUBOUT AREA ALL LAYERS

creates a rectangular Rubout area on the top and bottom side of the circuit board.

You can define the range of the Rubout area using the following drawing functions which can be found in the **GRAPHICS** toolbar:



POLYGON

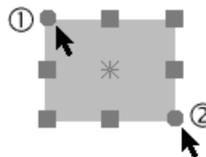
is defined by any number of handles set by mouse click. The polygon is finished by pressing **ESC** once the last handle has been set. The function is concluded by pressing **ESC** once again.

Figure 78:



 **RECTANGLE**
(default setting for Rubout area)
defined by two mouse clicks:

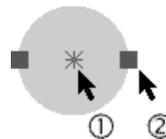
Figure 79:



The “rectangle” function can be terminated by pressing **ESC**.

 **CIRCLE**
defined from the centre point by two mouse clicks:

Figure 80:

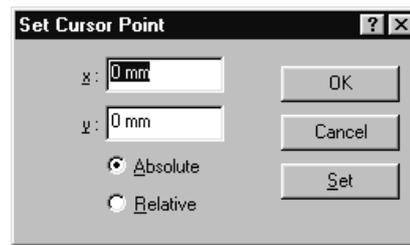


The **CIRCLE** function can be terminated by pressing the **ESC** key or by entering a circle diameter manually.



Note: Instead of clicking with the mouse, you can enter the cursor position numerically from the keyboard by selecting **SET CURSOR...** from the **EDIT** menu. The **SET CURSOR** dialog box will be displayed and you can enter the new cursor position in the text boxes:

Figure 81:



- › Click on **SET** to accept the values.

The Set cursor function corresponds to a mouse click at the position entered and can thus be used for drawing graphical objects precisely.

Three predefined layers are available for drawing Rubout areas, "**RUBOUTBOTTOM**" for the bottom side of the circuit board, "**RUBOUTTOP**" for the top face of the circuit board and **RUBOUT ALL LAYERS** for both sides of the circuit board (standard).

You will be creating the Rubout area on both sides of the circuit board around the 14 pin SMD IC in the following exercise.

How to generate a rubout area:

- › Click on  in the **FRONT TO END** toolbar to create a rubout area on both sides of the circuit board.

*The **RUBOUT ALL LAYER** layer is now active, the default for the rubout area shape is **RECTANGLE**.*

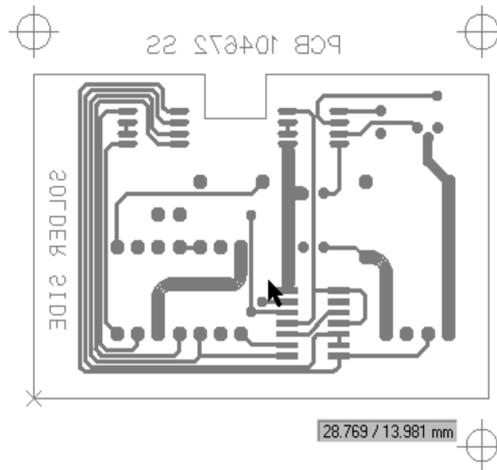


Note: You can select a different shape for the rubout area by clicking on  (**POLYGON**) or  (**CIRCLE**) on the **GRAPHICS** toolbar.

Use the view functions to position the rubout area precisely.

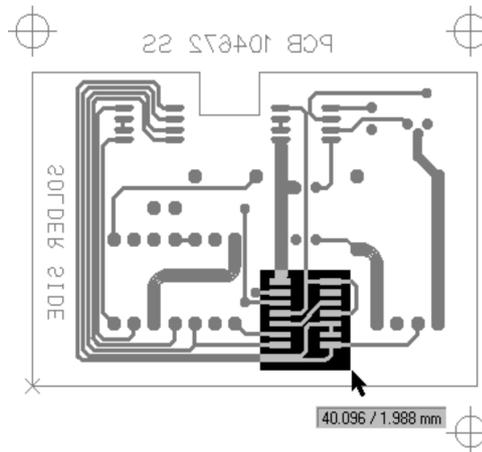
- › Click above and to the left of the 14 pin SMD IC to set the starting point of the rectangle:

Figure 82:



- › Move the cursor down and to the right to draw the rectangle around the SMD IC:

Figure 83:



- › Click again to set the end point of the rectangle.
- › Double Click on **ESC** to toggle from graphic mode back to selection mode.

The rubout area is now defined.



Note: If a Rubout area should be defined only on one Circuit board side, choose after click on  the corresponding layer (RuboutTop / RuboutBottom) from the List box LAYER in the standard tool bar.



Note: Naturally you can also define a number of rubout areas of different shapes on one circuit board.



Note: Since a rubout area is a graphical object just like any other, you can also select rubout areas resize, reshape, move and delete them.



Note: The residual copper area to be removed does not need to be defined precisely, but can simply be drawn over the conductor tracks and pads just like in the exercise. All the conductor tracks and pads located in this area are taken into consideration when calculating the isolation channels.



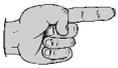
Note: Any text objects over which the rubout area is positioned must be located on an electrical layer, so that they are also taken into consideration in the calculation of the isolation channels for the rubout area.



Note: To save the current position of the exercise, select **SAVE** from the **FILE** menu. The document will be saved under the filename "TUTOR3" already entered.



Note: At the end of the exercise 3 you can close the document "TUTOR3" by clicking on **CLOSE** in the menu **FILE**.



Note: You are now familiar with the basis functions of *CircuitCAM* for preparing production data for circuit board prototyping.



Note: A list of the predefined layers and insulating jobs in *CircuitCAM* is given in the appendix; moreover we show you the parameters required for defining an output job and how you can match a translation file to your circuit board CAD program, to obtain your prototype circuit board quickly.



Note: Also use the sample circuit boards "TUTOR" and "EAGLETUTOR" to familiarize yourself with other *CircuitCAM* functions.



Note: Detailed information on every function is given in the *CircuitCAM* online help and in the reference manual.

7.0 Exercise 4

7.1 Customize isolation job

CircuitCAM incorporates a number of preconfigured jobs which you can use for calculating the isolation channels (see appendix).

Thus, for instance, in the **FRONT-TO-END** toolbar, the jobs “**INSULATEDFAULTTOP...**” and “**INSULATEDFAULTBOTTOM...**” are assigned by default to the button with the functions **INSULATE ALL**

LAYERS .

CircuitCAM has the **INSULATE** dialog box in the **EDIT** menu for customizing the preconfigured jobs and creating new jobs.

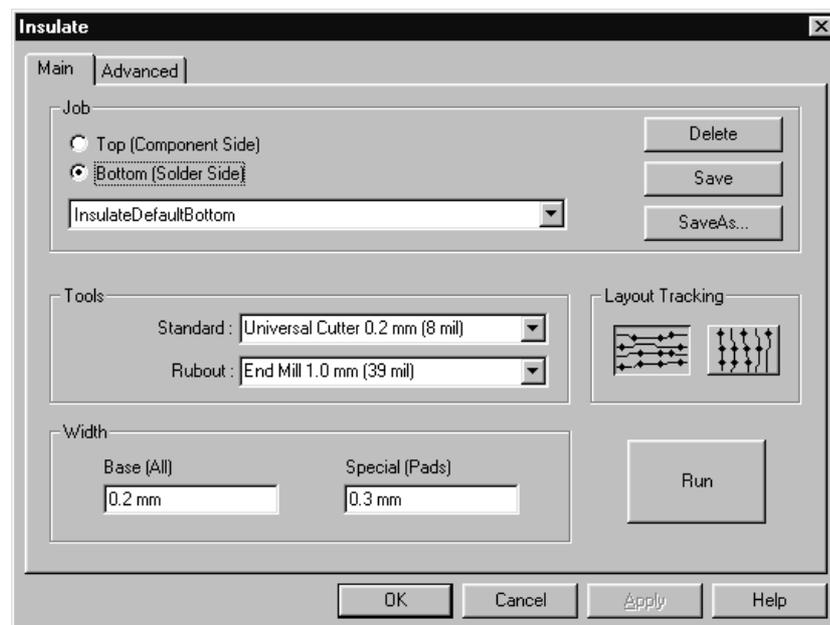
Open the **ISOLATE** dialog box by selecting **INSULATE...** from the **EDIT** menu. The dialog box is divided up into two pages to which you can jump by clicking on the tab.

The illustration in figure 84 shows the configuration for the “**INSULATEDFAULTBOTTOM**” default job.

7.1.1 CircuitCAM - Lite

Settings on the Start tab

Figure 84:



JOB:

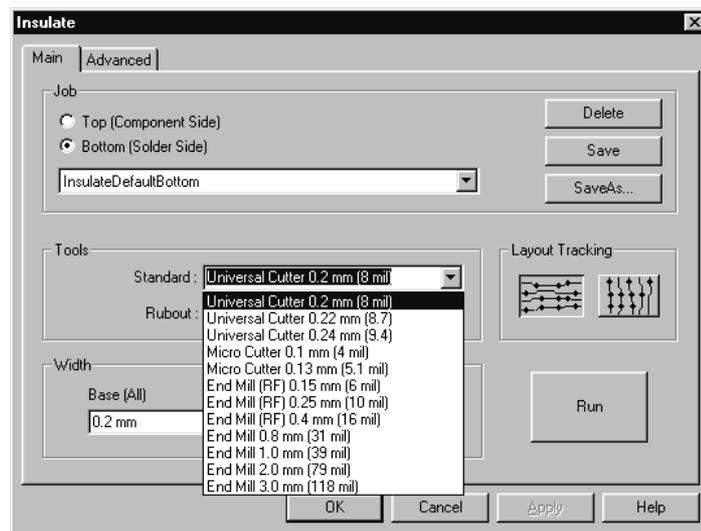
Select the side of the circuit board for which the job is to be executed. The corresponding layer, “**TOP**” or “**BOTTOM**”, will then be used automatically.

TOOLS:

Here you should define the tools that will be used for milling the circuit board. The Lite version has two tools available for removing the copper areas most efficiently and quickly.

Click on  to drop down one of the list boxes from which you can select the tool required:

Figure 85:



ISOLATION WIDTH:

Specify the width for the isolation around all electrical objects (conductor tracks and pads) in the Default text box.

You can enter the width of the isolation around the pads that have been created by flash apertures in the Special text box.

PREFERRED DIRECTION:

Clicking on one of these two buttons allows you to define the preferred direction of the conductor tracks on the side of the pcb selected. The milling tracks within the Rubout areas are then calculated in accordance with the preferred direction.

DELETE:

Clicking on the **DELETE** button deletes the highlighted Isolation Job.

SAVE:

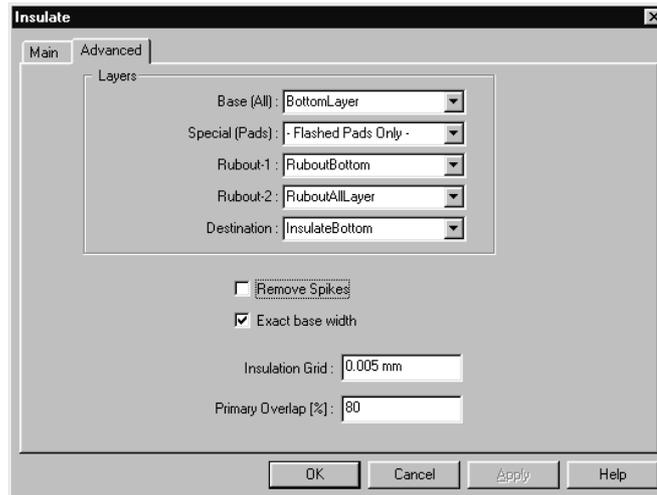
Clicking on the **SAVE** button saves the modified Isolation Job. The modification will also be saved to the underlying template file (*.CAT).

SAVE AS:

Clicking on the Save As button allows you to save the modified Isolation Job under a new name. The old Isolation Job remains unchanged. The modification will also be saved to the underlying template file (*.CAT).

Settings on the Advanced tab

Figure 86:



LAYERS:

You specify which layers are to be used in calculating the isolation channels and to which layer the result will be saved in the Layers Group Box.

The layer RuboutTop or RuboutBottom and RuboutAll layers serve as the source for definition of the areas in which all residual copper is to be removed.

REMOVE SPIKES:

Checking this check box removes all the small copper spikes or burrs up to the size of the diameter of the standard tool. **Note that the calculation and milling time will be longer.**

EXACT BASE WIDTH:

Checking this checkbox causes the isolation width specified in the **START** tab to be complied with precisely. Otherwise, the maximum achievable width for the same number of milling passes will be generated.

ISOLATION GRID:

This value specifies the precision to which the isolation tracks are calculated. A smaller value means higher precision which means a longer calculation time. A value of 0.005 mm should be selected for structures with a conductor track width and minimum distance of 0.2 mm. Higher values (0.05 mm) can falsify the result.



Note: Since the isolation procedure is grid-based, there is a linear relationship between the processing time and the number of grid points to be computed. The finer, thus, the isolation grid, the longer the calculation takes.

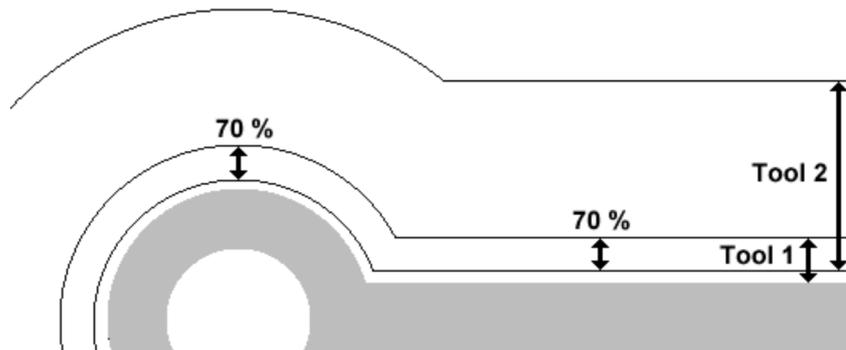
Recommended isolation grid: 0.005 mm

PRIMARY OVERLAP:

It is useful to use different milling tools if larger areas are to be milled efficiently. The conductor tracks and the pads are first isolated with a small milling tool to ensure high precision. The remaining isolation area can then be milled with a larger milling tool.

Specify the value by which the milling path of the next larger tool should overlap the milling path of the smaller tool in the **PRIMARY OVERLAP** text box. If this value is too small, the standard tool will have to be used unnecessarily at many points (at the corners) resulting in a longer milling time. If the value is too great, the rubout tool will get too close to the conductor tracks.

Figure 87:



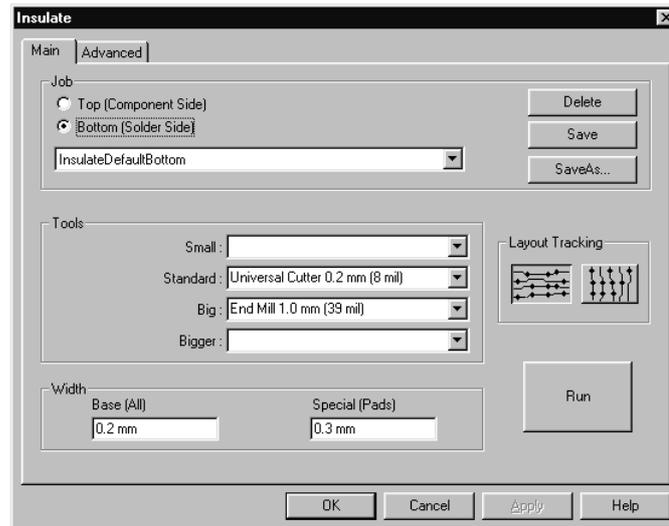
Note: The next larger milling path should overlap the smaller milling path by between 50% and 90%.

7.1.2 CircuitCAM - PCB

All of the *CircuitCAM* - Lite Version features are also available in the PCB version. However, this version has additional features as described below.

Setting in Main Index

Figure 88:

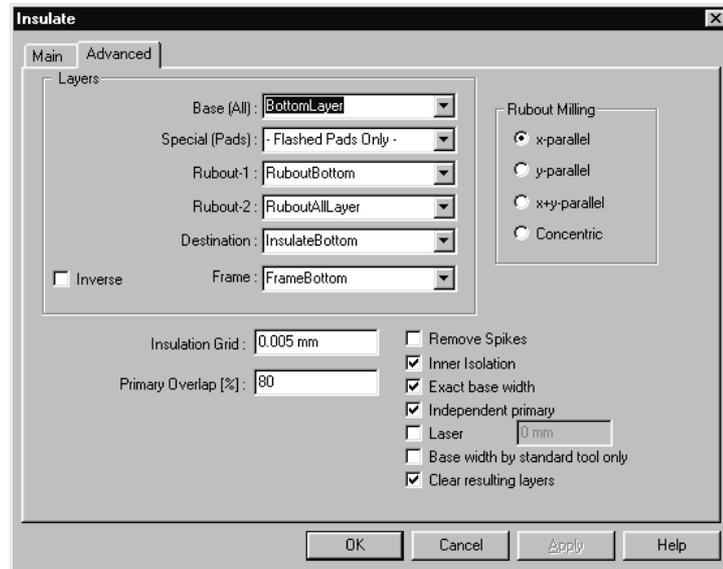


TOOLS:

In addition to the two tools in the Lite Version, the PCB Version has two additional tools: The **SMALL** tool for use on points on the circuit board which cannot be reached with the standard tool due to its diameter. The **LARGER** tool which is used as an additional rubout tool for removing larger copper areas.

Settings in Advance Index

Figure 89:



Note: The Source layer “Frame” and the checkbox “Inverse” as well as the checkbox “Clear Resulting Layers” are new in *CircuitCAM 5.0*. In areas on circuit boards which high voltage or high frequencies, it may be necessary to process the various areas with different insulation methods.

When an area is defined with the aid of the graphic function “Rectangle” or “Polygon” on the layer entered here and the checkbox INVERSE is switched off, the insulating jog is accomplished only in this area. If the checkbox “Inverse” is switched on, the insulating job is performed only outside of this defined area.

In each case, two insulating passes are required.



Caution! During the second pass, always switch off the checkbox “Clear resulting layers” because otherwise the result of the previous insulating path is lost!

RUBOUT MILLING:

In addition to the X or Y parallel rubout paths, concentric or X and Y parallel paths can be generated.

INNER INSULATION:

When this checkbox is switched off, only the outer isolation channels are created around the conductor, however, not the inner isolation channels on a closed conductor line.

INDEPENDENT PRIMARY:

When this checkbox is switched off, no primary cutting channel is generated between two elements closer together than the diameter of the standard tool (danger of short circuit). If the checkbox is switched on, a milling channel is always generated (danger of reducing the width of a conductor).

LASER

Only used in connection in LPKF *Protolaser*. When switched on, this checkbox prevents multiple superimposition of milling channels. This is important when a laser is used for “Insulation milling”. When a circuit board plotter is used, this checkbox must be switched off.

BASE WIDTH BY STANDARD TOOL ONLY

When this box is checked, only the standard tool is used for the base insulation width entered on the index card “Main”. This could be very important in the RF engineering sector. For standard circuit boards, the checkbox should be switched off to optimize the number of milling channels (big tool) and therefore the milling time.

CLEAR RESULTING LAYERS

When this box is checked, the milling paths for the resulting layers from insulating operations performed previously are deleted (default setting). If the checkbox is switched off, the new result is added to the previous.



Note: In the case of multiple insulating passes with the checkbox switched off, an unnecessary number of milling paths are generated. This can drastically increase the milling time.



Note: In the FRONT-TO-END toolbar, the button with the function **INSULATE ALL LAYER** is assigned the jobs “**INSULATE DEFAULT TOP**” and “**INSULATE DEFAULT BOTTOM**” as a default setting. Adapt these jobs to your requirements to obtain quick access to the desired insulating functions with the toolbar.

You can adapt the default job to your requirements as follows:

- › Select the job “**INSULATE DEFAULT BOTTOM**” on the **MAIN INDEX** card in the dialog box **INSULATE**.



Note: You can store the standard job under another name as a precautionary measure so that it will still be available later. Click **SAVE AS...** to store the default job under a different name. Enter the new name in the text box and click **SAVE**. After you have stored the job under a new name, select the job “**INSULATE DEFAULT BOTTOM**” again in the job index.

- › Change the parameters in the **MAIN** and **ADVANCED** index cards to meet your requirements.
- › Click **SAVE** on the main register card to **SAVE** the job “**INSULATE DEFAULT BOTTOM**” with the new parameters.
- › Repeat this step for the job “**INSULATE DEFAULT TOP**”.

If you now click  in the **FRONT-TO-END** toolbar, the default jobs with the parameters with you have changed are executed immediately.

You can create a new job as follows:

- › Select the job which you want to use as the basis for creating the new job on the **MAIN** register card in the dialog box **INSULATE**
- › Change the parameters in the **MAIN** and **ADVANCED** index cards to meet your requirements (see “CircuitCAM - Lite” on Page 79 and “CircuitCAM -PCB” on Page 83).
- › Click **SAVE AS...** on the **MAIN** index card to store the modified job under a new name.
- › Enter the new name in the text box and click **OK**.



Note: You can delete a job by selecting it on the JOB index card and clicking DELETE.

7.1.3 Insulating Frames (PCBs only)

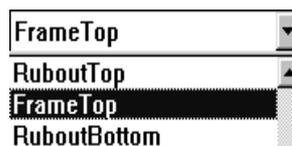
If you want to create circuit boards to be used in the high-voltage range or high-frequency range, you can increase or reduce the insulation intervals separately in certain areas with this function. Moreover, this function can be helpful for optimizing the milling times and tool costs.

It is possible to define a frame around an area on the circuit board layer so that this area can be assigned different insulating parameters than the rest of the circuit board side. The tools to be used can be redefined as well as the conductors for this framed area.

The following description gives an example. File used is the same as that used in the exercises, and should be opened beforehand. When this has been accomplished, proceed as follows to familiarize yourself with this function:

- › Select the layer *FrameTop* in the list box *Tutor1* in the standard tool list as the active layer (see figure below).

Figure 90:



Switch the *BottomLayer* to invisibly described below.

- › For this purpose select entry **VIEW, LAYER**, or click the button



("Layer").

- › Switch the **VISIBLE** box for the *BottomLayer* by clicking with the mouse.
- › Click on **CLOSE**.

Only the top layer is now visible on the screen.

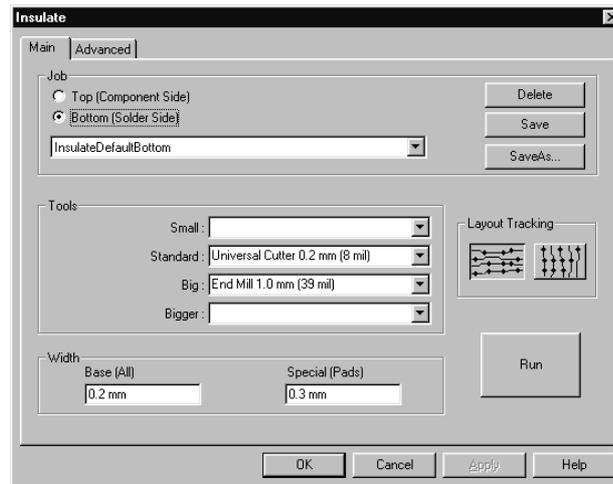
- › Select the **CLOSED PATH** menu **INSERT** or click the button 

"Closed line".

- › Mark the desired area with the mouse and press the **ESC** key to switch off the tool.
- › Press the **HOME** key on your keyboard to refresh the display.
- › Select the menu **EDIT, INSULATE**.

The following window appears:

Figure 91: Insulating - Start



Settings on Main Index Card

- › Select the radio button **COMPONENT SIDE (TOP)**.

Explanation on Windows and Functions:

JOB:

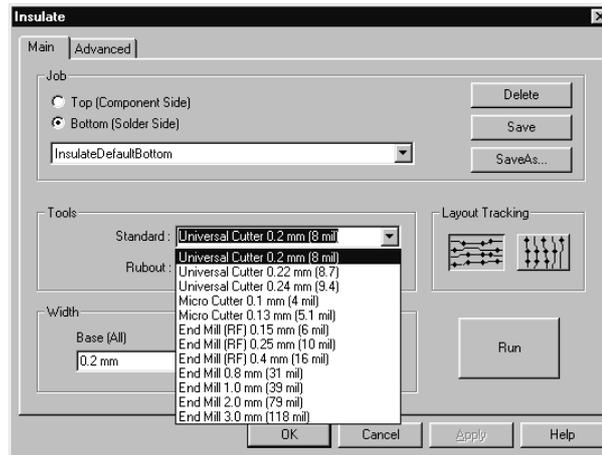
Select the side of the circuit board for which the job is to be performed. The appropriate layer, "**TOP**" or "**BOTTOM**" is then used automatically.

TOOLS:

Here you can define the tools to be used later for machining the circuit board. In the Lite Version, two tools are available to remove the copper area optimally and quickly.

Click  to open the list box from which you can select the desired tool.

Figure 92:



INSULATION WIDTH:

Indicate the width of the interrelation around all electrical objects (conductors and pads) in the text box **BASE**. In the text box **SPECIAL** you can specify the width of the insulation around the pads which are produced with flash apertures.

