Laser & Electronics

Manual CircuitCAM 4.0

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) 2001 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* 4.0 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.

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 numered. 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

2.0 Getting started 13

2.1 Starting CircuitCAM 13

2.2 Using online help 15

2.3 Show program information 15

2.4 Wizard (Processing Assistant) 16

2.4.1 Basic Wizard Functions 17

2.5 CircuitCAM application window 19

2.5.1 Graphic window 20
2.5.2 List window 21
2.5.3 Report window 21
2.5.4 Toolbars 22
2.5.5 Status bar 23

2.6 Opening the "Tutor" sample circuit board 24

2.7 Closing CircuitCAM 24

3.0 Preface to the exercises 25

3.1 "Tutor" sample circuit board 25

3.2 Listing of the exercise files 26

4.0 Exercise 1 27

4.1 Create new CircuitCAM document **28**

4.2 Customizing grid, unit and coordinates display . . . **29**

4.3 Import production data **30**

4.3.1 Aperture list 31
4.3.2 Tool list 36
4.3.3 Gerber data for solder and component sides 38
4.3.4 Excellon data 41
4.3.5 Importing the data using the import assignment . 43

5.0 Exercise 2 44

5.1 Checking production data **44**

5.1.1 Working with the view functions 44
5.1.2 Working in the layer list window 46

5.2 Preparing production data for prototyping **50**

5.2.1 Deleting graphical objects outside the circuit board50
5.2.2 Moving the circuit board outline to a separate layer53

5.3 Improved Coordinate Entry **56**

5.3.1 Placing text objects on separate layers 57
5.3.2 Isolating with the default job 58
5.3.3 Preparing for Production Data Export 59
5.3.3.1 Checking The Tool Lists 59
5.3.3.2 Export production data 62

6.0 Exercise 3 64

6.1 Using contour routing **64**

6.1.1 Modifiable Milling Direction and Sequence 68

6.2 Create rubout area **69**

7.0 Exercise 4 74

7.1 Customize isolation job. 74

7.1.1 CircuitCAM - Lite 74
7.1.2 CircuitCAM - PCB 78
7.1.3 Insulating Frames (PCB's only) 82
7.1.4 Color Assignment for Insulating Tools 88

8.0 Exercise 5 90

8.1 Creating a CircuitCAM template file 90

8.2 Move layer 92

9.0 Exercise 6 94

9.1 Using import assignments 94

9.2 Rotate layer 97

10.0 Appendix. 102

10.1 Gerber Format. 102

10.1.1 Use of Gerber Format 102
10.1.2 Structure of a Photoplotter. 103
10.1.3 Gerber Format Syntax 104
10.1.3.1 Drawing. 104
10.1.3.2 Flashing. 105
10.1.4 Aperture List 106
10.1.5 Units, Format and suppressed Zeros. 107
10.1.6 Extended Gerber 108

10.2 Predefined Layer 110

10.3 Importing DXF files. 111

10.4 TextFunction 116

10.5 Generating Ground Planes. 118

10.6 Design Rule Check 122

10.7 CircuitCAM template file 126

10.7.1 Creating your own template file 127
10.7.2 Saving an aperture list in the template file 128
10.7.3 Saving a modified isolation job 130
10.7.4 Saving a new layer 131
10.7.5 Saving the parameters 132

10.8 Prepared isolation jobs 133

10.9 Creating Drilling/Milling Tools 134

10.9.1 Definition of New Drilling Tool 135
10.9.2 Definition of New Milling Tool 139

10.10 Export Jobs 143

10.11 Batch Function (PCB Version Only) 146

10.12 Processing AL-front panels (PCB version) 152

10.12.1 Prepare front plate outline data 153
10.12.2 Preparing cutting and drilling data 154
10.12.3 Preparing text and scale data 156
10.12.4 Setting text data 158
10.12.5 Exporting and saving 160
10.12.6 Positioning the front panel on the ProtoMat . . 161

10.13 CircuitCAM 4.0 Hotkeys 164

10.14 Updapting Translation File 166

10.14.1 Structure of Translation Files 167
10.14.2 Tips for Adapting Translation File 173

11.0 Index 178

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. 128 MB
- VGA colour monitor (recommended resolution 1024 x 768 pixels)
- Microsoft Windows 95/98/ME or Windows NT 4.0/2000 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 tutorial
- License card

1.3 Installation

CircuitCAM is installed from one CD-ROM.

How to install *CircuitCAM*:

› Insert the CD-ROM in your CD drive.

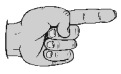
› Select **RUN...** from the Windows Start menu.

› Then click **BROWSE**.

› Select the *CircuitCAM4.0* 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 dialogue box and cancel the installation.

1.4 Directories created during installation

By default, the installation program creates a "LPKF 40" subdirectory in your program directory. Programs such as *CircuitCAM* and *BoardMaster* are saved here.

The following subdirectories are created in the "LPKF 40" 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** dialogue box is displayed; here you can personalize your *CircuitCAM* software:

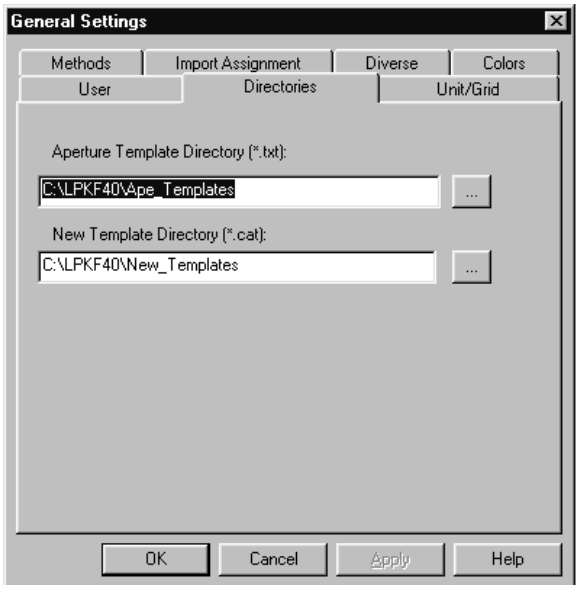
Figure 1:

› Please enter the data required in this dialogue box.

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

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

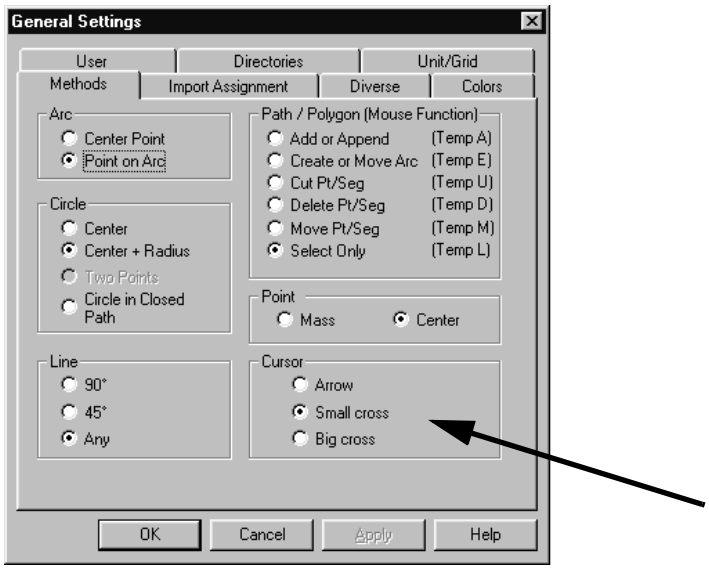
Figure 2:



This settings will be saved in the Windows registry.

Note: You can also select the new setting BIG CROSS (see menu below in this *CircuitCAM* Version 4.0 by clicking in METHODS under CURSOR.

Figure 3:Großes Kreuz



When set a large cross appears covering the entire screen. This new “Cross hair cursor” allows simple modification and creation of layout data.

2.0 Getting started

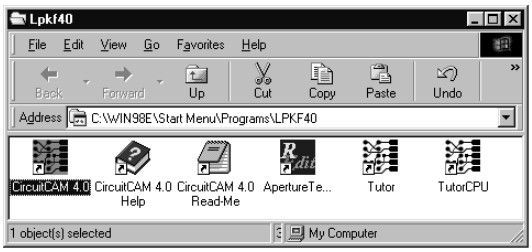
In this chapter, we would like to give you a first look at *CircuitCAM* and familiarize you with the user interface of *CircuitCAM*. We will show you how you can customize *CircuitCAM* to your requirements and how to use the online help when you require more information on a particular topic.