LAYOUT TRACKING:

You can define the layout tracking for the conductors on the selected circuit board layer by clicking one of the two buttons. The cutting paths within the rubout areas are then calculated according to the layout tracking.

DELETE:

You can delete the marked insulating job by clicking the **DELETE** button.

SAVE:

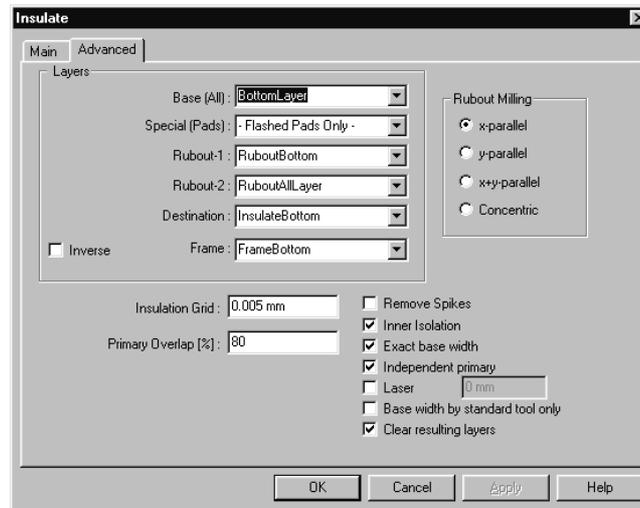
You can save the modified insulating job by clicking the **SAVE** button. Changes are also stored in the basic template file (*.CAT).

SAVE AS:

You can store the modified insulating job under a new name by clicking the **SAVE AS...** button. The old insulation job then remains unchanged. The change is also stored in the basic template file (*.CAT).

Settings on Advanced Index Card:

Figure 93: Insulation



Explanations on Windows and Functions:

LAYERS:

In the group box layers, you can specify which layer is to be used as the source for calculation of the insulation channels and on which layer the result is to be executed.

REMOVE SPIKES:

When this box is checked, all small copper spikes or hairs are removed down to the size of the diameter of the standard tool. **Caution! Longer calculation and milling time!**

EXACT BASE WIDTH:

When this box is checked, the insulation width set in the **MAIN** index is maintained precisely. Otherwise, a maximum achievable width is generated for the same number of milling channels.

INSULATION GRID:

This value specifies the accuracy with which the insulation pads are to be calculated. A smaller value results in a higher accuracy and longer calculation time. For structures with a conductor width of 0,2 mm and minimum interval, a value of 0.005 mm should be selected. Higher values (0.05 mm) can make the result erroneous.



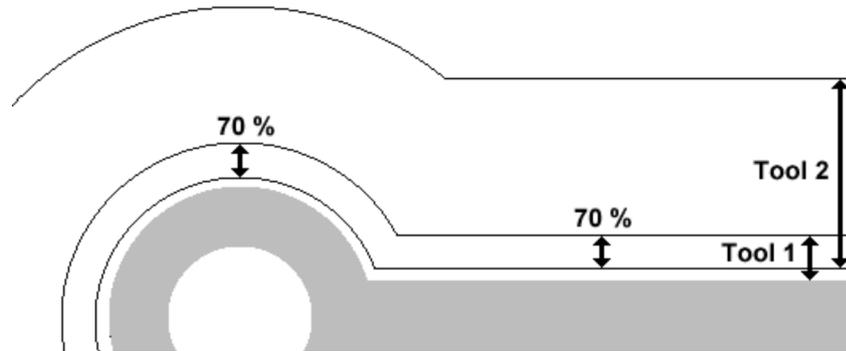
Note: Since the insulating operation is grid-based, the calculation time is linear to the number of grid points to be calculated. Therefore, the finer the insulation grid, the longer the calculation operation requires. Recommended insulation grid: 0.005mm (For extremely slow computers: 0.01mm)

PRIMARY OVERLAP:

In order to efficiently machine large surfaces, it is practical to use different milling tools. For high accuracy, first the conductors and pads are insulated with a small milling tool. The other insulation surfaces are then cut out with a larger milling tool.

Enter the value in the **PRIMARY OVERLAP** text box by which the milling pad of the next larger tool is to overlap the milling channel of the smaller tool. If the value is too small, unnecessary use of the standard tool is required in many points (in corners) resulting in a longer milling time. If the value is too large, the rubout tool comes too close to the conductors.

Figure 94:



Note: The next larger milling channel should overlap the smaller milling channel by 50 % to 90 %.

7.1.4 Color Assignment for Insulating Tools

In order to obtain a better overview of the tools used for insulating, it is possible to associate various colors with the tools. This possibility is particularly interesting because 4 different insulating tools (small, standard, big, bigger) can be used. For insulating widths, there are (standard, small, big, bigger) to use the tools optimally (no small insulating tool wide insulating path,...). To differentiate these four tools from one another, a layer can be created for each tool. These layers can be assigned different colors.

› Click the  button in the View toolbar.

The layer list appears (excerpts).

Figure 95: Layer List

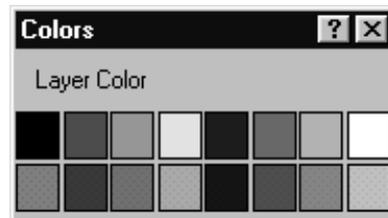


First possibility:

- › Click the color box for a layer in which the color is to be changed with the left mouse key.

The following window appears:

Figure 96:



- › Click the desired color with the left mouse key.

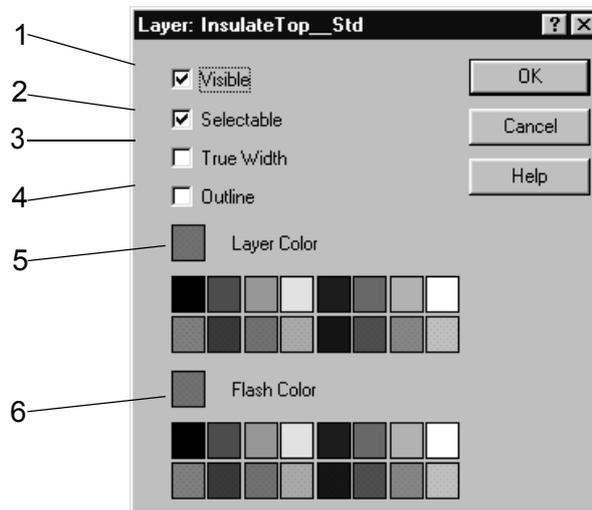
The corresponding color is assigned and the window closes automatically.

Second possibility:

- › Now move the mouse to the text designation of the layer, whose color you want to change and click once with the right mouse key.

The following window appears:

Figure 97: Color definition for Isolation tools



1- Visible: The selected layer can be switched visible or invisible.

2- Selectable: The layer can be protected against unintentional modification later by removing the check mark.

3- True width: When this box is activated (checked), the true width of the conductor is visible. Otherwise, only thin lines are displayed which do not correspond to the actual conductor image.

4- Outline: If this box is activated, only the outlines of the conductors to be processed with the tool are displayed.

5- Layer color: Display color for all elements except flashed pads.

6- Flash color: Separate display color for flashed pads.

In this window you can change the true width or outline display in addition to the colors.

This window can be closed by clicking **OK**.

After selecting a color, as described in the first possibility or clicking **OK** as with possibility two, you return to the layer List View level.

- › Click **CLOSE**.

8.0 Exercise 5

In this section you will learn how to create user-specific template files for *CircuitCAM* and how to correct production data in which the layers are coincident.

8.1 Creating a CircuitCAM template file

CircuitCAM offers the possibility of creating a template file with user-specific basic settings similar to a format template in a word processing program for a telefax, letter or report.

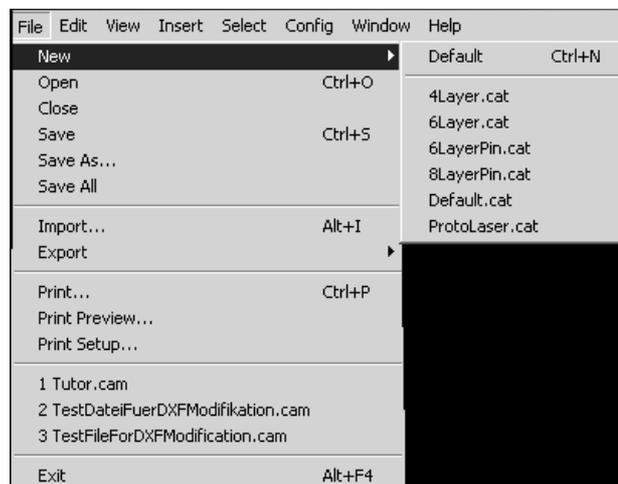
If, for example, you use the same aperture and tool list again and again, you can create your own template file which already contains both these lists. You will then not need to import these aperture and tool lists again for new circuit boards.

All current settings, such as name and colour of the layers are also saved in the template file. More informations you will find in the appendix.

CircuitCAM template files are also saved in the CAM format, but have the file extension "**CAT**" and are saved to the "**NEW_TEMPLATES**" subdirectory.

The available template files are listed in the **NEW** submenu of the **FILE** menu:

Figure 98:



Various template files are saved to the templates directory when you install *CircuitCAM*:

| Template file | Intended Purpose |
|----------------|---|
| Default.cat | Standard template |
| 4Layer.cat | Multilayer board, 4 layers when production is with sequential build-up method |
| 6Layer.cat | Multilayer board, 6 layers when production is with sequential build-up method |
| 6LayerPin.cat | Multilayer board, 6 layers when production is with registration pin method |
| 8LayerPin.cat | Multilayer board, 8 layers when production is with registration pin method |
| ProtoLaser.cat | Template with ProtoLaser tool lists |

Use the "**TUTOR1.CAM**" file you created in the previous exercise as the basis for creating the template file.



Note: The most recently used documents are always listed in the **File** menu so that you can open them quickly.



Note: Click on "**TUTOR1**" in the **File** menu to open the document "**TUTOR1.CAM**" if you have already closed it.

How to create a template file for the following exercises:

- › Click on **ALL** in the **Select** menu to select all graphical objects.
- › Press **DELETE** to delete the selected objects.
- › Select **SAVE AS** from the **FILE** menu.
- › Change to the templates directory "**NEW_TEMPLATES**" in the **SAVE AS** dialog box. This directory is located in the "**LPKF50**" directory.
- › Select the **NEW TEMPLATE (*.CAT)** line from the **FILE TYPE** list box.
- › Enter **TUTOR1** in the **FILE NAME** text box.
- › Click on **SAVE** to save the "**TUTOR1.CAT**" template file.
- › Select **CLOSE** from the **FILE** menu to close the "**TUTOR1.CAT**" template file.

*The "**TUTOR1.CAT**" template file will now be listed in the **NEW** submenu of the **FILE** menu. This file already contains the "**APETUTOR1**" aperture list and the "**TOOLTUTOR1**" tool list.*



Note: You should save this template file under the name "**DEFAULT.CAT**" if you use the same template very frequently.

The template file called "**DEFAULT.CAT**" is the file that is started automatically when *CircuitCAM* is opened.

A new document based on the template file "**DEFAULT.CAT**" can also be opened by clicking  in the **STANDARD** toolbar.

8.2 Move layer

The production data used in exercise 1 represented an ideal case. Naturally, it may be the case in practice that the various production data do not match up as a consequence of differing data origins and must first be corrected.

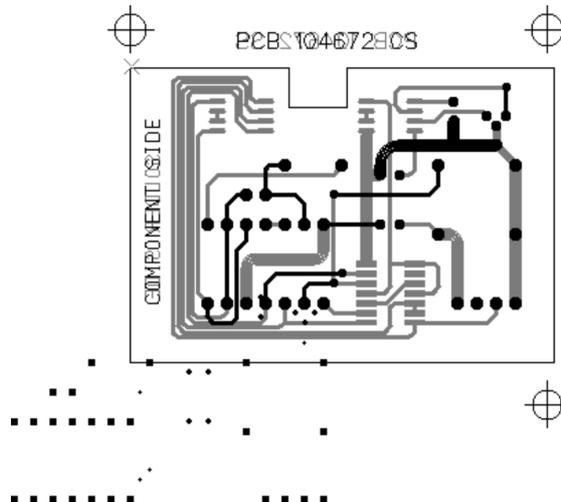
To illustrate the problem, first import the production data into *CircuitCAM* as you learned to do in exercise 1.

Do this by opening a new document based on the "**TUTOR.CAT**" template file:

- › Select **TUTOR.CAT** from the **NEW** submenu of the **FILE** menu.
- › Import the following production data from the "**DATA**" subdirectory into the new document:
 - „Tutor2.cmp"
 - „Tutor2.sld"
 - „Tutor2.drl"

The following image appears in the graphic window once the production data are imported:

Figure 99:



This exercise includes Excellon data which does not coincide with the Gerber data as a result of the specification of a different origin during output in the circuit board CAD program.

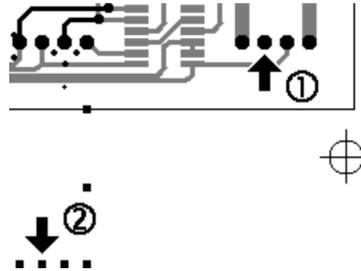
CircuitCAM includes the following function to make the layers coincide again:

 **MATCH UP LAYER**

How to move a layer:

Use a significant object that occurs in both the layers to be brought back into registration as your reference coordinates:

Figure 100:



Note: A significant object can be a drill hole and its solder pad, as long as it is designed by a flash. Both graphical objects have a center coordinate, so that selecting can be done very accurate. Don't use graphical objects which are drawn and not flashed. Only flashed elements can be processed with this function as only these have a defined center.

- › Click on  in the **GRID+UNIT** toolbar.
- › Select the lower drilling in the "**DRILLPLATED**" layer as your reference source.

When you move the mouse-pointer, a "rubber tape" steps up from the reference point.

- › Now you have to mark the upper pad of the layer "**TOP LAYER**" as target-point.

*The layer "**DRILL PLATED**" will be moved and the layers are registered.*



Note: Select **SAVE AS...** in the **FILE** menu and enter "**TUTOR5**". Now click on **SAVE** to save this exercise.

You have now learned how you can match offset layers. Since the next steps for prototype preparation are identical with Exercise 1, we will not list them again here.

To conclude the fifth exercise, you can now enter "**TUTOR5**" in the menu **SAVE AS** under **FILE** menu and click on **SAVE**. Now you can finish the exercise 5 by selecting **CLOSE** from the **FILE** menu.

9.0 Exercise 6

In this section you will learn how you can simplify the importation of production data by the use of import assignments and how to correct production data in which one layer has a different alignment.

9.1 Using import assignments

If you use production data with specified file extensions, e.g. ".drl", ".sld", ".cmp", you can use import assignments in *CircuitCAM* to automate and thus simplify the import of production data. Here you determine which file extension represents what type of production data and assign a layer and an aperture list and tool list to this extension. You will no longer need to specify this information when you later import a file with this extension.

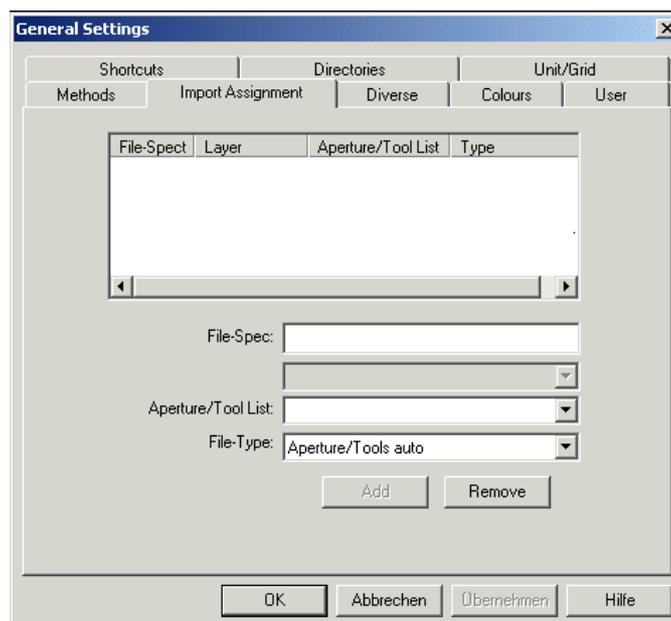
- › Open a new document by clicking on **DEFAULT NEW** in the menu **FILE**.

Save this new document as template file under a new name (Test.CAT) in the table "**NEW TEMPLATES**". Now the import-allocations are saved in a new CAT-file and not in default.CAT.

How to specify import assignments:

- › Open the **CONFIGURATION** menu and select the **GENERAL...** line to open the **GENERAL SETTINGS** dialog box.
- › Click on the **IMPORT ASSIGNMENTS** tab.

Figure 101:



- › Enter ***.DRL** in the **FILE SPECIFICATION** text box to specify the import assignment for the Excellon production data (*.drl).



Note: You can also use wildcards when defining the file specification:

The question mark "?" stands for a single character.

The asterisk "*" stands for a complete word or a group of characters.

- › Select **EXCELLON** from the **FILE TYPE** list box as the files with the extension ".drl" contain Excellon production data.

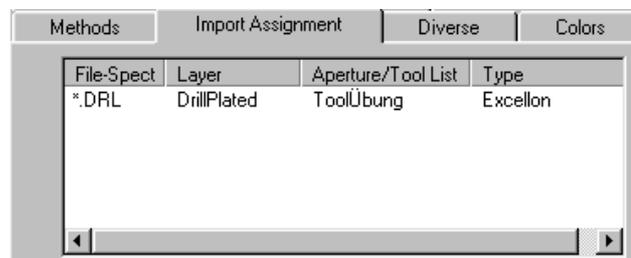


Note: Depending on the file type selected, either the LAYER list box or the TEMPLATE list box will be displayed in the IMPORT ASSIGNMENTS dialog box.

- › Select **TOOLEXERCISE** from the **APERTURE/TOOL LISTS** list box to use this tool list for the Excellon production data to be imported.
- › Select "DrillPlated" from the **LAYER** list box to assign the Excellon production data to this layer.
- › Click on **ADD** to accept the import assignment.

The available import assignments are displayed in the top list box:

Figure 102:



Note: You can amend the preferences for existing import assignments by selecting the import assignment from the list box and editing its parameters.

- › Click on **APPLY** to accept the modified parameters.

You can delete an existing import assignment by selecting the import assignment from the list box and clicking on **REMOVE**.

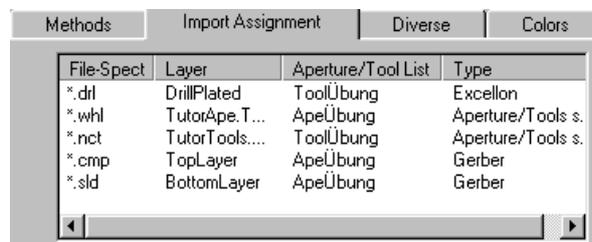
Repeat the works steps on the previous page, to define import instructions for the following production data:

| | Aperture list | Tool list |
|----------------------|---------------------------|---------------------------|
| FILE-SPECIFICATATION | *.whl | *.nct |
| APERTURE/TOOL LIST | ApeExercise | ToolExercise |
| FILE-TYPE | Aperture list / Tool list | Aperture list / Tool list |
| TEMPLATE | TutorApe.txt | TutorTools.txt |

| | Excellon data | Gerber data Component side | Gerber data Solder side |
|----------------------|---------------|----------------------------------|----------------------------|
| FILE-SPECIFICATATION | *.drl | *.cmp | *.sld |
| APERTURE-/TOOL LIST | ToolExercise | ApeExercise | ApeExercise |
| FILE-TYPE | Excellon | Gerber | Gerber |
| LAYER | DrillPlated | TopLayer | BottomLayer |

The following import assignments should now be displayed in the list box:

Figure 103:



- › Click **OK** to close the dialog box **GENERAL SETTINGS** and save the import assignments in the template file Text.CAT.



Note: The above indicated file-specifications, file types, layer... refer exclusively to the included Tutorial data. The file name and the Import assignments are very individual at each user. According to the chosen file-type the list box LAYER or the Listbox TEMPLATE is indicated in the dialog box IMPORT ASSIGNMENTS.

9.2 Rotate layer

In exercise 5 you learned how to correct production data in which the layers did not coincide. This exercise includes Excellon data which are offset against the Gerber data and are additionally rotated by 90°.

Open a new document for this exercise based on the **"TUTOR.CAT"** template file:

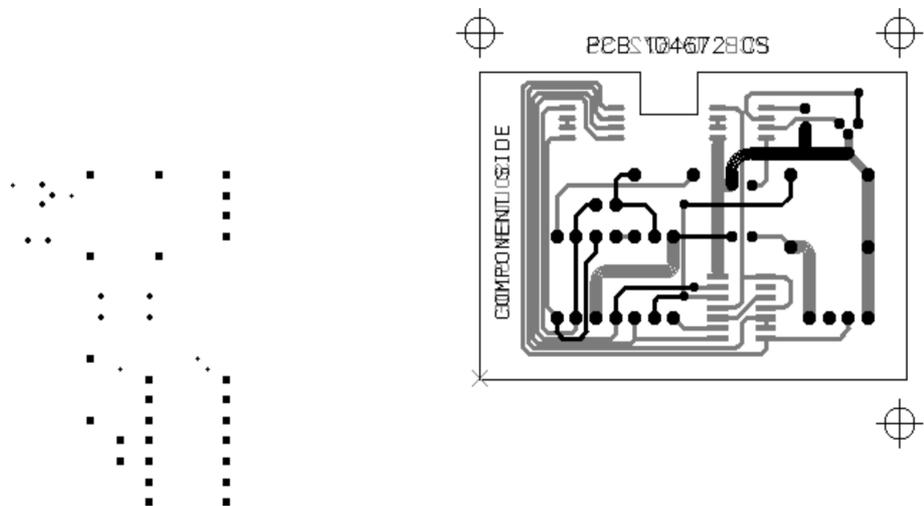
- › Select **"TUTOR.CAT"** from the **NEW** submenu of the **FILE** menu.
- › Import the production data from the "Data" subdirectory into the new document:
 - „Tutor3.cmp"
 - „Tutor3.sld"
 - „Tutor3.drl"



Note: Import the three files in a single operation. Thanks to the definition of the import assignments it is not necessary again to assign layers or tool and aperture lists in the IMPORT dialog box.

The following image appears in the graphic window once the production data are imported:

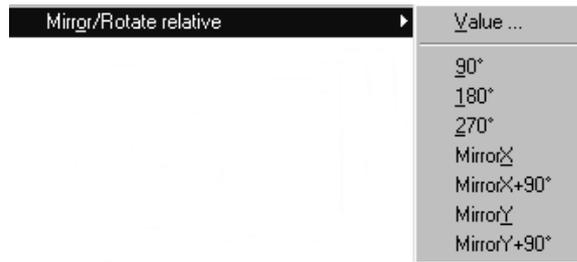
Figure 104:



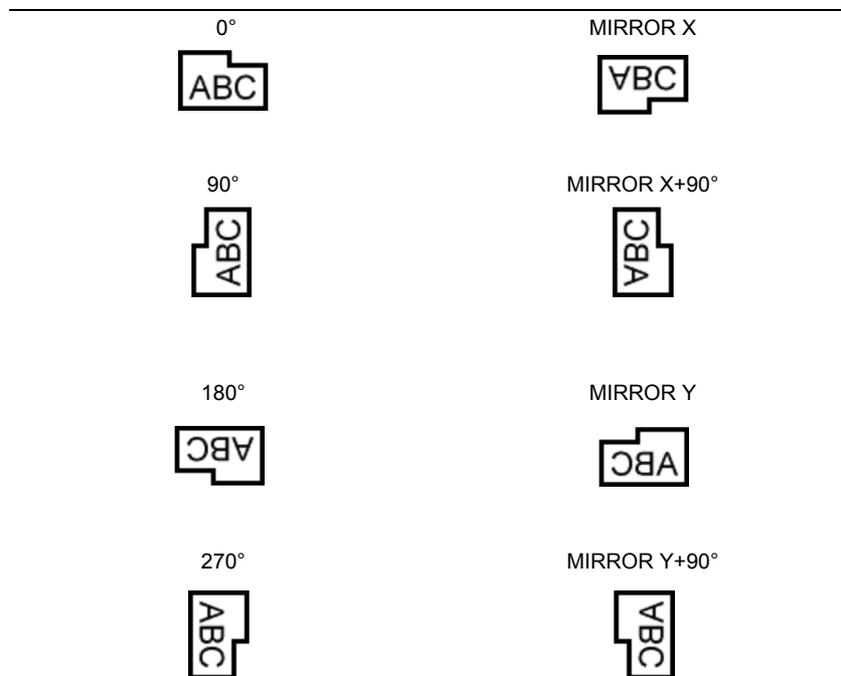
You will need the functions for mirroring or rotating graphical objects in addition to the Match-up-Layer functions you have already used.

You will find these functions in the **MIRROR/ROTATE** submenu of the **EDIT** menu:

Figure 105:

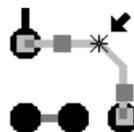


The effects of the various functions are illustrated by this example object



A selected object always rotates around the anchor point:

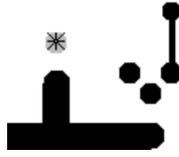
Figure 106:



You may also mirror/rotate around the anchor point of an object which itself will not be mirrored/rotated:

- › Select the anchor point by clicking on the object.
The anchor point is indicated by a cross:

Figure 107:



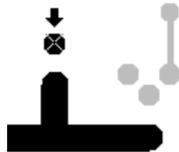
- › Select the objects you wish to mirror/rotate by dragging with the left mouse button rather than by clicking:

Figure 108:



In this way, the anchor point of the object previously clicked on is retained and can be used as the reference point for mirroring/rotation:

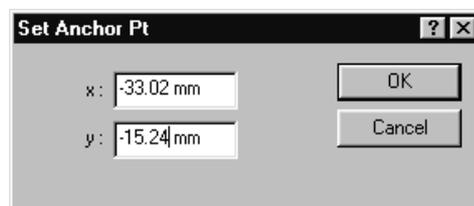
Figure 109:



You can also position this anchor point completely independently of any object:

- › Select **SET ANCHOR POINT...** from the **EDIT** menu.
*The **SET ANCHOR POINT** dialog box showing the current coordinates of the anchor point will be displayed:*

Figure 110:



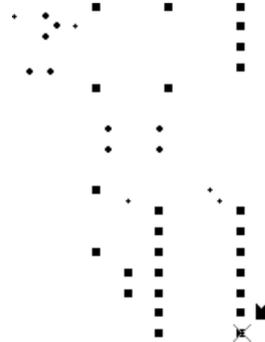
- › Enter the new coordinates for the anchor point in the text boxes and click on **OK**.

In the following exercise you will use the position of a drill hole as the reference point for the rotation of the complete layer.

How to rotate a complete layer:

- › Click on the drill hole at the bottom right to select it.

Figure 111:



The position of this drill hole is now the reference point around which the layer will be rotated.

The active layer "DRILLPLATED" is shown in the LAYER list box of the DEFAULT toolbar:

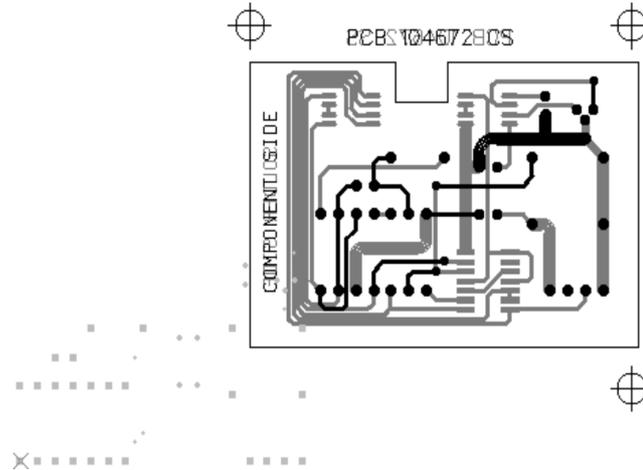
Figure 112:



- › Click on **LAYER** in the **SELECT** toolbar or the  button to select all the objects in the active layer "**DRILLPLATED**".
- › Select 270° from the **MIRROR/ROTATE** submenu of the **EDIT** menu to rotate the entire layer by 270° counter clockwise.

Following rotation, the layer will still have an offset over the Gerber data, but is aligned the same way:

Figure 113:



You can now register the layers against one another using the **MATCH UP LAYER** function, as shown in exercise 5.



Note: You can correct layers that are mirrored in the same way. Do this by selecting **Mirror X** or **Mirror Y** from the **MIRROR/ROTATE** submenu of the **EDIT** menu.



Note: You may use the tutorial files "TUTOR4.CMP", "TUTOR4.DRL" and "TUTOR4.SLD" if you wish to practise mirroring layers.



Note: We recommend that you first rotate or mirror the layer and then move it.



Note: To save the current position of the exercise, select **SAVE AS** from the **FILE** menu. Enter "TUTOR6!" as the filename and click on **OK**.

You have now learned how to define import assignments and how to make rotated and offset layers coincide. Since the further steps in preparation for Prototyping are identical with those in exercise 2, we will not list them here again.

To conclude the sixth exercise, you can now close the document by selecting **CLOSE** from the **FILE** menu.

10.0 Appendix

10.1 Gerber Format

This chapter shows you:

- Where Gerber is used
- The Gerber format syntax
- The difference between drawing and flashing
- The data format
- The structure of the aperture lists

10.1.1 Use of Gerber Format

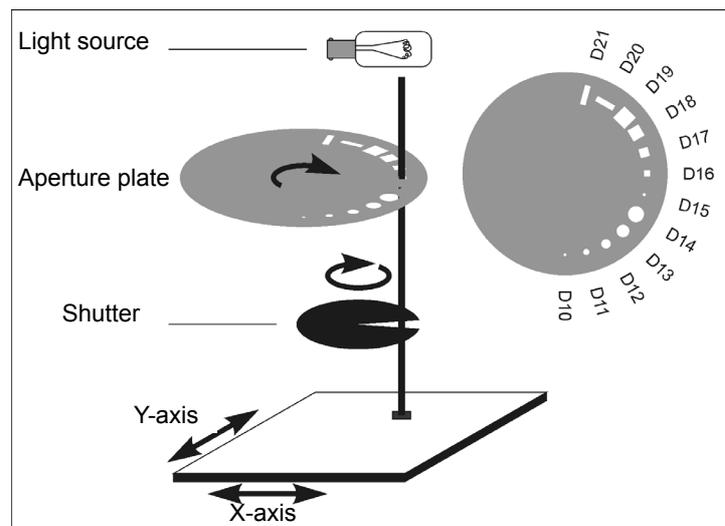
The Gerber format is used for photoplotter control. The photoplotter is used to create films used for etching, e.g. for production of circuit boards. Films are not used for producing circuit board phototypes with circuit board plotters. However, with the aid of the *CircuitCAM* software it is possible to use the same data for control of photoplotters for generation of the required milling data.

10.1.2 Structure of a Photoplotter

For better understanding of the Gerber format, it is most practical to first familiarize yourself with the principle structure of a photoplotter. For us, the most important constituents of a photoplotter are:

- the aperture plate
- the shutter
- light required for exposing the film

Figure 114: Schematic layout of photoplotter



As shown in the figure above, the aperture plate is a disc with apertures at around its circumference. Each of these apertures has a designation. As a rule, the smallest aperture has the designation D10, the next larger D11... and so forth. These apertures have different shapes. For example, they are circular, square or rectangular.

The shutter is located between the film and aperture plate. Its purpose is to allow the light from the bulb to pass through to the film only at certain moments.

10.1.3 Gerber Format Syntax

10.1.3.1 Drawing

Example 1:

A line is to be drawn with the aid of the photoplotters. This line is to have a width of 0.3 mm and a length of 2.3 mm. Moreover, it is to be round at the ends.

For this purpose, a number of commands are sent to the photoplotter. These commands are explained below.

First, it is necessary for the photoplotter to turn the aperture plate so that a round aperture with a diameter of 0.3 mm is placed in front of the shutter. In this example, this is aperture D12.

The command is: **G54D12***

G54 is the command to turn the aperture plate.
D12 indicates the new position of the aperture plate.

Next, it is necessary for the photoplotter to move to the starting point for the line without drawing anything. This means the shutter must remain closed.

The starting coordinate for this line are to be X=100 and Y=150.

This command is: **X100Y150D02***

X100Y150 indicates the position to be moved to.
D02 is the command to move with the shutter closed.

At the starting point for the line, the photoplotter is to open the shutter and move to the end point of the line. The end point is at the coordinates X:102 and Y:150.

The command is: **X102Y150D01***

X102Y150 indicates the position to be moved to.
D01 is the command to move with the shutter open.

Example 2:

Now, another line is to be drawn. This line is to have the same dimensions, however, the ends are to be rectangular. For this purpose, it is necessary to use a rectangular aperture. The edge length of this aperture is also 0.3 mm. On our aperture plate, this is aperture D17.

The command is: **G54D17***

G54 is the command to turn the aperture plate.
D17 is the rectangular aperture with edge length of 0.3 mm.

Next, it is necessary for the photoplotter to move to the starting point for the line without drawing anything. This means that the shutter must remain closed. The starting coordinates for this line are X=100 and Y=100.

The command is: **X100Y100D02***

X100Y100 indicates the position to be moved to.

D02 is the command to move with the shutter closed.

Now, the photoplotter is to open the shutter and move to the end point of the line. The end point is at the coordinates X=102 and Y=100.

The command is: **X102Y100D01***

X102Y100 indicates the position to be moved to.

D01 is the command to move with the shutter open.

10.1.3.2 Flashing

A rectangle was the result of the last command sequence.

Example 3:

It is also possible to generate a rectangle with fewer commands. This assumes that the desired shape is available as an aperture on the aperture plate.

A rectangular aperture with dimensions of 2.3 x 0.3 mm is required on the aperture plate. In our example, this is aperture D20.

It is necessary for the photoplotter to turn this aperture in front of the shutter.

The command is: **G54D20***

G54 is the command to turn the aperture plate.

D20 is the command to select aperture D20 (rectangle).

It is then necessary for the photoplotter to move to position **X:101** and **Y:50** and open and close the shutter here.

The command is: **X101Y50D03***

X101Y50 indicates the position to be moved to.

D03 is the command to open the shutter and close it again without moving.

The last command in any Gerber file must be the command **M02***, to define the end of the Gerber file.

This block of commands represents the complete Gerber file:

```
G54D12*
X100Y150D02*
X102Y150D01*
G54D13*
X100Y100D02*
X102Y100D01*
G54D30*
X101Y50D03*
M02*
```

10.1.4 Aperture List

The Gerber file specify only which apertures are to be used. However, this provides no information on the shape or size of the individual apertures. This information is obtained from a second file: The aperture list.

There is no defined format for the aperture list. Every CAM or CAE software has its own format for the aperture list.

Example:

```
! GAPFile Version 1.0
! Created: Thu Aug 6 17:23:45 1998
! Created by: ECAM 3.21a GAPFile Version 1.0
```

| | Type | Width | | Length | | Rotate |
|-----|-----------|---------|--------|---------|--------|--------|
| | | English | Metric | English | Metric | |
| D10 | ROUND | 3.93 | 0.100 | 0.00 | 0.000 | 0 |
| D11 | ROUND | 7.87 | 0.200 | 0.00 | 0.000 | 0 |
| D12 | ROUND | 11.81 | 0.300 | 0.00 | 0.000 | 0 |
| D13 | ROUND | 15.74 | 0.400 | 0.00 | 0.000 | 0 |
| D14 | ROUND | 19.68 | 0.500 | 0.00 | 0.000 | 0 |
| D15 | SQUARE | 3.93 | 0.100 | 0.00 | 0.000 | 0 |
| D16 | SQUARE | 7.87 | 0.200 | 0.00 | 0.000 | 0 |
| D17 | SQUARE | 11.81 | 0.300 | 0.00 | 0.000 | 0 |
| D18 | SQUARE | 15.74 | 0.400 | 0.00 | 0.000 | 0 |
| D19 | SQUARE | 19.68 | 0.500 | 0.00 | 0.000 | 0 |
| D20 | RECTANGLE | 90.55 | 2.300 | 11.81 | 0.300 | 0 |
| D21 | RECTANGLE | 11.81 | 0.300 | 90.55 | 2.300 | 0 |

The aperture list has a tabular structure. Each line contains information on one aperture. The information is subdivided into columns.

The name of the aperture is contained in the first column.

The second column describes the shape of the aperture.

Columns three and four indicate the size in the units mil and mm.

For round apertures, the diameter is specified here, for square apertures, the edge length.

For rectangular apertures, the extension in the X direction is specified in these columns.

Columns five and six contain the extension in the Y direction. These columns are not used for round or square apertures.

10.1.5 Units, Format and suppressed Zeros

The coordinates for the Gerber file do not provide any information on whether the values are to be interpreted in millimeters or inches. Moreover, the Gerber file does not indicate where the decimal point should be. For example, the second line of the sample double file could be interpreted as follows:

X100Y150D02* could mean
 X: 10.0 mm, Y: 15.0 mm
 or
 X:1.00 inch, Y: 1.50 inch.

This information can be present in the aperture list or contained in an additional file also containing production instructions. In our example, such a file is used:

```
! DSNFile Version 1.3
! Created: Thu Aug 6 17:23:45 1998
! Created by: ECAM 3.21a
!
DSNFile   Version 1.3
```

```
Database_type:   RS274X
Database_format: 2.3
Coordinate_mode: Absolute
Zero_suppression: Leading
Coordinate_units: Metric
```

The question regarding the units can be clarified quickly. In the last line, the expression `Coordinate_units: Metric` means that the coordinates must be read in mm.

The position of the decimal point can be read out of the combination of two pieces of information. These are the `Database_format` and the `Zero_suppression`.

Format 2.3 indicates the number with two digits in front of the comma and three digits behind the comma. Actually, the coordinates in the sample Gerber file should consist of five digits. However, as you can recognize, only two or three digits are used. The reason for this is that the zeros at the beginning of the coordinates (leading zeros) and at the end of the coordinates (trailing zeros) are not written. Here, we speak of suppressed zeros.

There are two possibilities for suppressing the zeros: Only the leading or only the trailing zeros are contained in the coordinates.

From the first line of the sample Gerber file, written with all zeros: X00100Y00150D02*, the following results when the leading zeros are suppressed: X100Y150D02*, and the following results when the trailing zeros are suppressed: X001Y0015D02*.

In order to correct the decimal point, it is necessary to know which zeros have been suppressed.

If the leading zeros were suppressed, the number of digits following the decimal point is decisive.

If trailing zeros were suppressed, the number of digits in front of the decimal point is decisive for the position of the decimal point.

In this example, you can see the line Zero_suppression: Leading. This means that the leading zeros are suppressed. This can also be recognized by the fact that the coordinates in the sample Gerber file never starts with a zero.

10.1.6 Extended Gerber

When the Gerber data is transferred from the customer to the template manufacturer, it frequently occurs that information required for production such as the aperture list is not received by the manufacturer. In this case, it is necessary to confer with the customer regarding the missing information often leading to unnecessary telephone calls and a delay in production.

For this reason, the Gerber format has been extended. This extended format is called "Extended Gerber" or "RS247X".

The advantage of this format is that all necessary information for reading the Gerber file correctly is already contained in the file itself. This means that no separate aperture list is required, because the definition of the aperture is already contained in the Extended Gerber file.

The following example shows the syntax for definition of units and formats as well as the definition of the apertures. This definition block is always located at the beginning of the Extended Gerber file:

```
%FSLAX23Y23*%
%MOMM*%
%ADD10C,0.10000*%
%ADD11C,0.20000*%
%ADD12C,0.30000*%
%ADD13C,0.40000*%
%ADD14C,0.50000*%

%ADD15R,0.10000X0.10000*%
%ADD16R,0.20000X0.20000*%
%ADD17R,0.30000X0.30000*%
%ADD18R,0.40000X0.40000*%

%ADD19R,0.50000X0.50000*%
%ADD20R,2.30000X0.30000*%
%ADD21R,0.30000X2.30000*%
```

Definition of the format is accomplished with the following line:

%FSLAX23Y23*%

FS stands for Format Statement.

L stands for leading zero suppression.

As a result, zeros appear only at the end of the coordinates.

A stands for absolute coordinates meaning that all coordinates are based on the zero point.

X23Y23 stands for the number of digits in front (2) and in back (3) of the decimal point. It is possible to define this differently for the X and Y axis.

The units for the Gerber data are defined in the next line: **%MOMM*%**

MO stands for mode.

MM stands for millimeter.

In the event that the Gerber data is present in inches, **IN** would be substituted for **MM**.

The apertures are defined by the following lines:

%ADD15R,0.1000X0.1000*%

AD stands for aperture definition

D15 is the name of the aperture

R is the shape of the aperture. R stands for rectangle; C for circle

0.1000X0.1000 stands for the size of the orifice in the X and Y direction.



Note: Further information on the Gerber Format is given on the following Internet page: www.barco.com, and there under *Electronic tooling systems - Products - data format descriptions*.

10.2 Predefined Layer

| Layer name | Use | Template file | | | | |
|-------------------------|---------------------------------------|---------------|--------------|--------------|--------|--------|
| | | default | Single Sided | Double Sided | 4Layer | 6Layer |
| TOPLAYER | Layout Component side | X | | X | X | X |
| BOTTOMLAYER | Layout solder side | X | X | X | X | X |
| LAYER2 | Layout Multilayer 2 | | | | X | X |
| LAYER3 | Layout Multilayer 3 | | | | X | X |
| LAYER4 | Layout Multilayer 4 | | | | | X |
| LAYER5 | Layout Multilayer 5 | | | | | X |
| DRILLPLATED | Holes (throughplated) | X | X | X | X | X |
| DRILLUNPLATED | Holes (not throughplated) | X | | X | X | X |
| BORDOUTLINE | Circuit board outline | X | X | X | X | X |
| TEXTTOP | Text for component side | X | | X | X | X |
| TEXTBOTTOM | Text for solder side | X | X | X | X | X |
| CUTTINGOUTSIDE | Contour routing outside | X | X | X | X | X |
| CUTTINGINSIDE | Contour routing inside | X | X | X | X | X |
| SOLDERMASKTOP | Solder resist mask component side | X | | X | X | X |
| SOLDERMASKBOTTOM | Solder resist mask solder side | X | X | X | X | X |
| SILKSCREENTOP | Component printing on component side | X | X | X | X | X |
| SILKSCREENBOTTOM | Component printing on solder side | X | | X | X | X |
| RUBOUTTOP | <i>Rubout</i> surface Component side | X | | X | X | X |
| RUBOUTBOTTOM | <i>Rubout</i> surface Component side | X | X | X | X | X |
| RUBOUTALLLAYER | <i>Rubout</i> surface All layers | X | | X | X | X |
| INSULATETOP_* | Insulation channels on component side | X | | X | X | X |
| INSULATEBOTTOM_* | Insulation channels on solder side | X | X | X | X | X |



Note: We recommend maintaining the names of these layers, because *CircuitCAM* provides these layers for preconfigured jobs.

10.3 Importing DXF files

When importing DXF files, the problem frequently arises that the data does not contain surface information, but rather only outline data which may also be split up into individual line segments. In *CircuitCAM* such data can be converted quickly and easily into surface data with the aid of the “Combine to Polygon” function. This means convenient processing of DXF data particularly in the RF and microwave range.

DXF is a data format which is used primarily by CAD programs for machine design for 2 and 3-D design. AutoCAD is the program used most frequently to generate DXF data. The major difference between DXF and Gerber data is that in the DXF format, one file is used for all layers and that the conductive copper surfaces of the circuit board are usually shown in the drawings only as contours and not as filled surfaces.

The procedure for importing DXF data is identical with that for the Gerber or Excellon data as a matter of principle:

- › Click **IMPORT** and select the DXF file to be imported.

CircuitCAM recognizes the format of this file.



Note: If *CircuitCAM* does not recognize the DXF format, check again to ensure that you selected the correct file. If so, select DXF manually in the list box “File type”.

- › Select one of the entries present in the **APERTURE LIST** box (DXF mil unit, DXF inch unit or DXF mm unit).

These lists vary in terms of different resolutions used for data import. In this manner, the size of the circuit board is defined. In contrast to extended Gerber format, for example, the DXF format does not contain any information on the size of the circuit board.

- › After selecting one of the entries, check the circuit board size (size x.y). If the values do not correspond with the expected size, select one of the other entries.
- › Check the size again. If necessary, use the “Preview” button to check the layout and its size.

If none of the entries leads to a realistic result, define your own entry with a resolution (unit), which provides correct representation.

- › Select the “**Top Layer**” in the layer list box.

In most cases, the layer name is already defined in the DXF file. In this case, selection of the layer is required only to continue in the program, because the **IMPORT** button releases all required parameters only after specifying.

In rare cases, layer names may not be specified in the DXF file. In this case, all data is assigned to the selected layer. This can be corrected in AutoCAD.

Recreate the DXF file and repeat the import process.

- › Click **IMPORT**.

*The DXF data appears in the *CircuitCAM* graphic window.*



Caution! Check again whether the dimensions of the display layout correspond to the expected size. Simple viewing is not sufficient, because although the data appears with the correct relationship, the layout can still be represented too large or too small by a certain factor.

- › Click the upper-left corner of the layout or an element close to it for exact checking.

- › Click the right-mouse key.

A pop-up menu opens.

- › Select **Measure** from this menu.

- › Move the mouse pointer to the opposite corner.

A rubber band is tensioned between the mouse pointer and the selected element and the length of the rubber band as well as the Y and X path (cathetus) is displayed in the bottom left corner of the CircuitCAM window (in the status bar).

You can check the layout size on the basis of these values.

If the layer names of the DXF file do not correspond to those in *CircuitCAM*, these names should be changed in order to use the prepared insulating and export jobs in *CircuitCAM*.

- › Select the layer names to be changed in the **LAYER** list box.

- › Select the entry **Layer** in the **SELECT** menu to select all graphic elements in the selected layer.

- › Select a suitable layer in the **LAYER** list box to move all elements to this layer.

Proceed in the same manner for all other DXF data layers.

All copper surfaces required as conductive copper (conductors, solder eyes) on the circuit board must be represented as filled surfaces in *CircuitCAM*. If this is not the case in the DXF files, it is necessary to modify the data.

Four different possibilities exist in the DXF data for defining copper surfaces:

Possibility 1: As continuous, closed line.

Figure 115:



Possibility 2: As closed line, however with individual line segments.

Figure 116:



Possibility 3: A continuous closed line surrounds one or more other lines. The differences between these surfaces defines the remaining copper surface.

Figure 117:



Possibility 4: Same as 3, however with individual line segments.

Figure 118:



How is this data modified?

Possibility 1.

- › Select the closed line surrounding the surface to be filled by clicking with the left-mouse key.
- › Select the function "**PATH TO POLYGON**" from the **EDIT DIVERSE** menu or press the key combination **Shift+P**.

The selected element appears as a filled surface (Polygon). Proceed in the same manner for all other elements.



Note: If the result is not as you expected, use the “Undo” function and check the element selected.

Multiple selection of more than one element can be used before executing the function "Path to Polygon". However, ensure that the selected elements are of the same type as described under possibility 1.

Possibility 2.

For this situation, modification is similar to possibility 1. However, it is first necessary to connect the individual line segments of a closed line to form a continuous line.

- › Mark all individual line segments of one or more closed lines with the aid of the various selection possibilities (left mouse click, Ctrl + left mouse click, Shift + left mouse click, left mouse click and pull rectangle; details in *CircuitCAM Manual*).

After completing selection, select the function "**COMBINE**" from the **EDIT CURVE** menu or the key combination **Ctrl+B**. Then proceed as described for possibility 1.



Note: If the result of the **COMBINE** function provides an erroneous result (line segments connected which were not to be connected), ensure that the value for the capture interval for the connect function was set realistically. This value defines the max. distance between line segments to be connected with one another. If the value is too large, adjacent segments are connected with one another which should not be connected. If the value is too small, segments which should be connected with one another may not be connected because their beginning or end points are not precisely on top of one another due to inaccuracies in the DXF data. The value for the capture interval is given on the tab “Diverse” in the Configuration General Setting menu. The value for the parameter "Threshold curve-combine" should be 0.01 to 0.05 mm. This value depends on the structure of the layout.

Possibility 3.

With this possibility, it is necessary to subtract one or more parameters from another larger parameter. This modification must be accomplished step for step.

First, it is necessary to modify the enclosing lines to form polygons as under possibility 1.

The outer, larger polygon (and only this polygon!) can be selected and changed to a particular polygon by selecting the function "**POLYGON WITH CUTOUTS**" in the **INSERT** menu. Holes or cutouts can be inserted in such a polygon

- › Select all the polygons within the larger polygon which are to be deleted as cutouts
- › After selecting the above, select the function "**CUTOUTS**" in the **INSERT** menu.

The result is a polygon with openings, which has precisely the expected design - if all selections have been performed correctly.



Note: If the result is not what you expected, use the “Undo” function and check the elements selected.

Possibility 4.

For this situation, modification is similar to possibility 3. However, it is first necessary to connect the individual line segments of an enclosing line to form one continuous line as in possibility 2. Then proceed as described for possibility 3.

After completing the above modifications, your layout can be further processed with all prepared insulating and export jobs. All jobs present for this purpose can be used.

A *CircuitCAM* file (**TestFileForDXFModification.CAM**) is part of the data sub directory. This file contains examples for the possibilities mentioned above. This file serves for practicing the procedures described above and obtaining a better impression of the descriptions.