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 LPKF40 program group will be found under **PROGRAMS** in the Windows 95/98/NT **START** Menu; it will contain the following icons:

Figure 4:

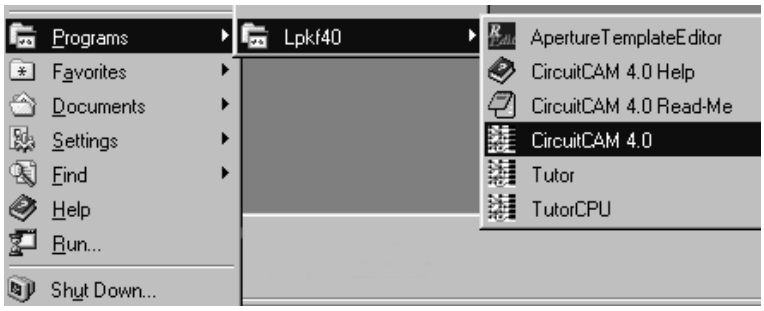


Note: Double-click on the READ ME icon to read the latest information on CircuitCAM not available when this tutorial was printed.

How to start *CircuitCAM*:

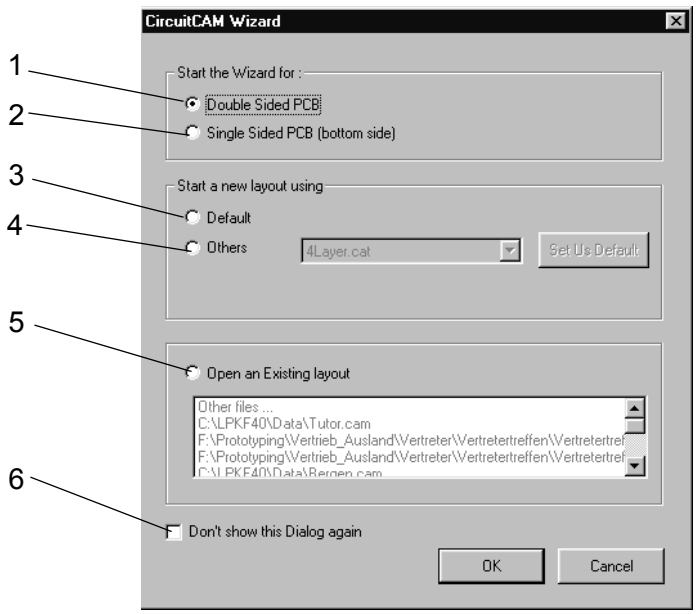
Select the *CircuitCAM* 4.0 shortcut from the LPKF 40 program group in the **START** Menu.

Figure 5:



The following window appears:

Figure 6:Start Up Fenster



1- When this option is marked, the Wizard (input assistant) for creating a double-sided circuit board is opened after clicking “OK”.

2- When this option is marked, the Wizard (input assistant) for creating a single-sided circuit board is opened after clicking OK.

3- When this option is marked, the Circuit-CAM standard screen with standard settings is opened after clicking “OK”.

4- When this option is marked, the CircuitCAM standard screen with settings corresponding to a selectable template file is opened after clicking OK..

5- When this option is marked, the CircuitCAM standard screen with a correspondingly selected existing *.cam file is opened after clicking “OK”.

6- When this option is marked, this window is no longer displayed in the future when the program is started after clicking “OK”. This can be switched back over by clicking the marking box “Display start dialog” under configuration\ General \ Diverse.

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.

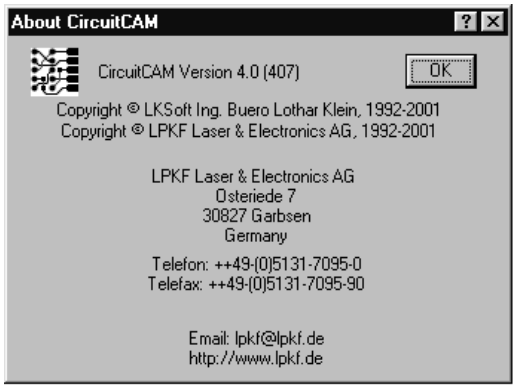
› Select an item from the menu bar or click on a button in a toolbar to obtain more detailed information on the function selected.

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:



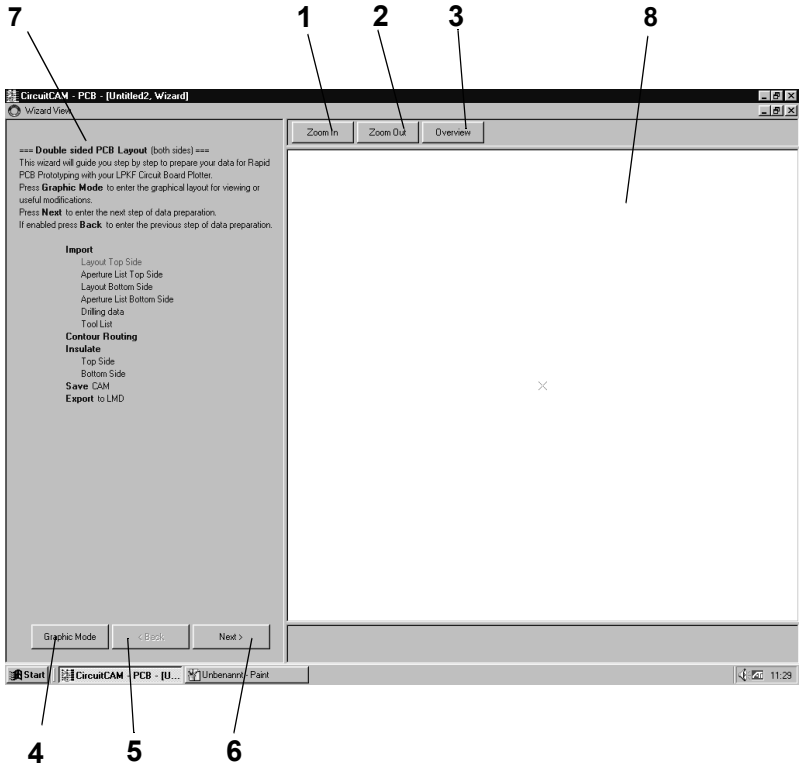
This dialogue box also contains information of how you may contact us.

2.4 Wizard (Processing Assistant)

The “Wizard” is new in this *CircuitCAM*, Version 4.0. It 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 opens automatically after starting *CircuitCAM* (illustration below), when it has been selected in the start-up window.

Figure 9:Import Wizard



1- **Zoom in**: Clicking this button increases the size of the data displayed (8).

2- **Zoom out**: Clicking this button reduces the size of the data display (8).

3- **Overview**: Clicking this button displays the entire overview in the data display (8).

4- **Graphic mode**: Clicking this button switches off the import Wizard.

5- **Back:** Clicking this button goes back one step in the processing sequence.

6- **Next:** Clicking this button jumps forward to the next operating step.

7- Report window: This window shows instructions, descriptions and aids on the corresponding operations.

8- This window shows the data in the current processing phase.

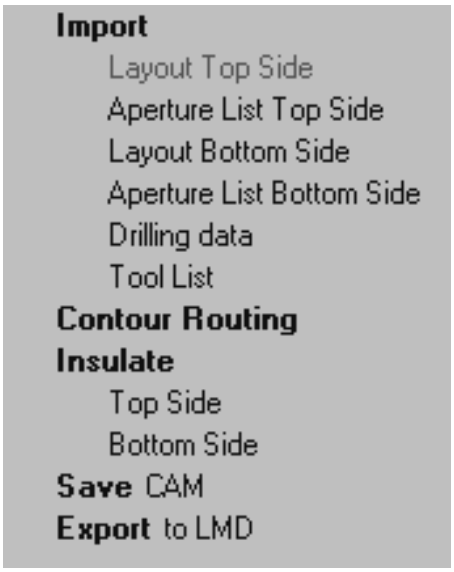
Note: In order to work in the normal graphic mode, the Import Wizard can be switched off by clicking graphic mode. If you want to change to the Wizard while processing in graphic mode, click the button.

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 for each operation. In comparison to Version 3.0, 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 correctif 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 aseparate 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:

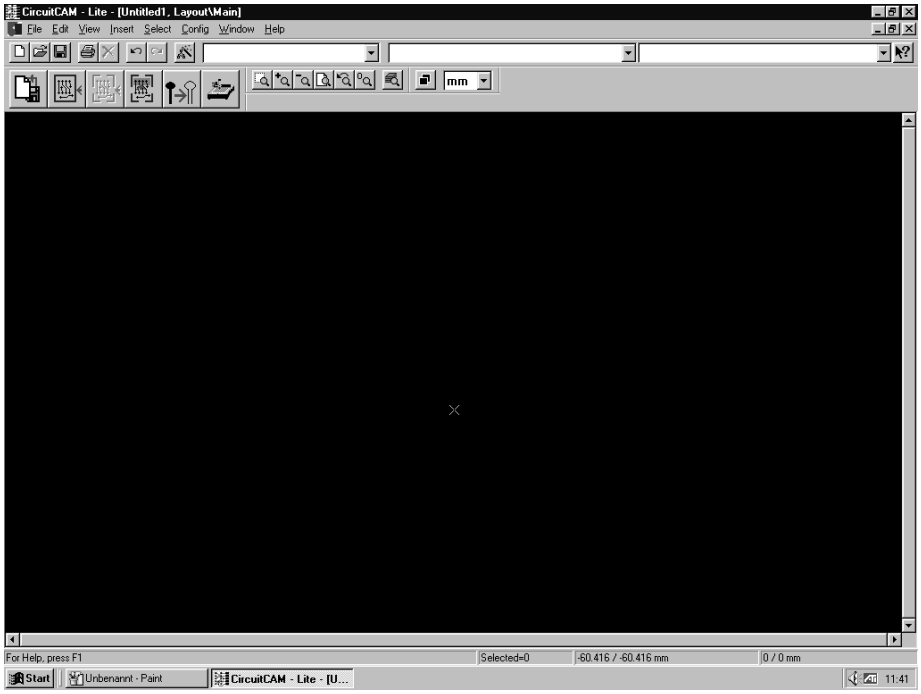
The completed CAM file can be exported in the LMD format which can be processed with the *BoardMaster* software to control the contour router accordingly

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.

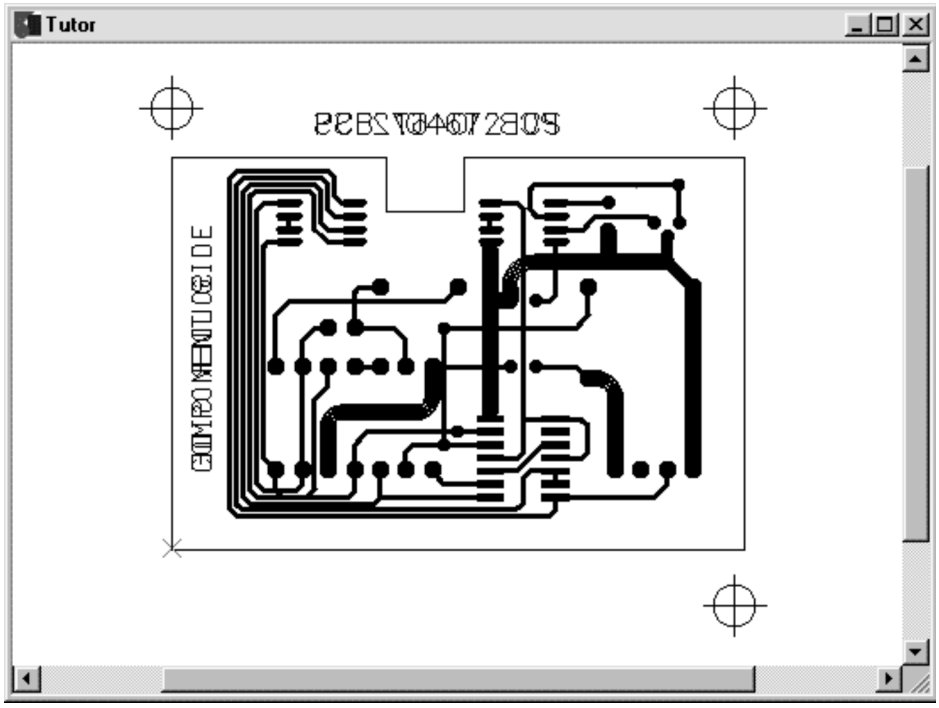
Note: You can open a number of document windows in CircuitCAM at the same time. The WINDOW menu gives you access to functions for arranging the various document windows and for switching between the document windows.

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.