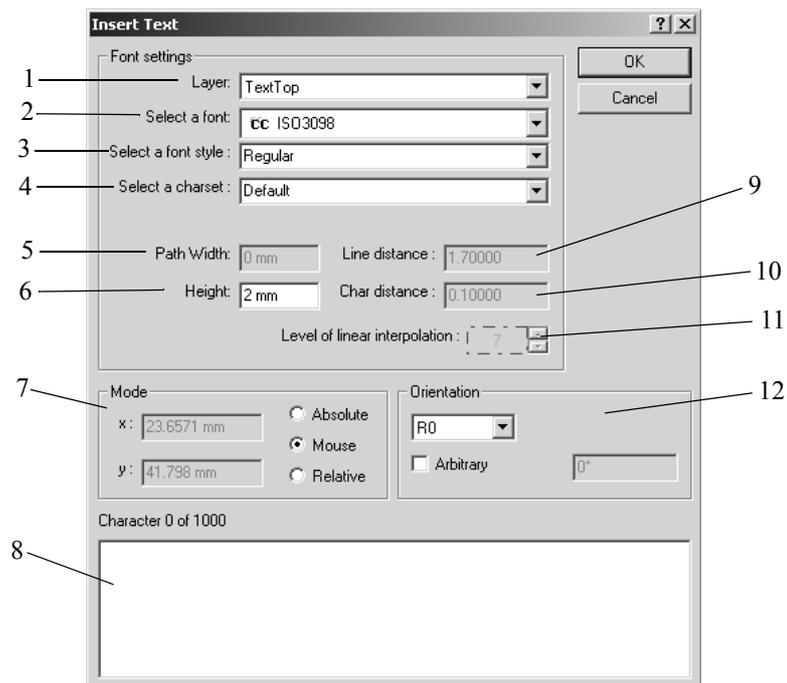
10.4 Text Function

In *CircuitCAM* it is possible to insert texts in a graphic, using the Windows fonts that are installed on the PC. Both TTF and TTC fonts are possible, depending on the country setting of the operating system.

- › For this purpose, click **TEXT, TOPLAYER** (or **BOTTOM LAYER**) in the **INSERT** menu.

The following window appears:

Figure 119:Text window



- | | |
|--|---|
| <p>1- Layer: List box for selection of layer on which text is to be located.</p> <p>2- Select a font: Here, it is possible to select the font for the text.</p> <p>3- Font style: Here, you can choose between regular, italic, bold and bold italic.</p> <p>4- Character set: Here, you can select different character sets within a font.</p> <p>5- Path width: Here, the path width for the text can be specified in mm.</p> <p>6- Height: Here, the height of the text can be specified in mm.</p> | <p>7- Mode: Radio buttons for setting the text position with the mouse (absolute or relative)</p> <p>8- Text entry window: Here, you can enter the text to be positioned on the layer.</p> <p>9- Line distance: Distance between the lines can be set</p> <p>10- Distance: Distance between the characters can be set</p> <p>11- Level of linear interpolation: Here the quality of the edge resolution of the Windows fonts can be modified in steps from 1 to 10</p> <p>12-Orientation: After activation of "arbitrary" an angle can be specified in the right box. When not activated, the angle can be turned" by exactly 90° and 180° in the right window.</p> |
|--|---|

- › Now open the **SELECT FONT** list.

The selection list for the fonts appears.

- › Select a font.

*The selected font appears in the display box **SELECT A FONT**.*

- › Click momentarily on the bottom white text entry window.

The entry cursor is activated and flashes in the text entry box.

- › Then you can enter the desired text in the bottom window.

- › Click **OK**.

- › Then position the text on the layer and arrest it by clicking once with the left mouse key.

A text created in this manner can only be processed with a tool. The tool used here is defined as **LPKFMILLINGTEXT TOOL** under **CONFIG - FORMAT CONFIGURATIONS - LPKF CIRCUIT BOARD PLOTTER TOOL LIST** and can be changed there.



Note: In the PCB version of *CircuitCAM*, DXF files with texts can also be imported and interpreted correctly.

This assumes, however, that true type fonts are used! MTEXT entities in the DXF file are not supported.

The major advantage is that it is no longer necessary to divide up the texts into vectors in *AutoCAD*, which would result in a lower data quantity and corresponding appearance.

10.5 Generating Ground Planes

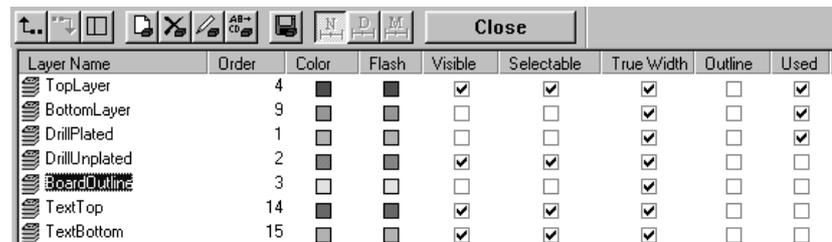
CircuitCAM offers the possibility of automatically and simply generating ground planes in the PCB layout. This means that extensive construction work is no longer necessary for ground planes when the EDA system does not provide such a function. This then reduces the milling work and provides a simple computer-oriented design of the layout.

The following instructions are an example. In this case, the same file was used which was used for the main exercises. If other files are used, the circuit board representations naturally look different.

Proceed as follows to familiarize yourself with this function:

- › Open the file “**TUTOR1.CAM**”
- › Switch all of the used layers to “Invisible” except for the TopLayer by “removing” the check in the list and close the *Layer* list window.

Figure 120: Layer for ground surface



| Layer Name | Order | Color | Flash | Visible | Selectable | True Width | Outline | Used |
|---------------------|-------|-------|-------|-------------------------------------|-------------------------------------|-------------------------------------|--------------------------|-------------------------------------|
| TopLayer | 4 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> |
| BottomLayer | 9 | ■ | ■ | <input type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> |
| DrillPlated | 1 | ■ | ■ | <input type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> |
| DrillUnplated | 2 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BoardOutline | 3 | ■ | ■ | <input type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextTop | 14 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextBottom | 15 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

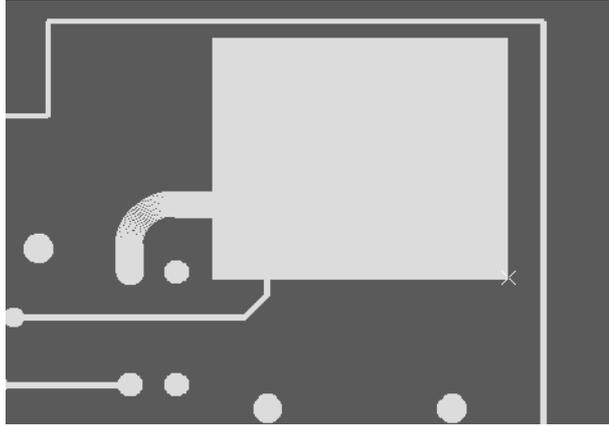
- › Select the **TOPLAYER** as the active layer (See illustration below).

Figure 121: Top layer



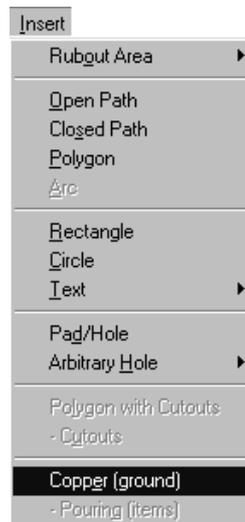
- › Construct the desired ground surface with the **RECTANGLE** or **POLYGON** function. Then press **ESC** to quit this function.

Figure 122:



- › Select the constructed surface by clicking with the left-mouse key in the vicinity of a corner and select the function **COPPER/GROUND** in the **INSERT** menu.

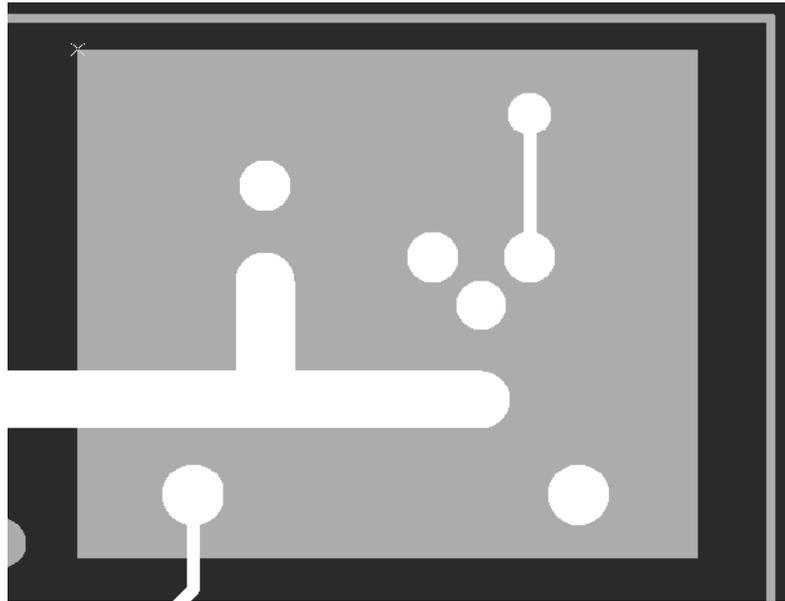
Figure 123:



- › Select all elements within the ground surface by pressing and pulling with the left mouse key. However, not the plane itself.

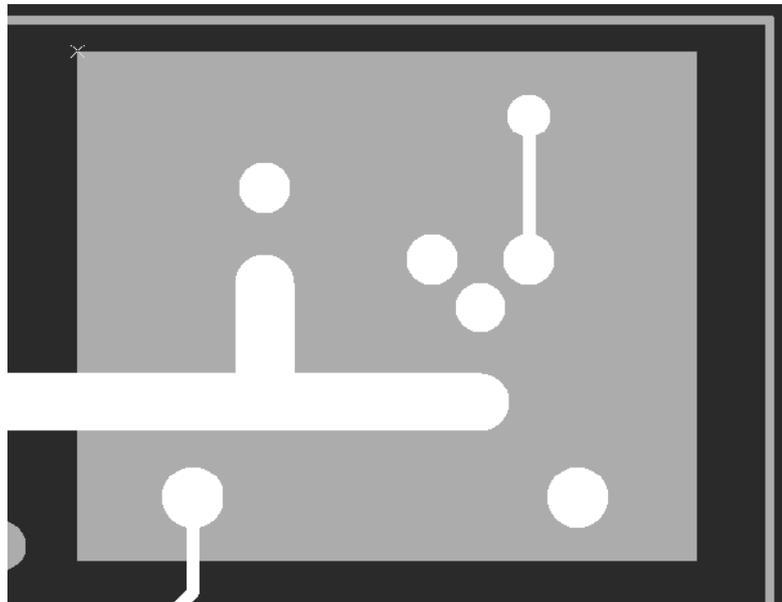
In this manner, all elements which are to be released from the ground surface are marked.

Figure 124:



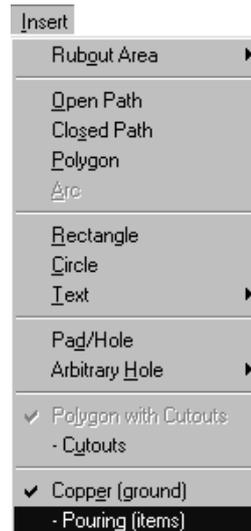
- › Now deselect all elements not removed, i.e. which are to be connected to ground. For this purpose, press the **SHIFT** key and click the left-mouse key in the vicinity of the element in question.

Figure 125:



- › Now select the function **-POURING (ITEMS)** in the **INSERT** menu.

Figure 126:

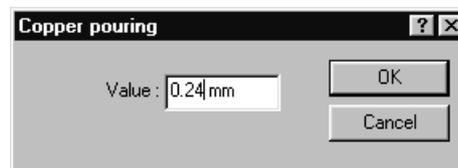


- › In the **COPPER POURING** box enter the value by which the conductors and solder pads located inside of the ground plane are to remain free. This is the insulation distance between ground and the other signals.



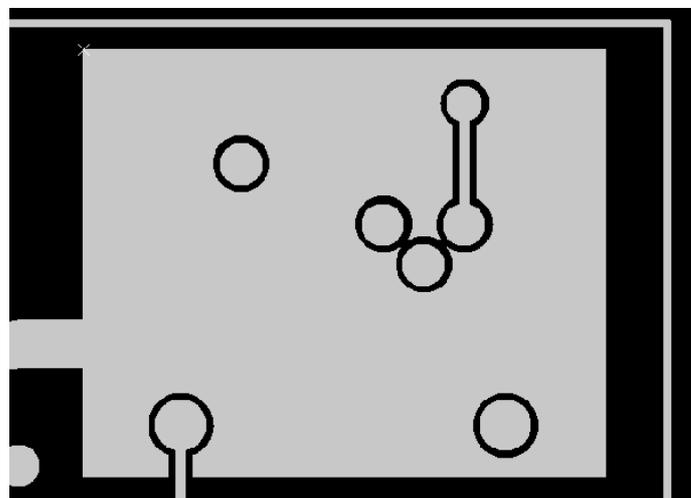
Note: This value should be adapted to the insulating tool to be used, e.g. for the universal cutter 0.2mm, the value should be set to 0.24 mm.

Figure 127: Copper pouring



The desired ground surface is then completed. The result appears in the graphic window.

Figure 128:



10.6 Design Rule Check

The *CircuitCAM* PCB version allows a “Design Rule Check”. This means that before starting insulating or data output, it is possible to check whether the desired minimum intervals have been maintained between conductors and/or pads.

For this purpose, the layer to be checked and the dimension of the minimum interval are simply defined. Those points at which the minimum interval is not maintained are indicated graphically in the layout. Moreover, the following intervals can be measured:

1. Distance from hole to hole
2. Distance from hole to conductor
3. Distance from hole or conductor to circuit board frame
4. Minimum width of remaining copper ring around holes

These measurement possibilities increase the reliability for circuit board prototyping considerably, because many errors can be eliminated before they do any damage.

For the following work instructions we again want to point out that this is an example. The basis for layout representations are the files from the exercises in the main chapter. If other data is used, some representations look different.

Proceed as follows to familiarize yourself with this function.

- › Open the file **TUTOR2.CAM** in the **FILE** menu

Checking the Distance between Two Layers:

- › Switch all layers except the TopLayer, DrillPlated and BoardOutline to invisible and close the *Layer* list window.

Figure 129:

| Layer Name | Order | Color | Flash | Visible | Selectable | True Width | Outline | Used |
|---------------|-------|-------|-------|-------------------------------------|-------------------------------------|-------------------------------------|--------------------------|--------------------------|
| TopLayer | 4 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BottomLayer | 9 | ■ | ■ | <input type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | 1 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillUnplated | 2 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BoardOutline | 3 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextTop | 14 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextBottom | 15 | ■ | ■ | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

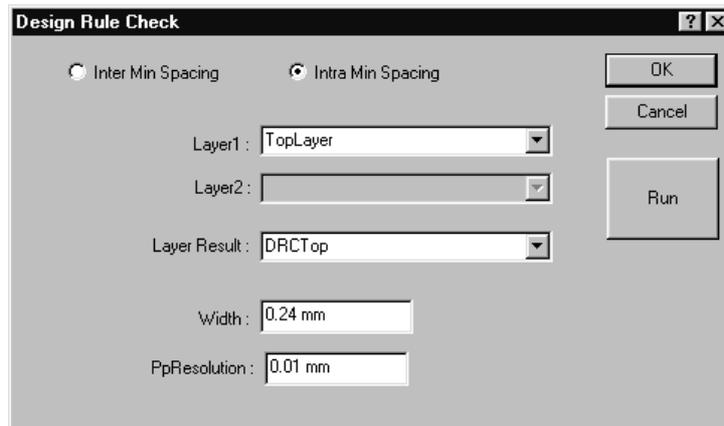
1. Checking the interval within a layer:

This Design Rule Check checks the minimum desired interval within a layer. This serves for checking whether a milling tool with the desired width (0.2 mm) can be used. On the other hand, it is also possible to check for the minimum interval required for the electrical properties.

- › Select the function **DESIGN RULE CHECK** in the **EDIT** menu

The following window appears:

Figure 130: Design Rule Processing



The display may vary depending on whether the radio button "**Enter min spacing**" or "**Intra min spacing**" is selected.

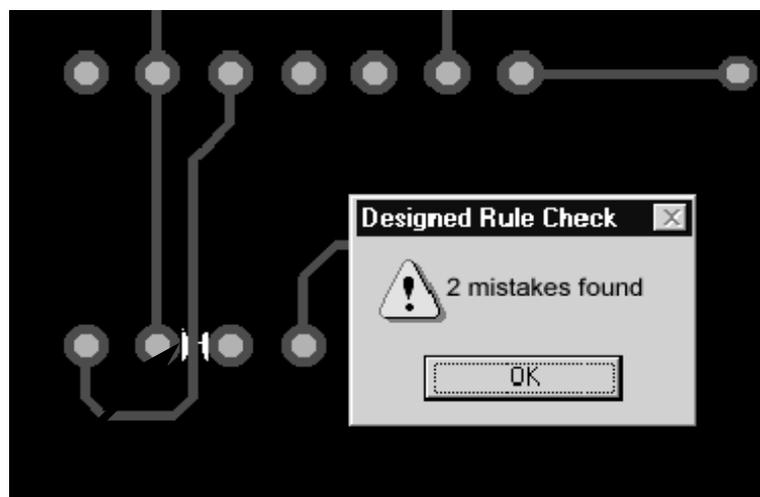
- › Select the layer to be checked in the **LAYER1** list.
- › Enter any desired name in the **LAYER RESULT** for the layer result on which the positions of any errors present are to be displayed graphically.
- › Enter the minimum interval to be checked under **WIDTH**.

If the possible use of a milling tool is to be checked, enter a value approx. 0.02 mm larger than the diameter of the tool in order to avoid later flaws resulting from rounding errors when insulating.

- › Select the value **0.5 mm** here to provoke errors.
- › Click **Run**

The following message appears:

Figure 131: Design Rule Check, Message



CircuitCAM reports 2 errors, i.e. the minimum interval of 0.5 mm was not maintained at 2 points. These two points are indicated graphically in the layout (see arrows).

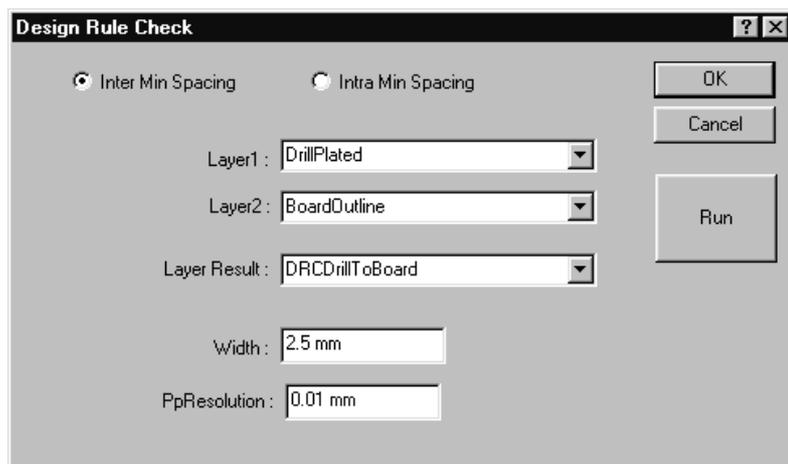
2. Checking the interval between 2 layers:

This Design Rule Check serves for checking the minimum interval of holes or components in relation to the circuit board limit.

- › Select **DESIGN RULE CHECK** in the **EDIT** menu

The following window appears:

Figure 132: Design Rule Check

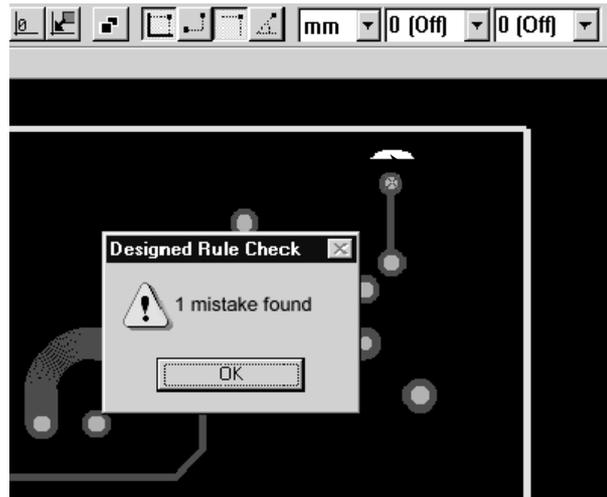


The illustration can vary depending on whether the radio button "Enter min spacing" or "Intra min spacing" is selected.

- › Select the first layer to be checked in the **LAYER1** list.
- › Select the second layer to be checked in the **LAYER2** list.
- › Enter in the **LAYER RESULT** box any desired name for the layer result on which the positions of any errors present are to be indicated graphically.
- › Enter the minimum interval to be checked under **WIDTH**.
- › Select the value **2.5 mm** here to provoke errors.
- › Click **RUN**.

The following message appears:

Figure 133: Design Rule Check, Message



CircuitCAM reports 1 error, i.e. the minimum interval of 2.5 mm was not maintained at 1 point. This point is indicated graphically in the layout (see arrow).

10.7 CircuitCAM template file

CircuitCAM also offers the option of creating a template file with customized basic settings comparable to the format template in a word processor.

The template file is where all settings located in the subentries of the menu bar are saved.

This also includes the names and colours of the layers, the imported apertures and tool lists, the isolation jobs and the export jobs, along with all entries located in **GENERAL SETTINGS** (with the exception of the “User” and “directories” tabs) of the **CONFIGURATION** menu. No graphic data will be saved in the template file.

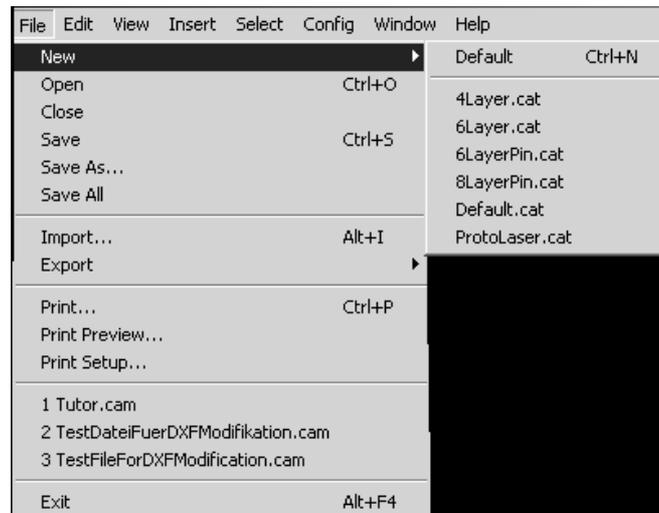
CircuitCAM template files are saved in the CAM format, but have the file extension “cat” and are saved to the “**NEW_TEMPLATES**” subdirectory.

After *CircuitCAM* has been installed, several template files will already be saved to in the template directory and “**DEFAULT.CAT**“ will be identical to “**DOUBLESIDED.CAT**“.

| Template file | Purpose |
|----------------|---|
| Default.cat | Default template file |
| 4Layer.cat | Multilayer board, 4 layers when production is with sequential build-up method |
| 6Layer.cat | Multilayer board, 6 layers when production is with sequential build-up method |
| 6LayerPin.cat | Multilayer board, 6 layers when production is with registration pin method |
| 8LayerPin.cat | Multilayer board, 8 layers when production is with registration pin method |
| ProtoLaser.cat | Template with ProtoLaser tool lists |

The available files are listed in the **NEW** submenu when you select it from the **FILE** menu:

Figure 134:



If, for example, you use the same aperture and tool list again and again, you can create your own template file which already contains both these lists. You will then not need to import these aperture and tool lists again for new circuit boards.

All current settings, such as name and colour of the layers are also saved in the template file.

Here you have the opportunity to remove layers that you do not require from the layer list, or to insert unique colour tags and permanently assign these new functions.



Note: You should save this template file under the name **“DEFAULT.CAT”** if you use the same template very frequently.



Note: The template file with the name **“DEFAULT.CAT”** is the file **CircuitCAM** opens automatically when it is started.

You can also open a new document based on the **“DEFAULT.CAT”** template file by clicking on  in the **DEFAULT** toolbar.

10.7.1 Creating your own template file

As already described previously, *CircuitCAM* gives you the opportunity to save your own settings in the form of template files.

An example of this would be changing an apertures/tool list and modifying isolation jobs for your own purposes. A basis for this would be the default template file **“DEFAULT.CAT”**.

› Start *CircuitCAM*

The template file called "DEFAULT.CAT" is the file that is started automatically when CircuitCAM is opened.

- › Store this template file under its own name.
- › Click **SAVE AS** in the **FILE** menu
- › Now select the dialog box opened as File of type **NEW TEMPLATE (*.CAT)**. Then select the directory **NEW_TEMPLATES**, where all other template files are located.
- › In Names, type in **OWN** and click on **OK**.

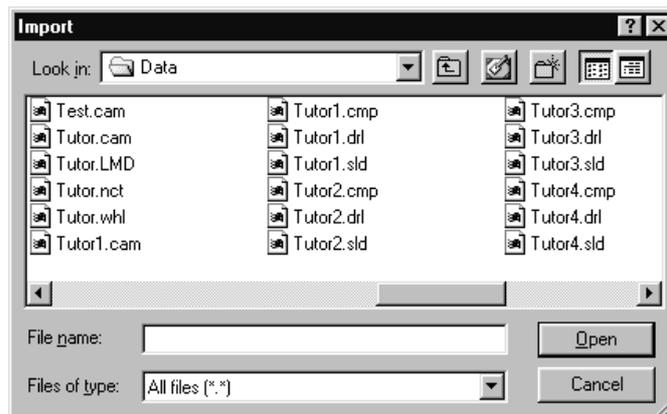
Your new template file “**OWN CAT**“ has now been saved. It contains precisely the same settings as the “**DEFAULT.CAT**“ template file.

All subsequent changes will, however, be saved to your own.cat file. This .cat file is available at the start of a new project in the File, New menu.

10.7.2 Saving an aperture list in the template file

- › Select **IMPORT** from the **FILE** menu.
The OPEN file menu field will be displayed:

Figure 135:



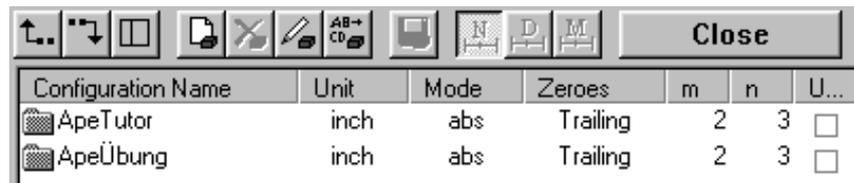
- › Open the file “**TUTOR.WHL**“ in order to import the blind-list and give a marking name, for example **APERTURES**, for the aperture-/tool list.

The file **TUTOR.WHL** is only an example in this case. You should carry out the same procedure with your own aperture list in future.

- › To import the aperture list **TUTOR.WHL**, make all the required settings in the **IMPORT** dialog box, as described in Exercise 1.
- › Select the **GERBERAPERTURE LIST...** line from the **FORMAT CONFIGURATIONS** submenu of the **Config** menu.

CircuitCAM shows all imported aperture lists in the list window.

Figure 136:

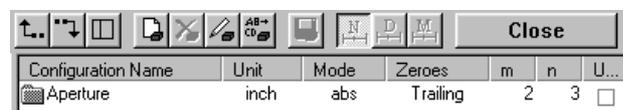


Let us assume that you only want to use your **APERTURES** list in future. This would mean that all other lists can be deleted.

- › Left click to select “**APETUTOR**“ and click on  .

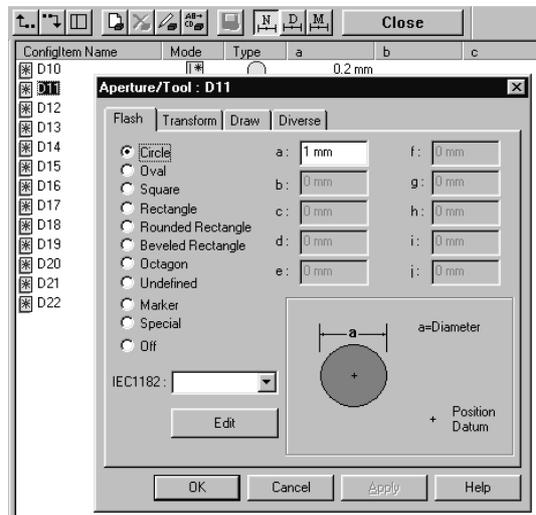
This action will delete this aperture list.

Figure 137:



Double-clicking on **APERTURES** opens the aperture list. Right-clicking on an aperture accesses the properties for the aperture. You may now change the shape or size of the aperture as required.

Figure 138:



By clicking **OK** you quit this dialog box.

- › Click on  to return to the list of the aperture lists.
- › Now highlight the **APERTURES** list and click on  (on the left, next to Close) to save this list to the “**OWN.CAT**“ template file.
- › Click  again.

The list of all format configurations appears.

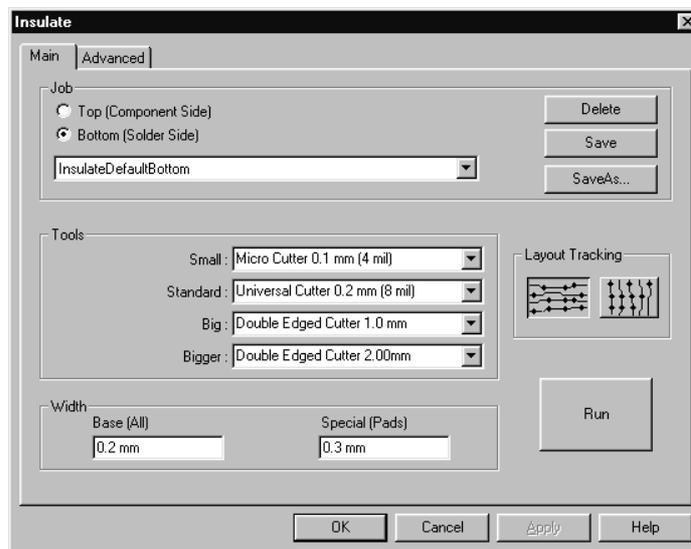
- › Select the entry **configsGerber** and then click  , to store all Gerber aperture lists present and permanently remove the deleted lists from the cat file.

10.7.3 Saving a modified isolation job

A further important point is the definition of your own custom insulation job.

To define an isolation job, open the **ISOLATE** submenu from the **EDIT** menu.

Figure 139:



By way of an example, we will redefine the isolation width for Special and the preferred direction of the conductor tracks for a job on the solder side (bottom).

- › Click on the **SPECIAL** edit field and enter the value 0.7 mm. Then click on the Vertical Conductor Track direction button.

Clicking on **SAVE** saves the changes under the name **“INSULATEDFAULTBOTTOM”** to your CAT file and thus automatically assigns it to the button in the **FRONT-TO-END** toolbar. Or you may click on **SAVE AS**, thus saving a new, additional job to the CAT file.

In both cases, this modification will always be available to you when you start a new project with this CAT file: File-New-own.cat.

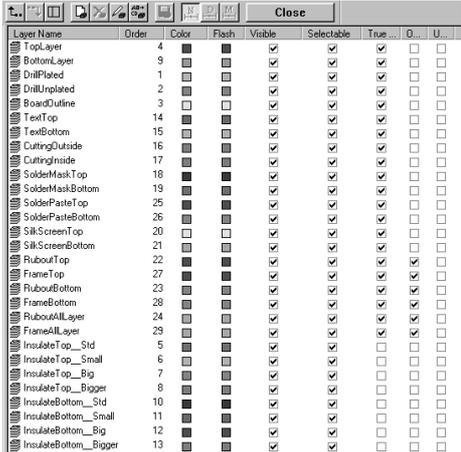
Clicking **OK** quits this dialog box.

10.7.4 Saving a new layer

Layer information, such as new layers or the layer parameters can also be modified and saved to the CAT file.

- › Click on  to open the layer list.

Figure 140:

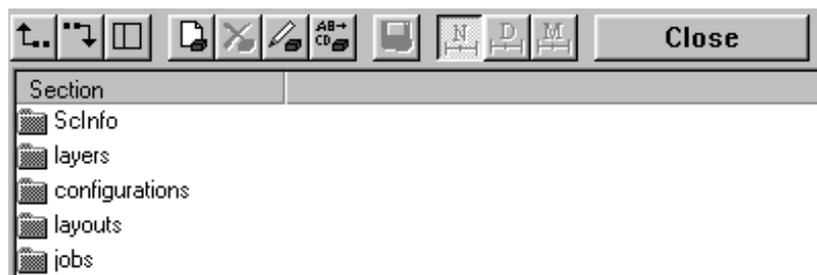


| Layer Name | Order | Color | Flash | Visible | Selectable | True | O... | U... |
|-----------------------|-------|-------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|--------------------------|
| TopLayer | 4 | | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BottomLayer | 9 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | 1 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillInplated | 2 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BoardOutline | 3 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextTop | 14 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextBottom | 15 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| CuttingOutside | 16 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| CuttingInside | 17 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderMaskTop | 18 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderMaskBottom | 19 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderPasteTop | 25 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderPasteBottom | 26 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SilkscreenTop | 20 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SilkscreenBottom | 21 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| RiboutTop | 22 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| FrameTop | 27 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| RiboutBottom | 23 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| FrameBottom | 28 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| RiboutAllLayer | 24 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| FrameAllLayer | 29 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Std | 5 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Small | 6 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Big | 7 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateTop_Bigger | 8 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Std | 10 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Small | 11 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Big | 12 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom_Bigger | 13 | | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

In this list you may now modify the various parameters by clicking on **VISIBLE** or **SELECTABLE**, or even add a new layer.

- › Do this by clicking on  and entering a new name for the layer. Once you have also assigned all the necessary parameters to this layer, click on . This takes you one level higher.

Figure 141:



- › Now highlight the **LAYERS** Entry, and click on the  button to save all the modifications in the layer list to your CAT file.

10.7.5 Saving the parameters

The settings for the parameters under General Settings are very personal and depend on the one hand on your individual preferences and on the other hand on the characteristics of the layout to be processed.

Open the **GENERAL SETTINGS** dialog box by clicking on **GENERAL SETTINGS...** in the **CONFIGURATION** menu. Take a look at all the tabs and your settings.

Detailed information on the various index cards is given in the *CircuitCAM* online help under **GENERAL SETTINGS**.

When you quit this dialog box by clicking on **OK**, all the entries for the **COLORS, UNIT/GRID, METHODS, IMPORT ASSIGNMENTS** and **DIVERSE** tabs are saved to the CAT file on which the document is based.

The entries on the **USER** and **DIRECTORIES** tabs are saved to the Windows Registry as these are user and computer-specific data.



Note: All other changes are always saved to the template file (*.CAT) on which the document (project) is based. *CircuitCAM* will prompt with an appropriate message if the underlying CAT file is unknown.

In this case, modifications you wish to use at a later date must be saved using File-Save As *.cat.

10.8 Prepared isolation jobs

The software supplied contains two prepared isolation jobs. You can find the settings for these jobs in the table below:

| | Job*: | |
|-------------------------------|------------------------------|---------------------------|
| | InsulateDefaultBottom | InsulateDefaultTop |
| <u>Isolation width</u> | | |
| Base | 0.2 mm | 0.2 mm |
| Special | 0.3 mm | 0.3 mm |
| <u>Tool</u> | | |
| Standard | Universal Cutter 0.2 mm | Universal Cutter 0.2 mm |
| Big | End Mill Cutter 1.0 mm | End Mill Cutter 1.0 mm |
| <u>Layer</u> | | |
| Base | BottomLayer | TopLayer |
| Special | - Flashed Pads Only - | - Flashed Pads Only - |
| Rubout1 | RuboutBottom | RuboutTop |
| Rubout2 | RuboutAll | RuboutAll |
| Summery | InsulateBottom | InsulatTop |
| <u>Isolation grid</u> | | |
| Isolation grid | 0.005 mm | 0.005 mm |
| Rubout milling | X-parallel | Y-parallel |
| Primary- overlapping | 80 % | 80 % |

10.9 Creating Drilling/Milling Tools

When working with *CircuitCAM* and the circuit board plotter, it may be necessary to modify present drilling or milling tools, i.e. to change their parameters or define new tools.

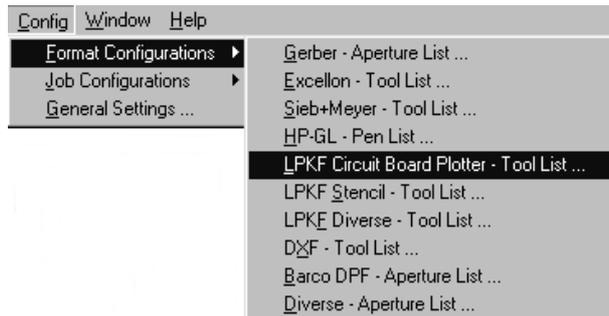
As a matter of principle, we differentiate between two types of tools: Drilling tools and milling tools for routing the contours of the circuit board or structuring the copper laminate.

The tools can be defined in a manner very similar to defining the apertures used for Gerber data. The terminology Flash for drills and Draw for cutters is used in the same manner as for the aperture definitions, because the same program routines are used within *CircuitCAM*.

The various LPKF tool lists are given in the **CONFIGURATION** menu under **FORMAT CONFIGURATIONS..._LPKF CIRCUIT BOARD PLOTTER TOOL LIST**:

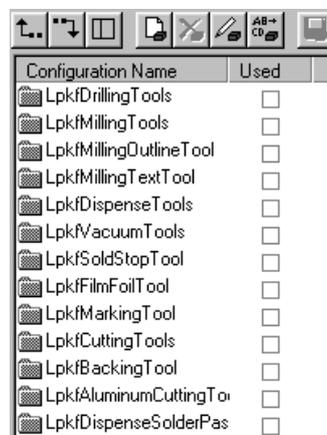
- › Click on **FORMAT CONFIGURATIONS ..._LPKF CIRCUIT BOARD PLOTTER TOOLS LISTS** in the **CONFIGURATION** menu (see illustration below).

Figure 142:



The following list window appears:

Figure 143:



All lists displayed here contain LPKF tools used for rapid prototyping. The most important are:

| | |
|------------------|--|
| LpkfDrilingTools | contains all tools for drilling the circuit board |
| LpkfMillingTools | contains all tools which can be used for insulating |
| LpkfCuttingTools | contains all tools for circuit board contour routing |

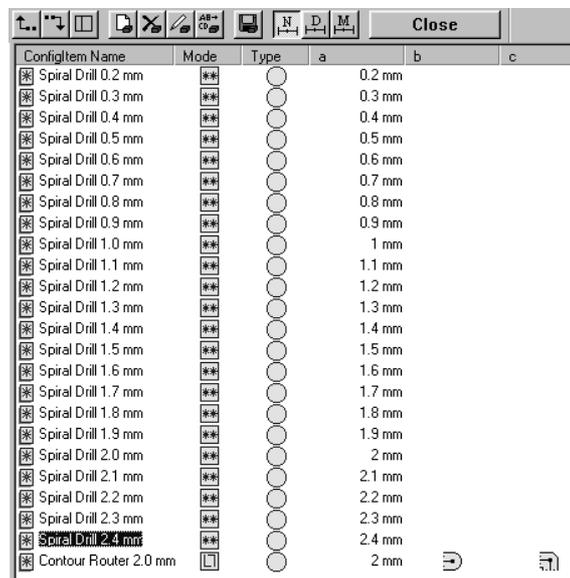
Two examples are given to show how a new drilling tool and new milling tool can be generated. You can also change the parameters for existing tools in the same manner as defining the individual parameters. Definition of the various parameters should be accomplished extremely carefully, because incorrect entries can have effects on the result.

10.9.1 Definition of New Drilling Tool

- › Double click the entry *LpkfDrilingTools*.

The list of drilling tools is opened and the following display appears:

Figure 144:

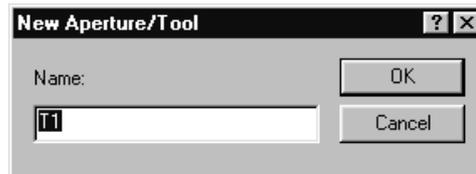


A drilling tool with a diameter of 2.5 mm is to be added to the list as an example.

- › Mark the tool following which the new tool is to be entered and click the button .

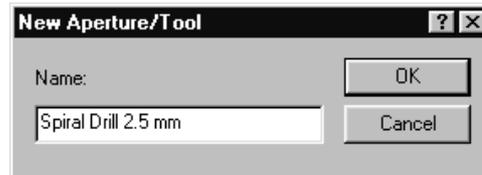
The following window appears:

Figure 145:



- › Enter the name of the new tool:

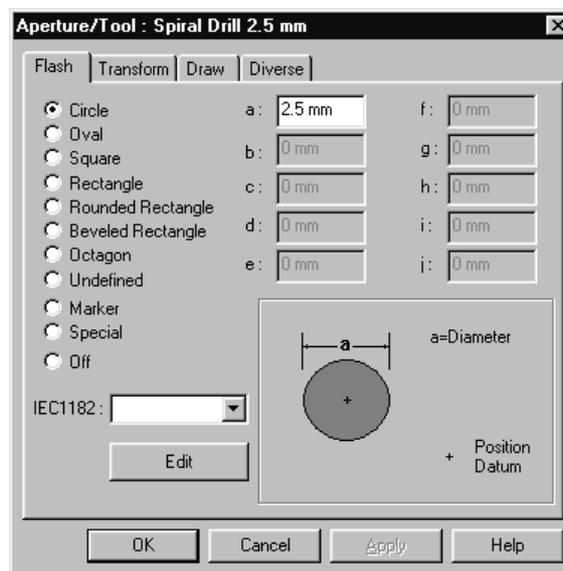
Figure 146:



- › Confirm the entry with **OK**
The new tool appears in the list .

Open the dialog box with 4 index cards for definition of the tool with a right-mouse click. In our example, only the index cards Flash and Diverse are of interest.

Figure 147: Aperture/Tool



The important entries to be made in the Flash index card are the shape of the tool and the diameter. The shape of a milling or drilling tool is always round. Mark the radio button *ROUND* (circle).

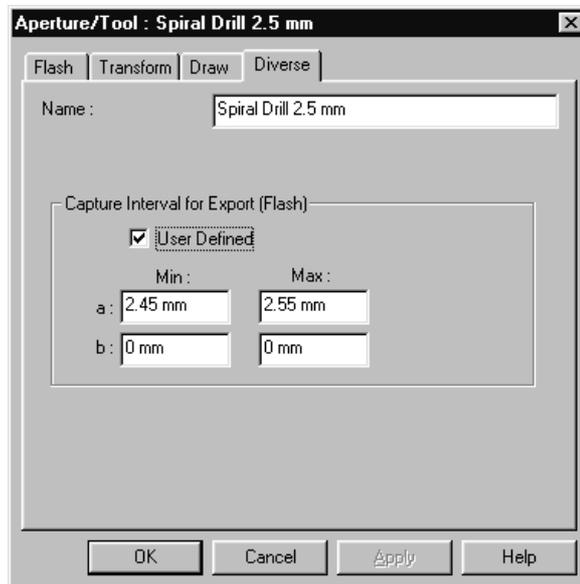


Note: This dialog window is also used for definition of highly varying apertures in Gerber format; for this reason, a large number of different shapes are specified here.

- › Enter the desired drill diameter under a:
- › Click the **DIVERSE** tab and enter the rest of the parameters.

The following window appears:

Figure 148:



This index card again shows the name of the tool. Important here is the entry of a capture interval to be used for the tool.

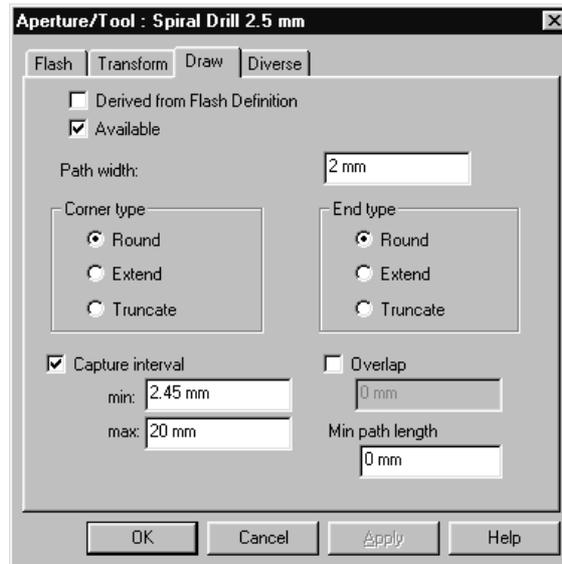
Circular elements on *DrillPlated* or *DrillUnplated* in the layout serve for correct assignment of the drilling tools. When defining the drilling tools, it must be ensured that every diameter is clearly assigned to a tool. Here, it is not permissible for the capture intervals of the individual tools to overlap (unclear assignment) or to have gaps (no assignment possible). (See Chapter „Checking The Tool List“ on page 62).

A capture interval of 2.45 to 2.55 mm is defined for the 2.5 mm drill. This means that all circles with a diameter of 2.45 mm (inclusive) up to 2.55 mm (exclusive) are assigned to this tool.

It must be ensured that the settings for the adjacent tools are adapted to these new entries. In our example this means that the capture interval for the 2 mm contour router (last tool) will have to be checked and corrected if necessary. This milling tool is then set for all holes with a diameter greater than 2.5 mm, i.e. all holes with a diameter greater than 2.5 mm are automatically milled.

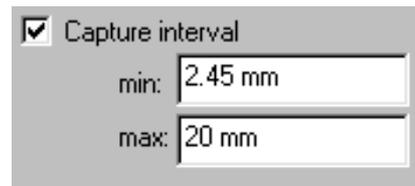
- › Close the dialog window by clicking **OK** and open the dialog window for the *2 mm Contour router* with a right-mouse click on this entry in the list.
- › Select the index card **Draw** in the dialog window.

Figure 149:



Change the values under capture interval according to the definition of the new drilling tool:

Figure 150:



This ensures that the milling tool is only used for holes with a diameter beginning with 2.55 mm. If the capture interval is not adapted, it is not clear for *CircuitCAM* whether the drill or milling cutter is to be used for a diameter of 2.53 mm. This diameter would be assigned to both tools.

- › Close the dialog window after changing with **OK**.

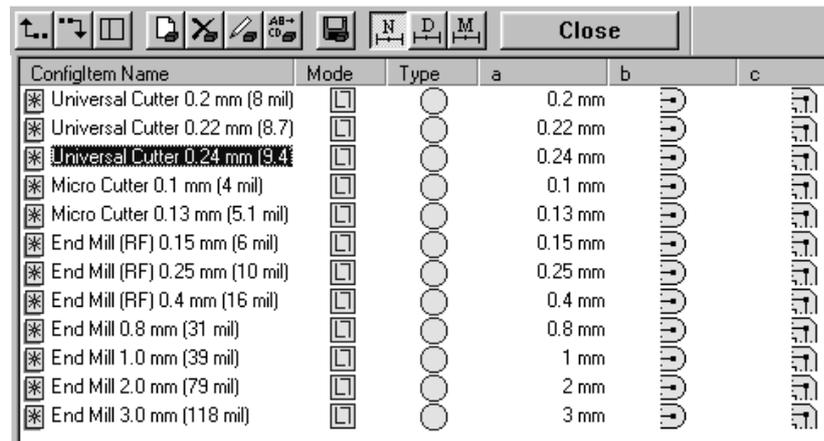
If the change is to remain stored for future layouts, save the new tool list as template file (see “Creating CircuitCAM template file” on Page 91).

10.9.2 Definition of New Milling Tool

- › Double click the entry **LPKF MILLING TOOLS**.

The list of insulating milling tools is opened and the following display appears:

Figure 151:

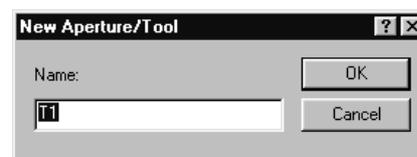


As an example, a new milling tool with diameter of 0.26 mm is to be added to the list.

- › Mark the milling tool, **below** which the new tool is to be added and click the button .

The following window appears:

Figure 152:



- › Enter the name of the new tool:
- › Confirm the entry with **OK**.

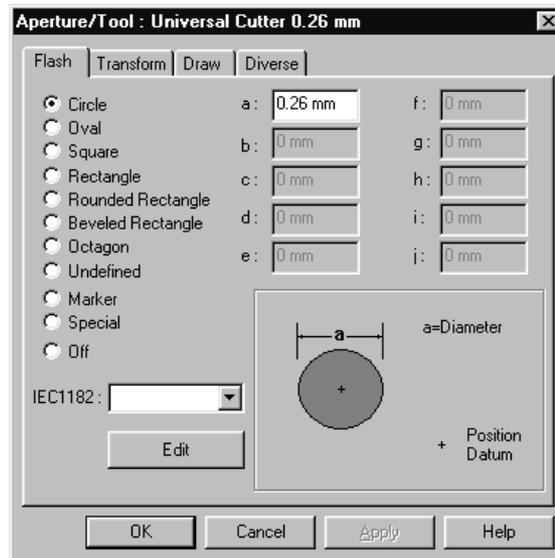
The new tool appears in the list.



Note: If you have defined a tool as described above and want to use it for working with the circuit board plotter, it must be ensured that it is defined with the same name and diameter in the **BoardMaster** tool library (*.TOL) used!

Click the right-mouse key to open the dialog box with 4 index cards for definition of the tool. In our example, only the index cards *Flash* and *Draw* are of interest.

Figure 153:



The important entries to be made in the Flash index card are the shape of the tool and diameter. The shape of a milling tool or drilling tool is always round.

- › Mark the radio button **CIRCLE**.

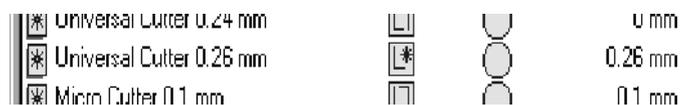


Note: This dialog window is also used for definition of highly varying apertures in Gerber format, for this reason, a large number of different shapes are listed here.

- › Enter the desired milling diameter under a: .
- › Click **OK** to confirm the entry and close the dialog box.