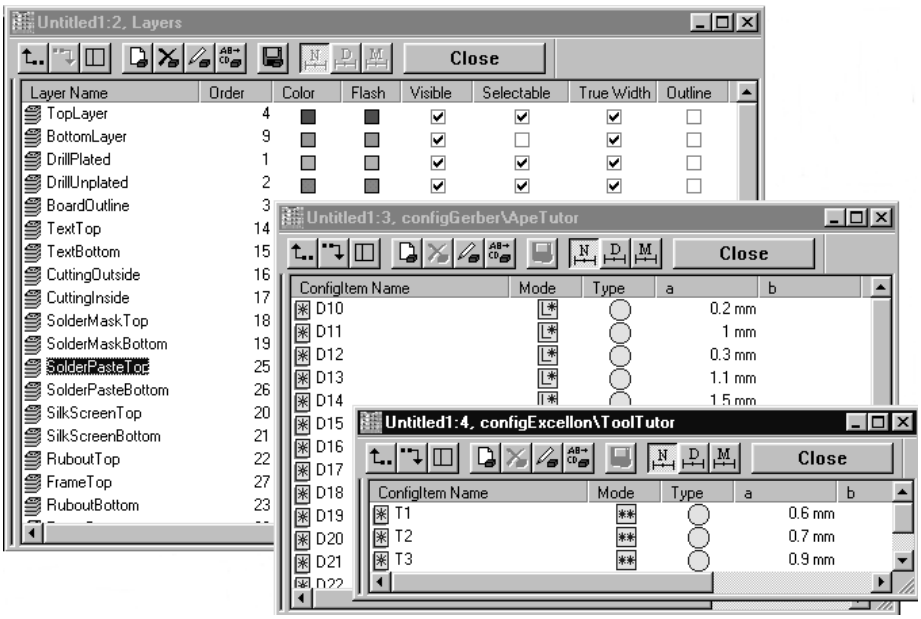
Note: When the mouse pointer is in the graphic window, clicking the right mouse button will access a context menu for editing selected objects:

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 31).

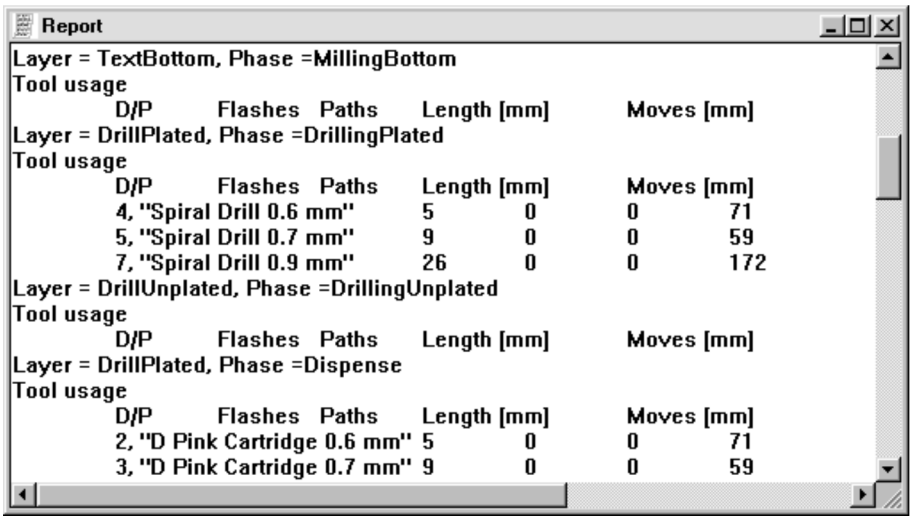
Figure 14:



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:



2.5.4 Toolbars

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

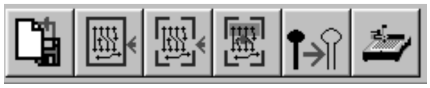
DEFAULT

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:

CircuitCAM Lite-version

CircuitCAM PCB-version

Functions for customizing the coordinates display and the match-up-layer icon.

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.

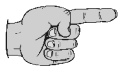
How to open the "Tutor" sample circuit board:

› Select the entry **OPEN** in the menu **FILE**.

› Select the directory C:\LPKF40\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:

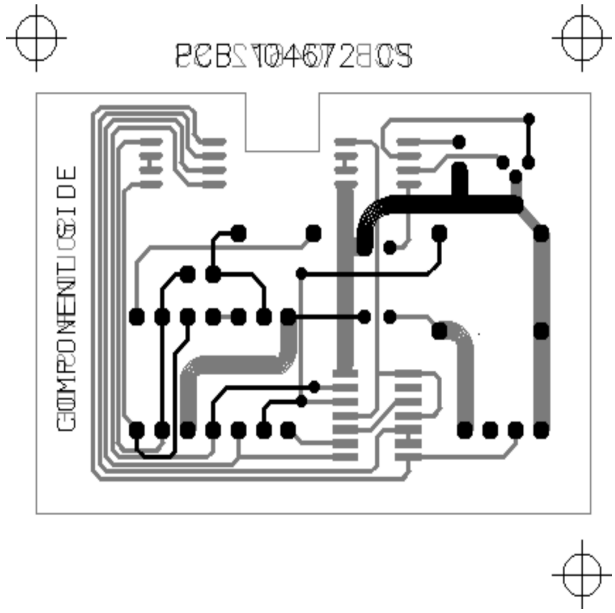
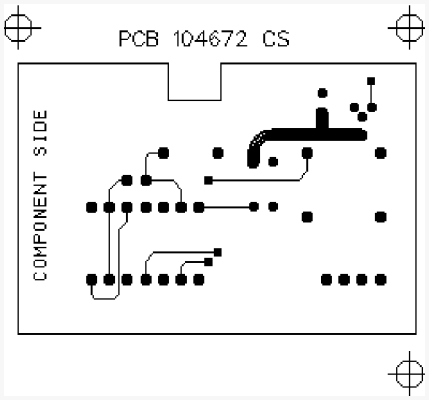
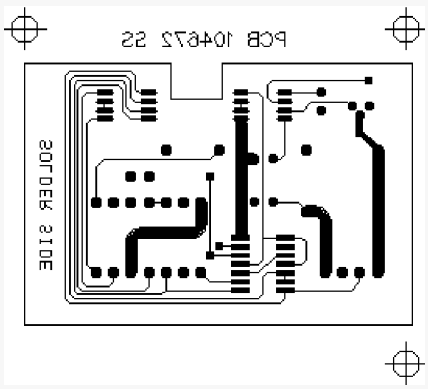


Figure 24:

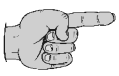
Componente side („TopLayer")	Solder side („BottomLayer")
	
(Standard color::red)	(Standard color::green)

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, *CircuitCAM* version 4.0 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

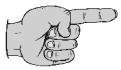
4.1 Create new CircuitCAM document

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 documentsare 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 dialogue 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 dialogue box and cancel the function.
This works in all dialogue boxes.

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

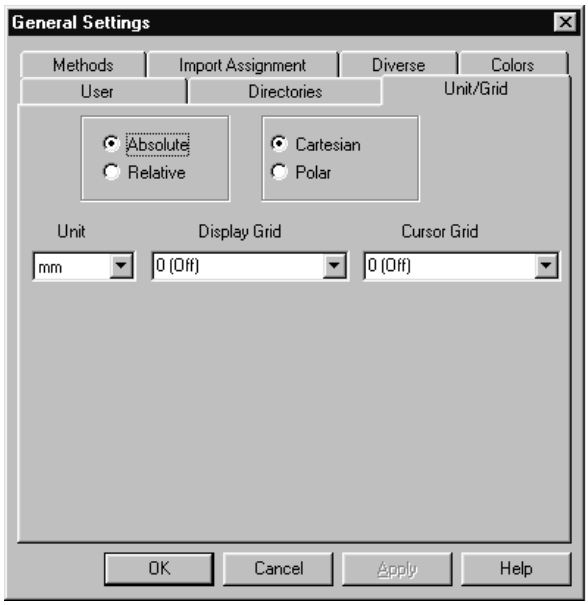
4.2 Customizing grid, unit and coordinates display

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.

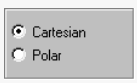
The functions for setting the grid, unit and coordinates display are grouped together in the **CONFIGURATION\GENERAL\UNIT/GRID** toolbar:

Figure 25:



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

Figure 26:

	Switching over the coordinate display in the status bar between **ABSOLUTE COORDINATES** and **RELATIVE COORDINATES**.
	Switching over the display format in the status bar between **CARTESIAN COORDINATES** and **POLAR COORDINATES.**

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 (Nur PCB Version)
- Extended Gerber	- DXF
- Excellon	- ODB++ (Nur PCB Version)
- Sieb & Meyer	- EESof Mask (Nur PCB Version)
- Barco (Nur 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 102.

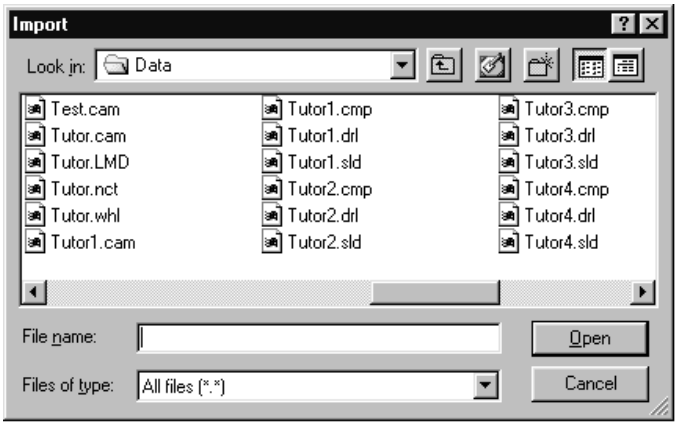
Note: An incorrect aperture list definition will be revealed by incorrect representation of the pads and conductor tracks in the graphic window, i.e these objects will be too small or too large in dimension, or displayed in the incorrect form.