The new tool appears in the list and is defined here as a milling and drilling tool:

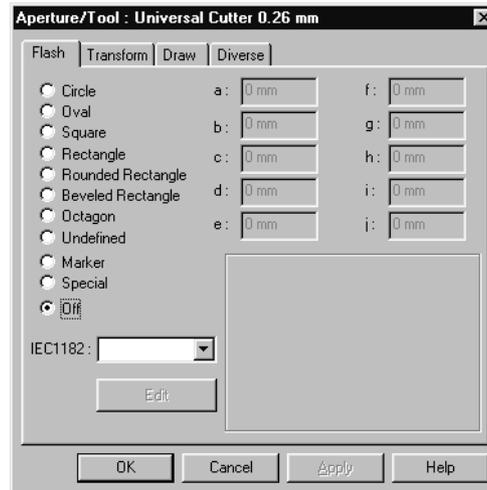
Figure 154:



-  This symbol in the tool list describes the tool as a drilling (Flash) tool as well as milling (Draw) tool.
-  This symbol in the tool list describes the tool as a drilling (Flash) tool.
-  This symbol in the tool list describes the tool as a milling (Draw) tool.

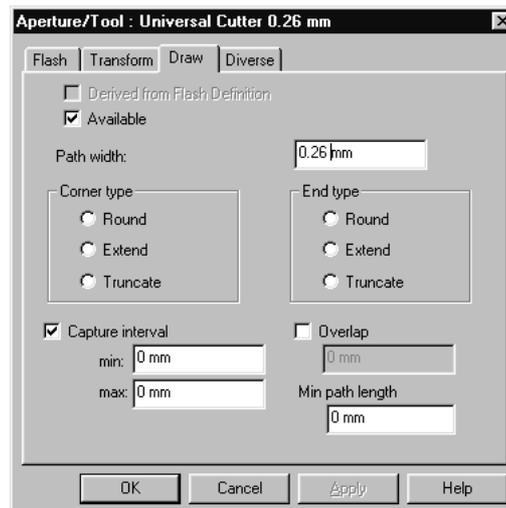
- › Click the tool again with the right-mouse key to define it as a pure milling tool, because a milling tool cannot be used simultaneously as a drilling tool. However, this is quite possible for an aperture inserted in Gerber data.

Figure 155:



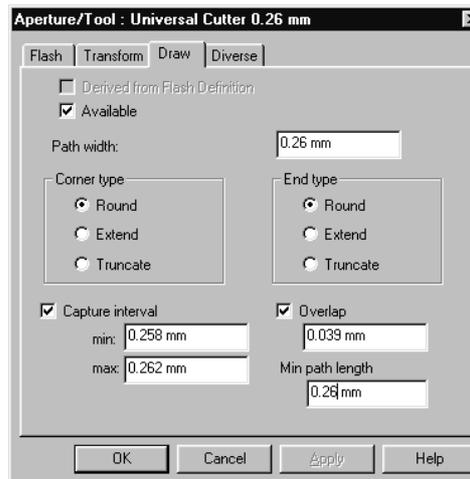
- › Click the radio button **OFF** on the *Flash* index card. This switches off the drilling (Flash) characteristics for the tool.
- › Now click the *Draw* index card. The following box appears:

Figure 156:



The check box **AVAILABLE** is switched on in this window and the tool diameter is entered. Other important parameters for a milling tool are the **CAPTURE INTERVAL**, **OVERLAP** and **MIN. PATH LENGTH**. Enter the values for these 3 parameters as shown in the following illustration:

Figure 157:



For the capture interval, a interval is entered within which this tool is to be used similar to drilling tools. In our example, all lines with a width of 0.258 (inclusive) and 0.262 are realized by a path with this tool.

All lines narrower than this are filled out with a smaller tool. All wider lines are filled out with this tool by cutting a number of paths until the exact width is achieved.

In contrast to the drilling tools, the list of milling tools can have overlaps and gaps within the capture interval. A line located outside a capture interval can still be realized by cutting a number of paths with a smaller tool. This is not possible with drills

The overlap indicates the amount by which parallel cutting paths with the same tool are to overlap without posing the danger of fine copper hairs remaining between the paths. *CircuitCAM* sets this value to 10 % of the tool diameter as a default value (checkbox switched off). We recommend a value of 15 % of the tool diameter. Switch On checkbox and enter value.

The parameter min. path length indicates the minimum length of a cutting path which can be generated during the insulating process. This parameter is particularly important for rubout tools such as end mills. These tools have a hollow shape so that copper could remain when the cutting path is shorter than the tool diameter. We recommend entering a value equal to the diameter of the tool for this parameter.

- › Complete the entry by clicking **OK**.
- › Click **CLOSE** in the list window.

The tool list is closed.



Note: Define the tool parameters carefully to avoid errors when exporting the BoardMaster production data.

10.10 Export Jobs

In addition to the possibility of using *CircuitCAM* as CAM software for circuit board plotters, it can also be used for preparation of production data for the circuit board manufacturer. Generation of multiple boards, ground areas, milling data etc. All these modifications can be exported as Gerber or Excellon data (only with PCB version).

CircuitCAM already has a number of prepared output jobs for the LPKF circuit board plotter:

| Job | Application |
|-------------------------------------|---|
| LpkfCircuitBoardPlotter: | Standard export job for circuit board plotter and all options |
| LpkfCircuitBoardPlotterGalvTHP: | Export job for circuit board plotter recommended for use of a galvanic U contacting system |
| LpkfCircuitBoardPlotterNoTHP: | Export job for circuit board plotter recommended when no through contacting possibility whatsoever is used. |
| LpkfCircuitBoardPlotter+Auto Contac | Export job for circuit board plotter with Auto-Contac Data output |
| LpkfSolderMaskCutting: | Export job for circuit board plotter for cutting Solder mask of the EasySolder system |

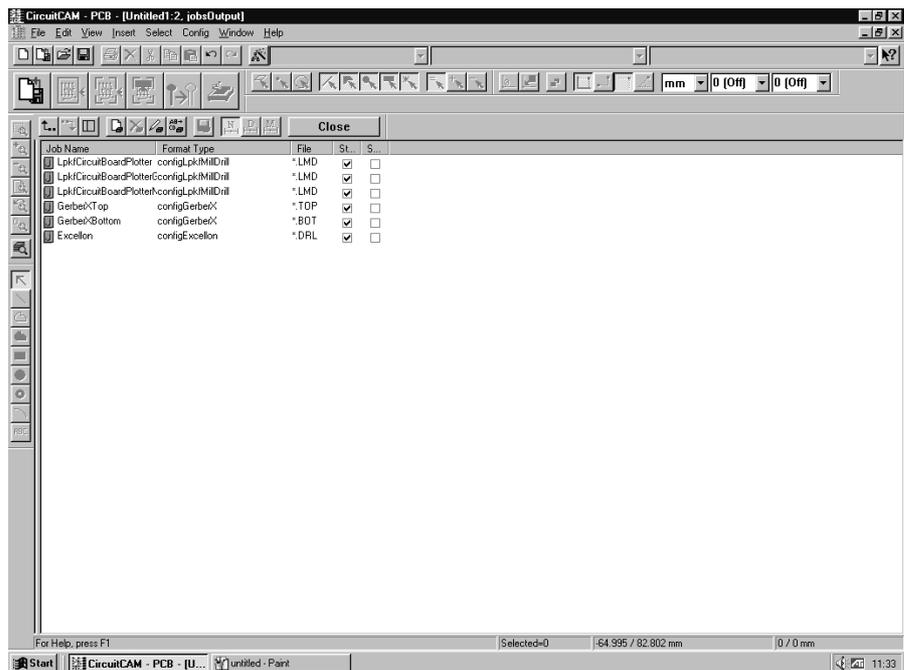
Use one of these jobs to output the production data you have generated and provide it for use by LPKF *BoardMaster* or other software.

The LPKFCircuitBoardPlotter-Job is assigned to the  button in the **FRONT-TO-END** Symbol bar. By clicking this button the production files will be saved in a LMD-file. If *BoardMaster* is open, the LMD-file will automatically be imported into *BoardMaster* and placed on the screen.

In the following you can see, which parameters are defined in the exported job.

- › Select in Menu **CONFIGURATION** in the submenu **JOB** the entry **EXPORT...** to show the listbox with all available export-jobs.

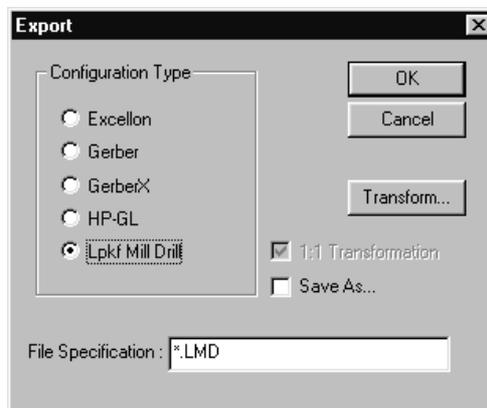
Figure 158: GerberXTOP,- Bottom + Excellon Jobs (only in the PCB-Version)



› Click the entry **Lpkf Circuit Board Plotter** with the right-mouse key.

The following window appears:

Figure 159: Export Job



The format in which the data is to be exported, the name of the exported file and whether the data is to be turned, mirrored or shifted (transformed) during export are defined in this window.

› Click **OK** to close the window.



Note: Click on "Transform" to open the Transform dialog box and modify the origin, scaling and alignment used to output the relevant layer.

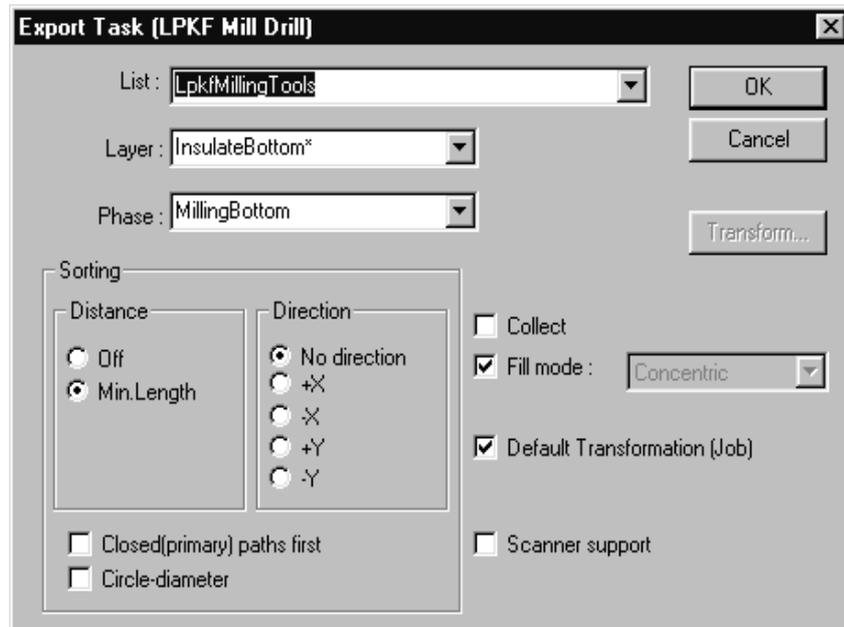
- › Double click the entry “**LPKF CIRCUIT BOARD PLOTTER**” to display the various entries (tasks) with the assigned tool list, layer and production phase:

Figure 160:

| Task | List | Phase | St.. | Fill | P... | C... |
|-------------------|------------------------|-------------------------|-------------------------------------|-------------------------------------|--------------------------|-------------------------------------|
| InsulateTop* | LpkfMillingTools | MillingTop | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BoardOutline | LpkfMillingOutlineTool | MillingTop | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextTop | LpkfMillingTextTool | MillingTop | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| InsulateBottom* | LpkfMillingTools | MillingBottom | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| BoardOutline | LpkfMillingOutlineTool | MillingBottom | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| TextBottom | LpkfMillingTextTool | MillingBottom | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | LpkfDrillingTools | DrillingPlated | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillUnplated | LpkfDrillingTools | DrillingUnplated | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | LpkfDispenseTools | Dispense | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | LpkfVacuumTools | Vacuum | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillPlated | LpkfBackingTool | DrillingBacking | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| CuttingOutside | LpkfCuttingTools | CuttingOutside | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| CuttingInside | LpkfCuttingTools | CuttingInside | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderMask.Top | LpkfSoldStopTool | CuttingSoldStopTop | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> |
| SolderMask.Bottom | LpkfSoldStopTool | CuttingSoldStopBottom | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> |
| DrillPlated | LpkfMarkingTool | MarkingDrills | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| DrillUnplated | LpkfMarkingTool | MarkingDrills | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderPasteTop | LpkfDispenseSolderPast | DispenseSolderPasteTop | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| SolderPasteBottom | LpkfDispenseSolderPast | DispenseSolderPasteBott | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

Now open the following window by clicking the desired job with the right-mouse key:

Figure 161: Export Task (LPKF Mill Drill)



The dialog box "Export Task (LPKF Mill Drill)" contains the following fields and options:

- List: LpkfMillingTools
- Layer: InsulateBottom*
- Phase: MillingBottom
- Buttons: OK, Cancel, Transform...
- Sorting section:
 - Distance: Off, Min.Length
 - Direction: No direction, +X, -X, +Y, -Y
 - Closed(primary) paths first
 - Circle-diameter
- Other options:
 - Collect
 - Fill mode: Concentric
 - Default Transformation (Job)
 - Scanner support

Here you can select the data format in which the job is to be exported.

You can also click on the entry for a line in this list window to open a dialog window where you can customize the export task to meet your requirements.



Note: These modifications should never be saved under the same CAT filename, otherwise you would destroy the original job. Either create new jobs or save any modifications to the original job in a new CAT file, should you need modifications to a job.

A second job is provided for double-sided PCB that are not to be through-hole-plated; this will make processing in BoardMaster simpler and faster:

“**LPKFCIRCUITBOARDPLOTTERNOTHP**” generates an almost identical LMD file. The difference is that the various production phases have been summarized into just two production phases: **MILLINGBOTTOM** and **MILLINGTOP**.

You can save modified or new jobs in the basic *.cat file in the job name list window.

› Mark the job to be saved and click  .

10.11 Batch Function (PCB Version Only)

The batch function allows automatic processing of various *CircuitCAM* functions. This allows the user to generate production data for the circuit board plotter in a simple manner.

Another advantage: infrequent or less experienced users can use CircuitCAM with very little information, such as where and how to start the batch mode.

The following chapter is intended to clarify the operation of the Batch function and put you in a position to generate a batch job customer-tailored to your requirements.

The sequence for a batch job reflects the manual work with the front-to-end tool bar:

- Import all required data
- Calculate circuit board contour
- Insulate the two sides of the circuit board
- Store CAM file and then
- Export the production data in LMD format

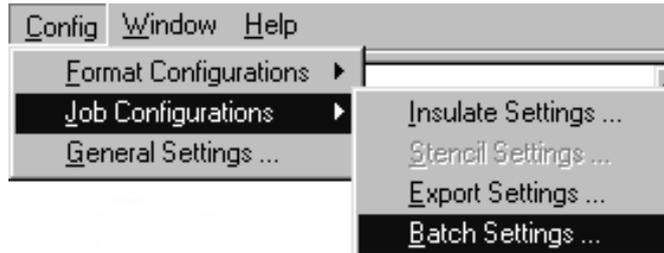
Functions such as modifying data, partial definition of the rubout surfaces, etc. is not possible in the batch mode.



Important: Automatic processing of a batch job is only possible with the default template file DEFAULT.CAT and the Batch-Job DEFAULT.

Definition or modification of a batch job can be accomplished in the list window batch settings in the **Configuration-Job** configurations menu:

Figure 162:



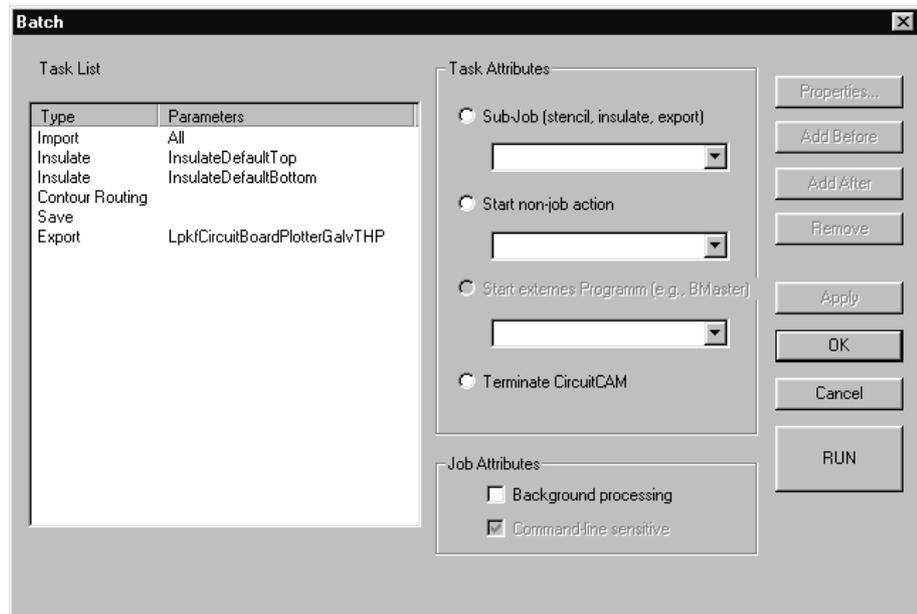
The following list window is opened:

Figure 163:



- › The batch dialog window can be opened by clicking the entry **Default** with the right-mouse key:

Figure 164:



The dialog window is subdivided into 3 areas:

1. The **task list** stores all individual tasks in the sequence to be processed in the batch job from top to bottom.
2. The **Task attributes** serve for selection or definition of the tasks to be performed.
3. With the buttons at the right, tasks can be added, deleted, changes confirmed and to the extent possible, the properties of some tasks described in greater detail.

For definition of a task, it is first necessary to define the type of task by switching on the corresponding radio button:

1. Jobs:

Here, a task from the areas Insulate, Export or Stencil (*CircuitCAM* Stencil Version only) can be incorporated into the batch job. All jobs defined in the areas are available, e.g.:

| | |
|-------------------------|---|
| InsulateDefaultTop | for insulating the solder side |
| InsulateDefaultBottom | for insulating the solder side |
| LpkfCircuitBoardPlotter | for exporting the production data for the plotter |

2. Other tasks:

These are tasks which are not based on a job but rather whose properties are defined in the batch dialog window, e.g. importing data, contour generator and saving data.

3. *CircuitCAM* is closed at the end of the batch job when the radio button **CIRCUITCAM IS CLOSED is clicked. If you want to terminate *CircuitCAM*, this entry must always be the last in the task list. Any messages occurring during the batch run are logged in a text file (*.rep).**

The new task can be inserted in front of or behind the task marked in the task list by clicking **ADD ▲** or **ADD ▼**.

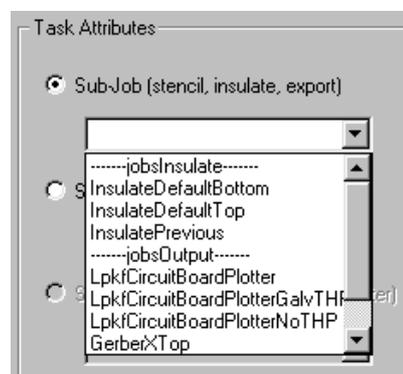
The marked task can be deleted from the task list by clicking the **DELETE** button.

Definition of a new task in the task list.

1. Sub-Job tasks:

- › Mark the task before or in back of which the new task is to be inserted.
- › Click the radio button **JOB**.
- › Select the job to be added in the task list as a new job in the *List Box*.

Figure 165: Task attributes



- › Click the button **ADD ▲** or **ADD ▼**, to insert the new task in front of behind the marked task.

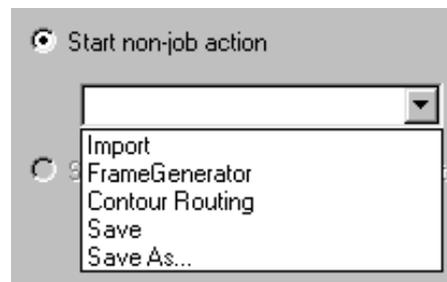


Note: Pay attention to the sequence of tasks in the task list. It is not practical to isolate the data before importing or export the data before isolating.

2. Non-Jobs

- › Mark the task before or in back of which the new task is to be inserted.
- › Click the radio button *Non Jobs*.
- › Select the job to be added in the task list as a new job in the *List Box*.
- › Click the button **ADD** ▲ or **ADD** ▼, to insert the new task in front of behind the marked task.

Figure 166:

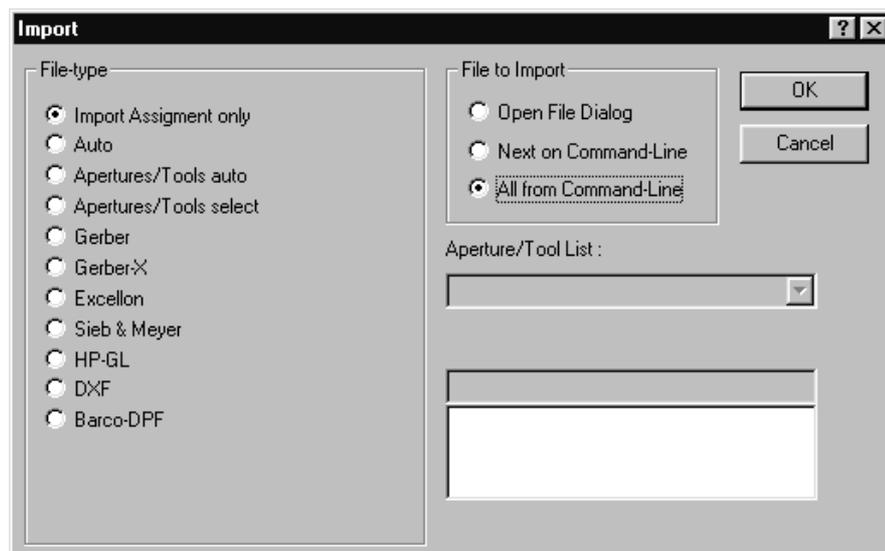


For **Non Jobs**, a dialog window is then opened automatically to define the properties of the job:

Import: Properties

After creating an import task, the following dialog window appears:

Figure 167: Import



It is subdivided into the areas **File type** and **File to import**. The file type defines the file format to be imported or that the import assignment is to be used for importing.

The simplest and quickest method to import production data with the aid of a batch job is to assign the import assignments. This means it is only necessary to define one single import task in the batch job.

Only this possibility is described here, because it is the most suitable possibility for 99 % of all cases.



Important: This assumes that the import assignment for your files to be imported has been defined in the template file **DEFAULT.CAT** (see applicable Chapter in this Manual).

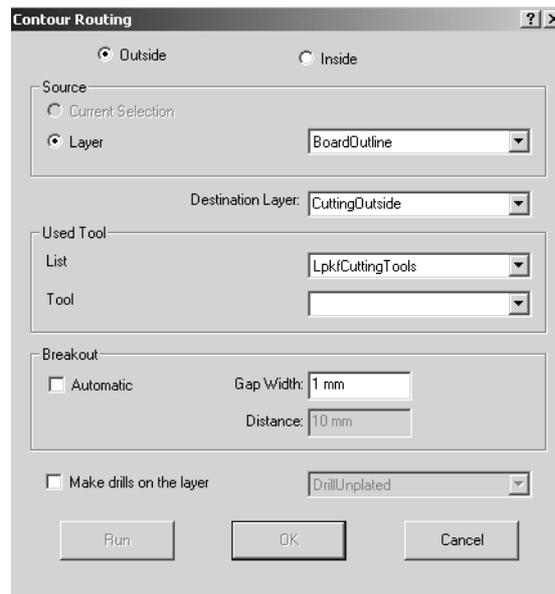
The import task is defined as in the figure above:

- › Switch on the radio button **IMPORT ASSIGNMENT ONLY**.
This causes CircuitCAM to import all files defined in the import assignments
- › Switch on the radio button **ALL FROM COMMAND LINE**.
This selects all files from the command line (see "Execution of batch job").
- › Click **OK**.
The dialog window closes and all settings for this import task are saved in the task list.

Contour Routing:

After creating a contour routing task, the following dialog window appears:

Figure 168: Contour Routing



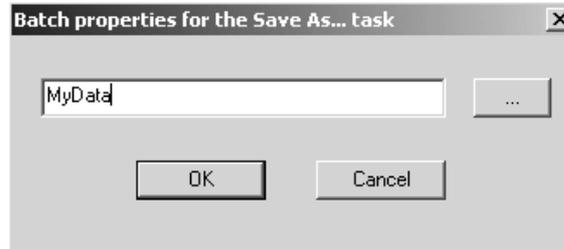
Here you can see the contour routing settings already familiar from manual work. You can make the settings according to your desires or requirements.

Click **OK** to close the dialog window and save all settings for this contour routing task in the task list.

Save As...

After calling the Save as... task, the following dialog window appears

Figure 169:



- › Enter you the name want to assign to the project in the text box.



Note: If you use the Save task, the name of the file imported first is automatically used for your project (without name extension).

Press **OK** to close the dialog window and save all settings for this contour routing task in the task list.

Executing a batch job:

3 different possibilities are available for executing a batch job:

1. Move the files to be imported to the *CircuitCAM* button on the desktop with Drag&Drop.
Select all files to be imported in Microsoft Explorer or a similar file manager program and pull these files to the *CircuitCAM* button on the desktop.

2. By double-clicking on a button on the desktop. The command line for the button must contain all required parameters such a file name and path.

e.g. : C:\LPKF50\CCAM\CircuitCAM.EXE
C:\LPKF50\DATA\Sample.TOP C:\LPKF50\DATA\Sample.BOT.....

Disadvantage: If you do not want to change the command lines for the button each time, the files to be imported must always have the same names and the same name extension and the files must always be located in the same directory.

3. By clicking the Run button in the batch dialog window in *CircuitCAM*. This method of executing a batch job is suitable primarily when all files have already been imported and all subsequent processes such as the contour generator, insulating and exporting are to be performed automatically. In this case, the name assigned to the template file or the batch job is irrelevant.

The file **DEFAULT.CAT** included with the program already contains a batch job with the name **DEFAULT**. This batch job is prepared for the Drag&Drop run possibility.

Caution: It is still necessary for you to set your specific import assignment.

10.12 Processing AL-front panels (PCB version)

This aluminium processing is based on sample data. It is only meant to show the procedure by means of an example. If you have to process your own front plate, then you need to replace the sample co-ordinates with your own data.



Note: Key labels and menu terms are shown in bold type.



Note: Keep exactly to the working steps described below, in order to avoid processing disruptions and breakdowns in functions.

Notes on working with aluminium

- Harder aluminium alloys are better suited to processing than soft alloys.
- The two-fluted end mill is a particularly suitable tool, given its good splinter removing properties.



Caution! The milling depth per pass must not be more than half the diameter of the working tool.

- › Start *CircuitCAM* from the Windows start menu or by double-clicking on the *CircuitCAM* icon on the desktop.

If your assistant „Wizard“ is still active, close it and change to the regular *CircuitCAM* graphics display.

10.12.1 Prepare front plate outline data

First determine the size of the front plate.

- › To do this, select layer **BOARDOUTLINE** from the layer list field. The front plate in the example has the dimensions 100 mm x 130 mm.

- › Click on the button  "Rectangle" in the **Graphics** toolbar (figure 170).

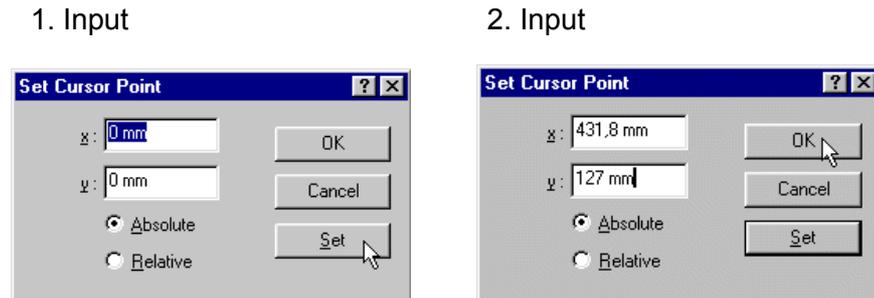
Figure 170: Graphics toolbar



- › Select **EDIT, SET CURSOR** from the menu toolbar (figure 171) or press the keys **SHIFT + C**.

The following window appears:

Figure 171: Setting the cursor



- › The following details have to be entered into the corresponding window:

x= 0 x= 431.8
y= 0 -> **SET** y= 127 -> **SET** and then **OK**.

A yellow rectangle is displayed on the screen.

- › Press **ESC** on your keyboard, to switch off the tool.
- › Press the key **HOME** on your keyboard in order to have an overview of the drawn rectangle.

The filled rectangle which is still represented by a polygon must now be converted into an empty rectangle.

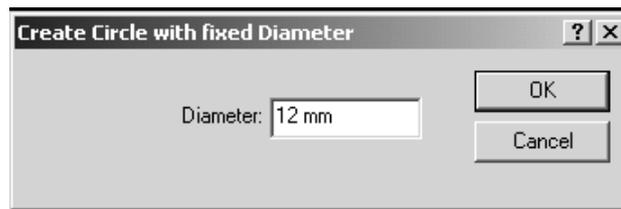
- › Mark the rectangle by clicking on it.
The rectangle is now displayed in a light yellow.
- › Go to the **EDIT** menu bar under **DIVERSE** and select function **CONVERT TO CLOSED OUTLINE PATH** or use the shortcut **SHIFT + W**.

The previously filled rectangle changes to a closed rectangular box which describes the dimensions of the frontplate.

10.12.2 Preparing cutting and drilling data

- › Select the layer **BOARDOUTLINE** from the layer list field.
- › Click on the button  "Circle" in the **Graphics** toolbar.
- › Enter a diameter of 12 mm (following figure) and close the window by clicking on **OK**.

Figure 172: Diameter



- › Now position this object on the front panel of the coordinate (x= 50mm, y= 30mm). To do this, use function **SET CURSOR POINT** by pressing key combination **SHIFT + C** and enter the mentioned coordinates.
- › Press **ESC** on the keyboard.
- › Use the same procedure to place two more circles with 5 mm diameter on the coordinates
 1. circle x= 20 mm, y= 65 mm
 2. circle x= 20 mm, y= 75 mm.

The filled circles must be converted to empty circle representations, as it was previously done with the rectangle.

- › To do this, mark all circular objects using the mouse and convert them using **EDIT/ DIVERSE/ CONVERT TO CLOSED OUTLINE PATH** (or use **SHIFT + W**).

Now place a rectangular aperture of the size 60 mm x 30 mm on the starting coordinate x= 20 mm, y= 90 mm.

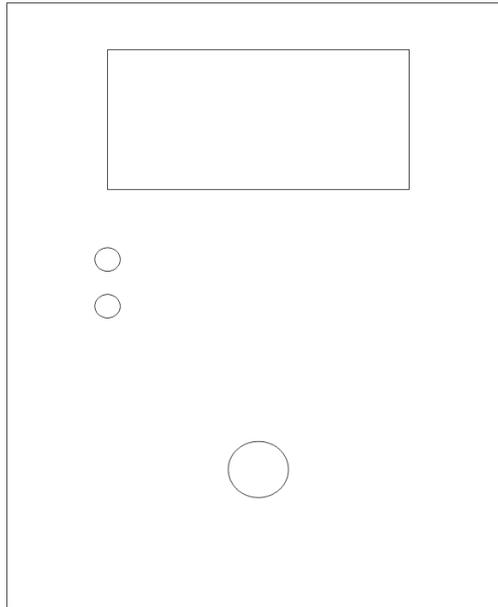
- › Select layer **BOARDOUTLINE** again.
- › Click on the  button and enter the starting coordinates (x= 20 mm, y= 90 mm) by pressing **Shift + C** on the keyboard. Then click on **SET**.
- › To determine the size of the rectangle, add the starting coordinates to the desired size of the rectangle and enter these data.

$$x = 80 \text{ mm } (60 \text{ mm} + 20 \text{ mm})$$

$$y = 120 \text{ mm } (90 \text{ mm} + 30 \text{ mm})$$
- › Click on **SET** and then on **OK**.
- › Press **ESC** on the keyboard.
- › Then convert into an empty rectangular object using Marking with mouse and pressing **SHIFT + W** on the keyboard.

The result should look like this:

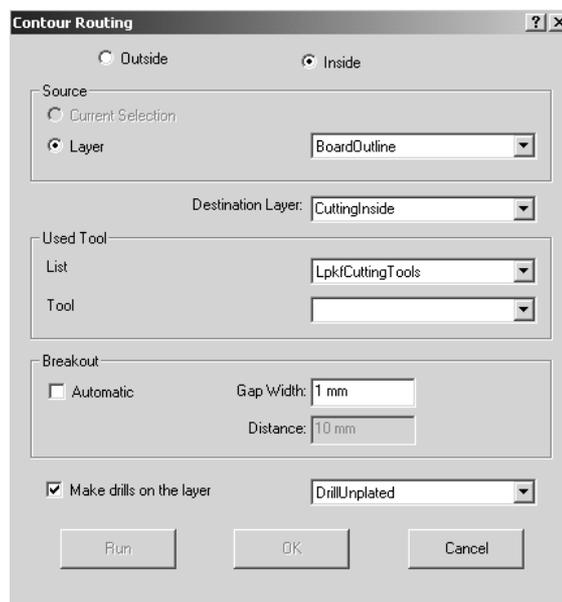
Figure 173:



Now use the possibilities of the contour generator to create the milling data for the breakouts.

- › Mark the 4 objects with the mouse by opening a rectangle.
- › Go to the menu **EDIT** and open the **CONTOUR ROUTING** dialog and select the settings as in the following figure.

Figure 174: Contour milling, dialog



- › Start the process by pressing on **START**.

Following this, there are milling lines and start drill holes created for all breakouts. The start drill hole provides the optimum and gentle start point for the subsequent milling process.

10.12.3 Setting text data

Starting with *CircuitCAM* Version 5.0 all fonts (TTF and TTC) installed on the computer can be used for writing texts.

To be better able to adjust the text which follows, it is advisable to switch on a **Display** and / or **Cursor grid** from the **Grid / Units** toolbar (following figure).

Figure 175: Grid/Unit toolbar



- 1- Display Grid
- 2- Cursor Grid

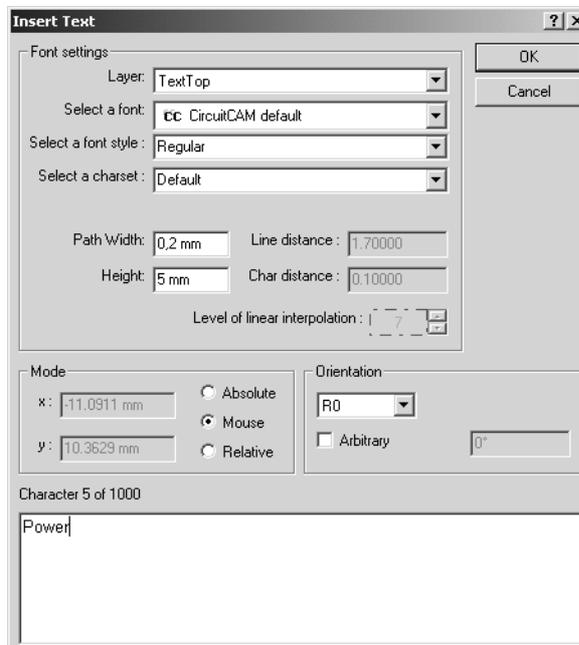


Note: The display grid and the cursor grid should be set to the same value.

- › Select the text program by clicking on  from the **Graphics** toolbar.

The following window appears:

Figure 176: Entering text

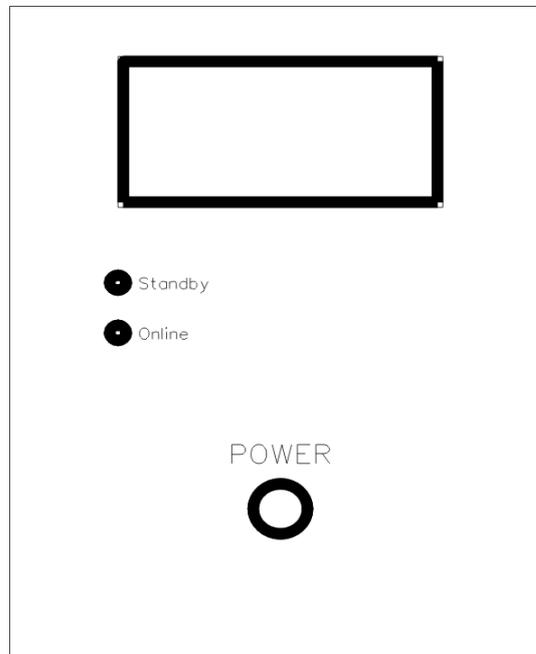


- › Choose all settings as in figure 176 and quit this dialog with **OK**.
- › To position the text, use the mouse and place it above the big round drill hole, then click on the left mouse key.

Repeat the process for labeling the small drill holes. Try to put the text „Standby“ in a letter height of 3mm at the drill hole at the top and the text „Online“ at the drill hole at the bottom.

The finished result might look like the following:

Figure 177: Result



10.13 Exporting and saving

- › Click on **SAVE AS...** in the menu toolbar and save the object under a new name.
- › Select **FILE, EXPORT, LPKF, LPKFCIRCUITBOARDPLOTTER** from the menu toolbar.

10.14 Preparing data for fiducial recognition

Circuit board plotters that are provided with a camera system for fiducial recognition must be given the required positioning information via *Circuit-CAM*. Well utilizable fiducials are simple round drilled holes of a diameter of 1.5 mm.

After having finished the insulation of your layout, you then place the fiducial holes.

- › First set the cursor and screen grid to an appropriate value.
- › Then select the layer **FIDUCIAL** from the layer list field.

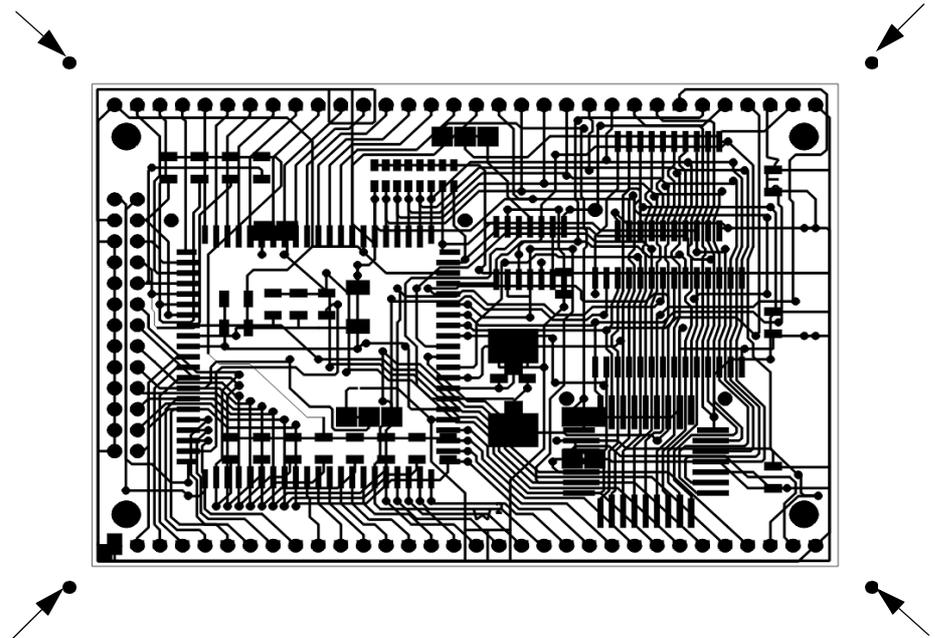
- › Click on the  button in the **GRAPHICS** symbol bar and enter the diameter of 1.5mm. Set the four fiducial circles at right angles around the layout.



Note: Using the cursor display BIG CROSS makes it easier to position the circles at right angles.

The result should look like the following:

Figure 178:



The four circles around the layout represent the fiducial points (see arrows in figure 178).

- › Use the corresponding output job to put out the data als LMD file.

10.15 Producing step-and-repeats (Only PCB Version)

CircuitCAM can be used to create step-and-repeat directly on the CAM file.

Having finished all working processes (insulating, contour generation and so on) at your layout, you can create the step-and-repeat.

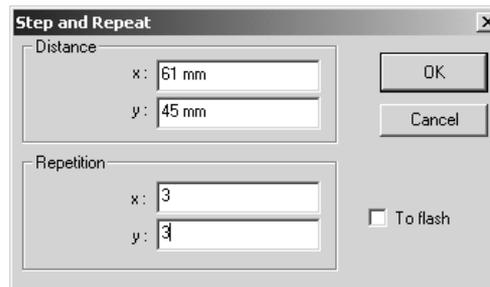
- › First mark all objects of all layers by using the  button in the **MARKING** symbol bar.

All objects are now displayed in light color.

- › Go to **EDIT / DIVERSE** and open the **STEP AND REPEAT** dialog or press **SHIFT + R** on the keyboard.

The following figure is displayed:

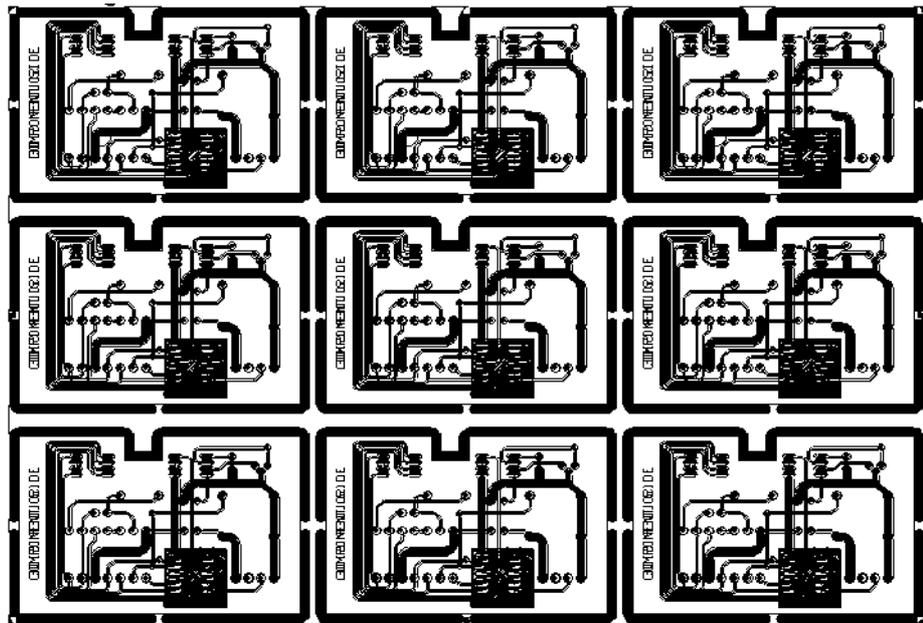
Figure 179: Step and Repeat Dialog



- › Add the desired distance to the value already entered and enter that value under **DISTANCE**.
- › Then enter the desired number of repeats.

Quit this dialog with **OK**. The result could look like the following figure:

Figure 180:



10.16 CircuitCAM 5.0 Hotkeys

| Key Combinations | Functions | Reference |
|-----------------------------|-----------------------------------|-------------------|
| General: | | |
| F1 | Help about current subject | |
| Ctrl + Tab | Toggle within CircuitCAM windows | |
| Ins | View - Previous View | |
| Home | View - Overview | |
| End | View Pan | Cursor Position |
| Page up | View - Zoom In | Cursor Position |
| Page down | View - Zoom Out | Cursor Position |
| Del | Delete | Selected elements |
| Esc | Cancel | |
| Cursor keys | Move Grafic: up, down, left right | |
| Ctrl+Shift+Left Mouse Klick | Set Anchor Point | Cursor Position |
| | | |
| File Menu: | | |
| Ctrl+N | New Graphic Window (default.cat) | |
| Ctrl+O | Open | |
| Ctrl+N | Save | |
| Ctrl+P | Print | |
| | | |
| Edit Menu: | | |
| Ctrl+Z | Undo | Last Action |
| Ctrl+Y | Redo | Last Action |
| Ctrl+X | Cut | Selected Elements |
| Ctrl+C | Copy | Selected Elements |
| Ctrl+V | Paste | Clipboard |
| Ctrl+Space | Duplicate | Selected Elements |
| Space | Move | Selected elements |
| Shift+C | Set Cursor | Enter Coordinate |
| Shift+A | Set Anchor Point | Enter Coordinate |
| Shift+N | Set Zero Point | Anchor Point |
| Shift+L | Move Lyer to Zero Point | Anchor Point |
| Shift+X | Measure | Anchor Point |
| Shift+O | Properties | Selected Element |
| | | |
| Curve Menu: | | |
| Ctrl+A | Add Point | Anchor Point |
| Ctrl+U | Cut point/Segment | Anchor Point |
| Ctrl+D | Delete Point/Segment | Anchor Point |
| Ctrl+M | Move Point/Segment | Anchor Point |
| Ctrl+E | Move/Create Arc | Anchor Point |

| | | |
|--|-------------------------------|---|
| Ctrl+G | Create Gab | Anchor Point |
| Ctrl+R | Inverse Direction | Selected Elements |
| Ctrl+B | Combine | Selected Elements |
| Ctrl+K | Connect to polygon | Selected Elements |
| | | |
| Diverse Menu: | | |
| Shift+Q | Close Curve | Selected Elements |
| Shift+P | Path to Polygon | Selected Elements |
| Shift+Y | Object to Polygon | Selected Elements |
| Shift+W | convert to close Outline Path | Selected Elements |
| Shift+K | Convert to Circle | Selected Elements |
| Shift+Z | Center point | Selected Elements |
| Shift+M | Mass Point | Selected Elements |
| Shift+R | Step & Repeat | Selected Elements |
| Shift+G | Create Bounding Box | Selected Elements |
| | | |
| Change Order Menu: | | |
| Ctrl+J | Move to Next position | Selected Element |
| Ctrl+I | Move to Prev. Position | Selected Element |
| Ctrl+F | Move to First Position | Selected Element |
| Ctrl+L | Move to Last Position | Selected Element |
| | | |
| Configuration - GeneralSetting Methods: | | |
| Temp.A+Left Mouse Klick & Drag | Add/Append Point | Active if cursor changes its shape above selected element |
| Temp.E+Left Mouse Klick & Drag | Create / Move Arc | Active if cursor changes its shape above selected element |
| Temp.U+ Left Mouse Klick | Cut Point / Segment | Active if cursor changes its shape above selected element |
| Temp.M+Left Mouse Klick & Drag | Delete Point / Segment | Active if cursor changes its shape above selected element |
| Temp.L+Left Mouse klick | Select only | Active if cursor changes its shape above selected element |

10.17 Updating Translation File

A translation file serves to convert the aperture list (ASCII file) of any CAD system so that it can be read into *CircuitCAM* without further modification. This allows users who continuously have to work with different aperture lists to work considerably faster, because manual definition of the individual apertures is eliminated. Moreover, they can use the translation files for converting tool lists in Excellon or Sieb&Meyer format.

CircuitCAM already has a series of prepared translation files for the common CAD systems; nevertheless, it may be necessary to have to adapt a translation file to your specific requirements.



Note: The translation files must have the name extension **.txt** .



Note: The translation files are located in the *CircuitCAM* subdirectory “Ape_Templates” or in the directory you specify for the aperture/tool templates in *CircuitCAM* under **GENERAL SETTINGS**.

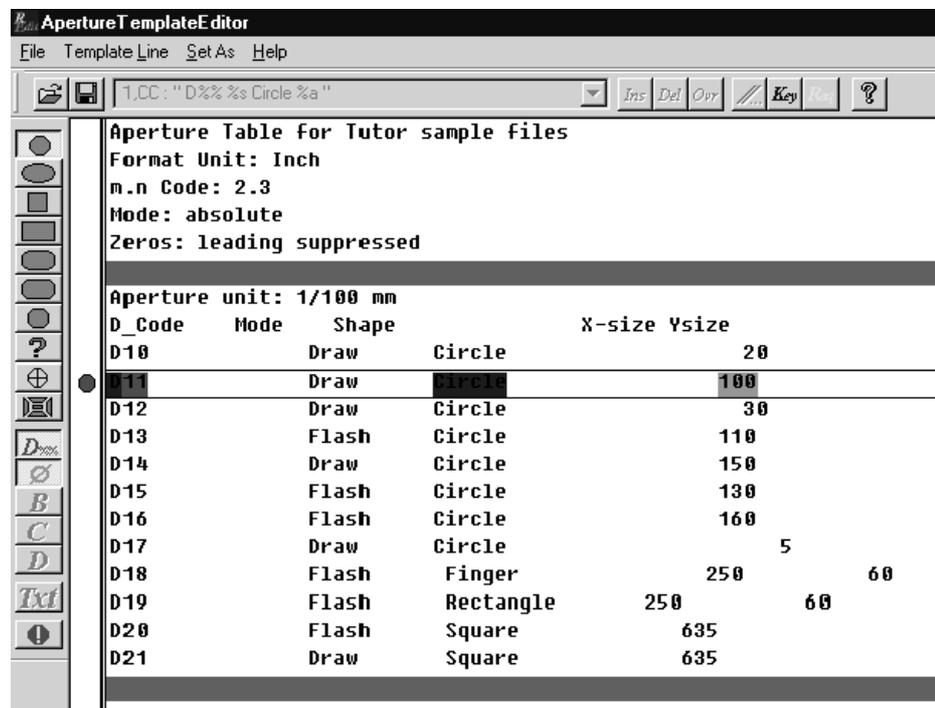
10.17.1 Structure of Translation Files

Translation files are files in ASCII format you can edit with any text editor. For example, Windows provides the text editors **EDITOR** and **WORDPAD**.

However, the use of the program Aperture Template Editor supplied *CircuitCAM* Version 5.0 is simpler than using a text editor.

This program has a graphic, syntax-oriented surface and avoids typing errors such as can occur when a text editor is used. The Aperture Template Editor button was placed in the Start menu and on the desktop during installation of *CircuitCAM*. Information on use of this program is given in its online help.

Figure 181: Aperture Template Editor



A translation file is subdivided into three sections:

- Permanently defined parameters
- Aperture definition
- Variable parameters



Note: The permanently defined and variable parameters need to be defined in the translation file only when the format and aperture definition in the aperture list do not correspond to the default settings.

Section 1: Permanently Defined Parameters

The following table is a list of the permanently defined parameters available with possible parameter values and the corresponding default setting.

| Parameter name | Parameter value | Standard setting for parameter value |
|--|---|--------------------------------------|
| APE_SCALE | <real number> | 1 |
| APE_UNITS | INCH MIL MM | MIL |
| COMMENT | <Sign> <Sign pair> [<Sign> <sign pair>] | |
| CONFIGURATION | ConfigExcellon ConfigGerber ConfigHpgl ConfigSiebMeyer | ConfigGerber |
| COOR_MN | <number>.<number> | 2.3 |
| COOR_MODE | ABSOLUTE INCREMENTAL | ABSOLUTE |
| COOR_UNITS | INCH MM | INCH |
| COOR_ZEROS | DECIMAL LEADING TRAILING | TRAILING |
| KEY_STRING | <String> | |
| REQUIRED_STRING | <String> | |
| Sign declaration: Alternative (or) [...] 0 or multiple repetition | | |

APE_SCALE

If the units for the aperture size do not coincide with any of the possible parameters under **APE_UNITS**, it is possible to scale the units with these parameters. Example: The aperture sizes are given in 1/100 mm. Set **APE_UNITS** to **MM** and **APE_SCALE** to 0.01.

APE_UNITS

Defines the units for the aperture size within the aperture list (MIL, INCH or MM).

COMMENT

Defines one or more characters preceding a comment line. A number of characters/character pairs are separated from one another by space. Example: Comment: **COMMENT @ # //** means that each line beginning with “@”, “#” or “//” is a comment line.

CONFIGURATION

Defines the format defined for this translation file (Gerber, Excellon, HP-GL or Sieb&Meyer)

COOR_MN

Defines the M.N-Code for the Gerber format. If the Gerber data is always generated with the same number of digits in front of following the decimal point, the corresponding values are set here.

COOR_MODE

Defines the coordinate mode for the Gerber format. For Gerber data with absolute values, **COOR_MODE ABSOLUTE** is set, for relative values, **COOR_MODE INCREMENTAL** is set.

COOR_UNITS

Defines the unit for the Gerber data in inches (**INCHES**) or metric (**MM**).

COOR_ZEROS

Defines the setting for leading zeros in the Gerber format. If the Gerber data is always generated with leading zeros, **LEADING** is set here, if it is generated with trailing zeros, **TRAILING** is set here. If leading as well as trailing zeros are output, either **TRAILING** or **LEADING** is set.

KEY_STRING

(Only for automatic recognition)

Defines the key word which must be contained in the aperture list so that this translation file is used for conversion of the aperture list. If the key word coincides, the scan procedure is terminated immediately.

REQUIRED_STRING

(Only for automatic recognition)

Defines the keyword which must be contained in the aperture list for this translation file to be taken into closer consideration during automatic recognition. Even when the keyword matches, the scanning operation is continued to find the best suitable translation file. All translation files without this keyword are ignored immediately during the scanning operation. This allows the range of translation files which could be used to be limited.

Section 2: Aperture Definition

Each aperture within the aperture list must be capable of being assigned precisely to the translation file by one of the entries in the aperture definition. The apertures differ in terms of the D-code, mode, shape and size. Each line in the aperture list is compared with entries in the translation file for matching. If a match is found, the aperture information is set, otherwise the corresponding aperture is ignored.

The following syntax is used for the aperture definition:
 1,<DRAW_FORMAT><FLASH_FORMAT>:"<APE_FORMAT>"

| Parameter <DRAW_FORMAT> | Definition |
|----------------------------|--|
| N | Aperture is only used as a flash aperture, never as draw aperture. |
| C | Aperture is used as round or circular draw aperture. |
| Q | Aperture is used as square draw aperture. |

| Parameter <FLASH_FORMAT> | Meaning | Dimension Parameter in <APE_FORMAT> |
|-----------------------------|---|---|
| N | Aperture is only used as a draw aperture, never as flash aperture. | |
| C | Aperture is used as round or circular flash aperture.. | %a (Durchmesser) |
| Q | Aperture is used as square flash aperture. | %a (side length) |
| R | Aperture is used as rectangular flash aperture. | %a (side length X) %b (side length Y) |
| O | Aperture is used as an oval or finger-shaped flash aperture. | %a (side length X) %b (side length Y) |
| T | Aperture is used as an octagonal shaped flash aperture. | %a (diameter)er) |
| D | Aperture is used as rectangular flash aperture with rounded corners. | %a (side length X) %b (side length Y) %c (corner radius) |
| B | Aperture is used as rectangular flash aperture cut corners. | %a (side length X) %b (side length Y) %c (cut length) |
| S | Aperture is used as special flash aperture. The actual form must be designed and implemented in <i>CircuitCAM</i> . | |

| Parameter <APE_FORMAT> | |
|--|--|
| <p>The <APE_FORMAT> parameter is shown in quotation marks (""") and defines the exact reproduction of each aperture (aperture line) in the aperture list. Both fixed and variable entries may be used here. The space is used as the default separator between the various parameters.</p> <p>.</p> | |
| Variable entries | Meaning |
| %% | is used for the aperture code |
| %s | ignores a string from space to space (may be used as often as necessary) |
| %a %b | are used for size parameters (diameter, side length) |
| <p>Fixed entries</p> <p>All strings appearing between the variable values in the aperture list are permitted as fixed entries. Multiple consecutive spaces in the aperture list are represented as a single space in the translation file.</p> <p>Tip: All entries after the last relevant parameter can be ignored. (There is no need to use the %s variable for this.)</p> | |

Example 1

Information in aperture list:

```
1 D12 Draw circle 8 8 Aperture No.12
2 D20 Flash rectangle 20 60 Aperture No.20
```

Matching definition in the translation file:

```
1,CN:"%s D%% Draw circle %a"
1,NR:"%s D%% Flash rectangle %a %b"
```

Example 2

If the separating character between the individual parameters is not a space, the corresponding character is given in brackets []

Information in aperture list:

```
12 Flash Oval X0.6Y1.20
20 Flash Rectangle X1.0Y0.7
```

Matching definition in the translation file:

```
1,NO:"%% Flash Oval [X]%a[Y]%b"
1,NR:"%% Flash Rectangle [X]%a[Y]%b"
```

Section 3: Variable Parameters

Some CAD systems provide the possibility of writing more precise specifications in the Gerber format into the aperture list in addition to the aperture information. Variable parameters can be used for evaluation of this information to eliminate having to readapt the translation file each time this additional information is changed.

After conversion, a search is made in the aperture list for the character chains defined for the variable parameters and when a match is found they are assigned to the corresponding parameter.

| Variable parameters | Meaning |
|---|---|
| Coordination-unit | |
| 100 : "<text>" | The rsgistered string <text> defines the coordination-unit INCH. |
| 101 : "<text>" | The registered string <text> definesthecoordination-unit MM. |
| Coordination-mode | |
| 110 : "<text>" | The registerd string <text> defines the coordination-mode ABSOLUTE. |
| 111 : "<text>" | The registered string <text> defines the coordination-mode RELATIVE. |
| Before- and after-comma postitions | |
| 120 : "<text>%d<text>" | The registered string <text> defines the M-Code (Number of before comma positions). &d presents the sign from the aperture list - M code. |
| 121 : "<text>%d<text>" | The registered String <text> defines the N-Code (number of the after-comma positions). &d presents the sign from the aperture list N-Code |
| 122 : "<text>%1d.%1d<text>" | The registered string <text> defines the M.N-Code (number of the after- comma positions). The first variable &1d presents the sign who defines the aperture list M-Code. The second variable &1d presents the sign who defines theaperture list N-Code. |
| Koordinate-seros | |
| 130 : "<text>" | The registered string <text> defined the coordination-zeros as LEADING. (trailing suppressed.) |
| 131 : "<text>" | The registered string <text> defines the coordination-zeros as TRAILING. (leading suppressed) |
| 132 : "<text>" | The registered string <text> defines the coordination-zeros as DECIMAL. |

Example 1

Information in aperture list:

```
The units in the Gerber data are metric
m.n format = 3.2
Values absolutes
Leading zeros suppressed
```

Matching definition in translation file:

```
101 : "The units in the Gerber data are metric"
110 : "Values absolute"
122 : "m.n format = 1%d.1%d"
131 : "Leading zeros suppressed"
```

10.17.2 Tips for Adapting Translation File

If the graphic display of the circuit board does not coincide with the desired result, incorrect conversion of the aperture/tool list by the corresponding translation file is usually the cause.

For example if conductors pads are displayed in the wrong shape or size, the aperture/tool definitions in the translation file do not correspond to the format of the aperture/tool list to be read in. As a remedy, the translation file can be adapted specifically to the aperture/tool list as described previously.

However, frequently only one parameter value is incorrectly defined in the translation file so that this problems can be solved quickly. The following table provides a list of the parameters which could cause erroneous graphic display:

Example of translation files for various CAD systems

| Incorrect display in the graphic window:: | Incorrect setting in the translation file: | Parameter in the translation file to modify: |
|--|--|---|
| Objects are displayed too big or too small by a factor of 2.54. | Coordinates unit | APE_UNITS 100 : "<text>" 101 : "<text>" |
| Objects are shown concentrated around the coordinates origin. | Coordinates mode ABSOLUTE instead of INCREMENTAL | COOR_MODE 110 : "<text>" 111 : "<text>" |
| Objects are displayed in a diagonal row from bottom left to top right.. | Coordinates mode INCREMENTAL instead of ABSOLUTE | |
| Objects are displayed too big or too small by a factor of 0.1/10/100 . | Resolution (number of integer and decimal places, M.N code) | COOR_MN 120 : "<text>%d<text>" 121 : "<text>%d<text>" 122 : "<text>%1d.%1d<text>" |
| Some graphical objects are shown concentrated around the X axis or Y axis. | Coordinate zeroes | COOR_ZEROS 130 : "<text>" 131 : "<text>" 132 : "<text>" |

Examples of aperture lists from various CAD systems and the associated translation files prepared by LPKF are listed below



Note: These translation files were created according to the aperture list available to LPKF. Deviations of individual structures or aperture definitions are imaginable. LPKF cannot assume any guarantee for completeness or correctness of the translation files provided. Modifications must be made by the user.

Alegro

Translation file:

```

APE_UNITS MIL
CONFIGURATION ConfigGerber

1,CN : "LINE      %a D%%"
1,CC : "CIRCLE   %a D%%"
1,QQ : "SQUARE   %a %S D%%"
1,NR : "RECTANGLE %a %b %S D%%"
1,NS : "FLASH    %s %a D%%"

100 : "INPUT-UNITS      INCHES"

```

Matching aperture list:

| INPUT-UNITS | INCHES | | | | |
|-------------|---------------|-----|---|---|-----|
| WHEEL | 1 | | | | |
| LINE | 4 | | | | D26 |
| LINE | 6 | | | | D27 |
| LINE | 8 | | | | D24 |
| LINE | 10 | | | | D25 |
| LINE | 20 | | | | D29 |
| LINE | 25 | | | | D28 |
| CIRCLE | 60 | | | | D12 |
| CIRCLE | 65 | | | | D13 |
| CIRCLE | 80 | | | | D10 |
| RECTANGLE | 27 | 72 | | 0 | D21 |
| RECTANGLE | 30 | 80 | | 0 | D17 |
| RECTANGLE | 34 | 84 | | 0 | D19 |
| RECTANGLE | 46 | 86 | | 0 | D22 |
| RECTANGLE | 50 | 100 | | 0 | D15 |
| RECTANGLE | 56 | 9 | 6 | 0 | D23 |
| RECTANGLE | 72 | 27 | | 0 | D20 |
| RECTANGLE | 80 | 30 | | 0 | D16 |
| RECTANGLE | 84 | 34 | | 0 | D18 |
| RECTANGLE | 100 | 50 | | 0 | D14 |
| FLASH | TQ60-40-15-45 | | | 0 | D11 |

EAGLE

Translation file:

```

APE_UNITS INCH
CONFIGURATION ConfigGerber

1,CN : "D%% draw %a"
1,NC : "D%% round %a"
1,NQ : "D%% square %a"
1,NR : "D%% rectangle %a %s %b"
1,NT : "D%% octagon %a"
1,NC : "D%% annulus %a"
1,NO : "D%% oval %a %s %b"

```

Matching aperture list:

```

D10      draw      0.004
D11      draw      0.006
D12      draw      0.008
D50      round     0.024
D51      round     0.028
D52      round     0.032
D53      round     0.036
D80      square    0.024
D81      square    0.028
D120     octagon   0.024
D121     octagon   0.028
D115     rectangle 0.080 x 0.048
D116     rectangle 0.056 x 0.118
D117     rectangle 0.118 x 0.056

```

ECAM

Translation file:

```

APE_UNITS MIL
CONFIGURATION ConfigGerber

1,CC : "D%% ROUND %a"
1,QQ : "D%% SQUARE %a"
1,NC : "D%% DONUT %a"
1,NT : "D%% OCTAGON %a"
1,NS : "D%% TARGET %a"
1,NS : "D%% CUSTOM %s %a"
1,NS : "D%% THERMAL %a"
1,NR : "D%% RECTANGLE %a %s %b"

```

Matching aperture list:

```

! GAPFile Version 1.0
! Created: Wed Mar 27 14:33:37 1996
! Created by: ECAM (tm) Version 3.21e
GAPFile Version 1.0 (c) 1991 CAD Solutions, Inc.
!
!      Type      Width      Length      Rotate  Offset  Swap  Tool  Size  Legend!
!      English  Metric  English  Metric  English  Metric  Code  No.  English  Metric !
-----+-----+-----+-----+-----+-----+-----+-----+-----+-----+
D10  ROUND      6.00    0.152    0.00    0.000    0  0.00    0.000  D10  0  0.00    0.000  D0
D11  ROUND     10.00    0.254    0.00    0.000    0  0.00    0.000  D11  0  0.00    0.000  D0
D33  SQUARE     50.00    1.270    0.00    0.000    0  0.00    0.000  D33  0  0.00    0.000  D0
D34  SQUARE     58.00    1.473    0.00    0.000    0  0.00    0.000  D34  0  0.00    0.000  D0
D38  OBLONG     75.00    1.905    25.00    0.635    0  0.00    0.000  D38  0  0.00    0.000  D0
D39  OBLONG     25.00    0.635    75.00    1.905    0  0.00    0.000  D39  0  0.00    0.000  D0
D67  RECTANGLE  60.00    1.524    38.00    0.965    0  0.00    0.000  D67  0  0.00    0.000  D0
D68  RECTANGLE  38.00    0.965    60.00    1.524    0  0.00    0.000  D68  0  0.00    0.000  D0
D69  RECTANGLE  92.00    2.337    14.00    0.356    0  0.00    0.000  D69  0  0.00    0.000  D0

```

PADS

Translation file:

```
APE_UNITS MIL
CONFIGURATION ConfigGerber

1,NC : "%% %a 1 F"
1,CN : "%% %a 1 D"
1,NQ : "%% %a 2 F"
1,QN : "%% %a 2 D"
```

Matching aperture list:

POST PROCESSOR REPORT

```
-----
LAYOUT NAME      :- S4224D
LAYOUT TITLE     :- LISTA APERTURE PHOTOPLOTTER
```

SYMBOL AVAILABILITY REPORT

| Position | Size | Shape | Usage | Orient | Length | Count |
|----------|------|-------|-------|--------|--------|-------|
| 126 | 30 | 1 | F | | | 101 |
| 10 | 8 | 1 | D | | | 0 |
| 71 | 12 | 1 | D | | | 0 |
| 73 | 20 | 1 | D | | | 0 |
| 78 | 5 | 1 | D | | | 0 |
| 21 | 80 | 1 | F | | | 7 |
| 144 | 52 | 1 | D | | | 0 |
| 145 | 52 | 2 | F | | | 20 |
| 136 | 44 | 1 | D | | | 0 |
| 137 | 44 | 2 | F | | | 244 |
| 57 | 58 | 1 | D | | | 0 |
| 58 | 58 | 2 | F | | | 20 |
| 76 | 40 | 1 | D | | | 0 |
| 20 | 40 | 2 | F | | | 90 |
| 48 | 60 | 1 | D | | | 0 |
| 56 | 60 | 2 | F | | | 4 |
| 82 | 120 | 1 | D | | | 0 |
| 33 | 48 | 1 | D | | | 0 |
| 34 | 48 | 2 | F | | | 6 |
| 30 | 36 | 1 | D | | | 0 |

```
SHAPE: 1 = ROUND
        2 = SQUARE
```

PROTEL**Translation file:**

```
APE_UNITS MIL
CONFIGURATION ConfigGerber

1,CC : "D%% ROUNDED %a %s %s MULTI"
1,NC : "D%% ROUNDED %a %s %s FLASH"
1,NR : "D%% RECTANGULAR %a %b %s FLASH"
1,QR : "D%% RECTANGULAR %a %b %s MULTI"
1,NO : "D%% ROUNDED %a %b %s FLASH"
1,NT : "D%% OCTAGONAL %a %b %s FLASH"
```

Matching aperture list:

| | | | | | |
|-----|-------------|--------|--------|-------|-------|
| D11 | ROUNDED | 1.000 | 1.000 | 0.000 | MULTI |
| D12 | ROUNDED | 7.333 | 7.333 | 0.000 | MULTI |
| D26 | RECTANGULAR | 62.000 | 62.000 | 0.000 | FLASH |
| D27 | RECTANGULAR | 66.000 | 66.000 | 0.000 | FLASH |
| D28 | RECTANGULAR | 75.000 | 75.000 | 0.000 | FLASH |

11.0 Index

A

| | |
|--------------------------|--------|
| About the program | 14 |
| Action | |
| Cancel | 49 |
| Active layer | 45 |
| Anchor point | 93 |
| Move | 48, 63 |
| Ape_Templates | 10 |
| Aperture | 20 |
| Aperture List | 101 |
| Aperture list | |
| Edit | 33 |
| aperture list | 122 |
| ApeTutor | 39, 86 |
| Application window | 18 |
| ASCII file | 28 |

B

| | |
|-----------------------------|-----|
| Basic settings | 120 |
| Batch Function | 139 |
| Batch job (Executing) | 144 |
| BoardOutline | 51 |
| BottomLayer | 38 |

C

| | |
|--|-----|
| CAD | 42 |
| CAM data | 25 |
| CAM format | 3 |
| Cancel function | 26 |
| Caution | 4 |
| CCAM | 10 |
| Changing the language | 11 |
| Check box | 44 |
| Circuit board | |
| Tutor | 24 |
| Circuit board CAD program | 28 |
| Circuit board outline | 50 |
| Move to different layer | 51 |
| Select | 50 |
| CircuitCAM | 4 |
| Exit | 23 |
| Start | 13 |
| Version number | 14 |
| CircuitCAM - PCB | 74 |
| CircuitCAM Lite | 70 |
| Circular selection | 47 |
| Computer Aided Design (CAD) | 3 |
| Computer Aided Manufacturing (CAM) | 3 |
| Contacting LPKF | 14 |
| Context menu | 19 |
| contour generator | 148 |
| Contour routing | 60 |
| Conventions | 4 |

| | |
|---------------------------|-----|
| coordinates | 102 |
| Coordinates display | 27 |
| Create new document | 26 |
| Cursor grid | 28 |
| Cursor keys | 43 |

D

| | |
|---|---------|
| Danger | 4 |
| Data | 10 |
| Import | 28 |
| Data export | 58 |
| Data select | 29, 36 |
| Data subfolder | 25, 58 |
| Default job | 38, 53 |
| Default settings | 27 |
| Design Rule Check | 117 |
| Display grid | 28 |
| Display unit | 28 |
| Document | 26 |
| Document window | 18 |
| DoubleSided.cat | 86, 120 |
| Draw | 32 |
| Drawing | 99 |
| Drilling Tool (Definition) | 128 |
| Drilling/Milling Tools (creating) | 127 |
| DrillPlated | 39 |
| DXF files importing | 106 |

E

| | |
|-----------------------|---------|
| Error messages | 20, 59 |
| Exercise files | 25 |
| EXPORT | 150 |
| export jobs | 136 |
| Extended Gerber | 33, 103 |

F

| | |
|-----------------------|-----|
| Fiducial | 150 |
| File type | 30 |
| Flash | 32 |
| Flash selection | 47 |
| Flashing | 100 |
| Format | 102 |
| front plate | 145 |
| FRONT TO END | 21 |

G

| | |
|----------------------------|-----|
| General preferences | 11 |
| Gerber | 97 |
| Extended | 103 |
| Syntax | 99 |
| Gerber Format | 97 |
| Gerber format syntax | 97 |
| Gerber-X | 33 |
| Graphic mode | 68 |
| Graphic window | 19 |

| | |
|-------------------------------|-------------|
| Graphical objects | |
| Mirror/Rotate | 92 |
| Select | 46 |
| Grid | 27 |
| Ground plane generating | 113 |
| H | |
| Hotkeys | 153 |
| HP-GL | 59 |
| I | |
| Import | 28 |
| Gerber data | 36, 39 |
| Multiple files | 36 |
| Import assignments | 89, 92 |
| Import production data | 28 |
| Incorrect display | 162 |
| InsulateBottom | 54 |
| InsulateDefault | 70 |
| InsulateTop | 54 |
| Isolate | |
| Isolation width | 71, 72 |
| Primary overlap | 73 |
| Isolation grid | 54 |
| Isolation job | 38 |
| Customize | 70 |
| Isolation jobs (prepared) | 126 |
| J | |
| Job | 126 |
| K | |
| Keyboard functions | 42 |
| L | |
| Layer | 20 |
| Active | 45 |
| Colour | 44, 85, 121 |
| Move | 87 |
| Not selectable | 50 |
| Properties | 44, 50 |
| Registration | 88 |
| Rotate | 92 |
| Vordefinierte | 103 |
| Legend | 4 |
| Line selection | 47 |
| List window | 20 |
| Aperture list | 32, 35 |
| Layer | 43 |
| LPKF BoardMaster | 53 |
| M | |
| Materials supplied | 9 |
| Milling contour | 60 |
| Insert webs | 62 |
| Milling Tool-new (definition) | 132 |

| | |
|--------------------------------|--------|
| Milling-drilling plotter | 58, 60 |
| modified isolation jobs | 124 |
| Move graphical objects | |
| To different layer | 45 |

N

| | |
|---------------------|-------------|
| New_Templates | 10, 85, 120 |
| Note | 4 |

O

| | |
|----------------------------|----|
| Offset | 95 |
| On-line help | 14 |
| Order | 45 |
| Outline illustration | 44 |
| Overview | 42 |

P

| | |
|---------------------------------|--------|
| parameters | 125 |
| Personalize CircuitCAM | 11 |
| Photoplotter | 98 |
| POLYGON | 65 |
| Polygon selection | 47 |
| Predefined Layer | 105 |
| Preferences | 11 |
| Folders | 11 |
| Prepared isolation jobs | 126 |
| PRIMARY OVERLAP | 73 |
| Product ID | 11 |
| Production data | 19, 26 |
| Checking | 42 |
| File extensions | 89 |
| Import | 28 |
| Offset | 87 |
| Output parameters | 87 |
| Preparing for prototyping | 46 |
| Program group | 13 |
| Protolaser | 76 |

R

| | |
|-----------------------------|--------|
| Read Me file | 13 |
| Rectangle selection | 47 |
| Reference coordinates | 88 |
| Refresh | 43 |
| Registration mark | 46, 48 |
| Release number | 11 |
| Report window | 20, 59 |
| Right mouse button | 33, 44 |
| RuboutBottom | 67 |
| RuboutTop | 67 |

S

| | |
|--------------------------|------------|
| Saving a new Layer | 125 |
| Scale | 42 |
| Selection | 42 |
| Selection mode | 48, 49, 68 |

| | |
|---------------------------|--------|
| Selection switches | 47, 48 |
| Set anchor point | 94 |
| Set cursor | 66 |
| SHORTCUTS | 12 |
| Shortcuts | 12 |
| SMD | 24 |
| STANDARD | 37 |
| Start Menu | 13 |
| Status bar | 48 |
| Show/Hide | 22 |
| step-and-repeat | 151 |
| Subfolders | 10 |
| suppressed Zeros | 102 |
| System requirements | 9 |

T

| | |
|-------------------------------------|--------|
| Template file | 10, 26 |
| Create | 85 |
| template file | 120 |
| Template file creating | 121 |
| Text Function | 111 |
| TextBottom | 53 |
| TextTop | 53 |
| Tool | 20 |
| Tool list | 91 |
| Toolbar | |
| Move | 22 |
| ToolTutor | 34, 86 |
| TopLayer | 38 |
| Translation file | 10 |
| Translation File (Updating) | 155 |
| Translation Files (Structure) | 156 |
| True width | 44 |
| Tutor.cat | 86 |

U

| | |
|------------------------|-----|
| Unit | 46 |
| Units | 102 |
| Untitled file | 26 |
| User interface | 13 |
| User preferences | 11 |

V

| | |
|--------------------|----|
| Visual check | 45 |
|--------------------|----|

W

| | |
|--------------|----|
| Wizard | 15 |
|--------------|----|

Z

| | |
|-------------------|----|
| Zoom in | 42 |
| ZOOM OUT | 42 |
| Zoom out | 42 |
| Zoom window | 42 |