How to import the aperture list:

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

The **OPEN** dialogue box is displayed:

Figure 27:

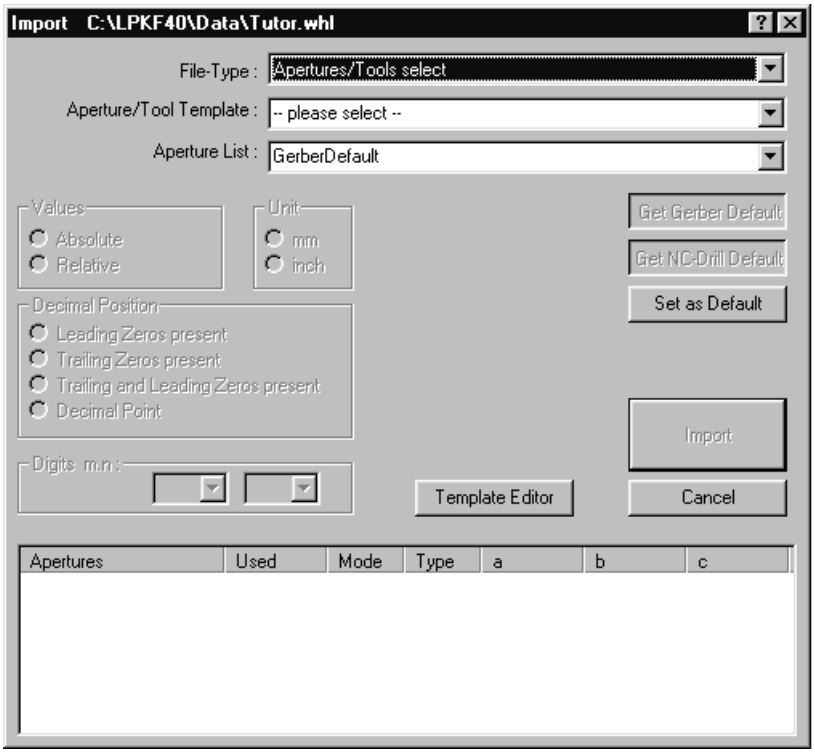


› Select the file**TUTOR.WHL** and click on **OPEN** to import the aperture list.

The Import dialogue 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.

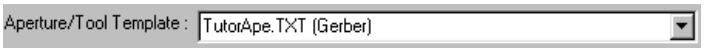
APERTURE/TOOL TEMPLATE:
Appears if the file type **APERTURES/TOOLS** is selected.
Here you should select a translation file for your production data. *CircuitCAM* needs this translation file to convert the data correctly as the aperture and tool lists used by circuit board CAD programs are not standardized.
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.

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

Figure 29:

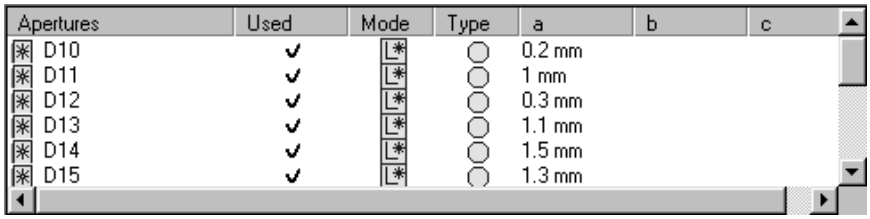


› Enter **„APETUTOR1“** as the name for the aperture list in the listbox labeled **APERTURE LIST**.

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 dialogue box.

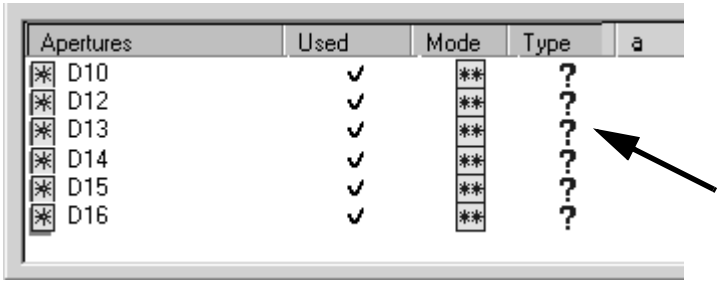
Figure 31:



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).

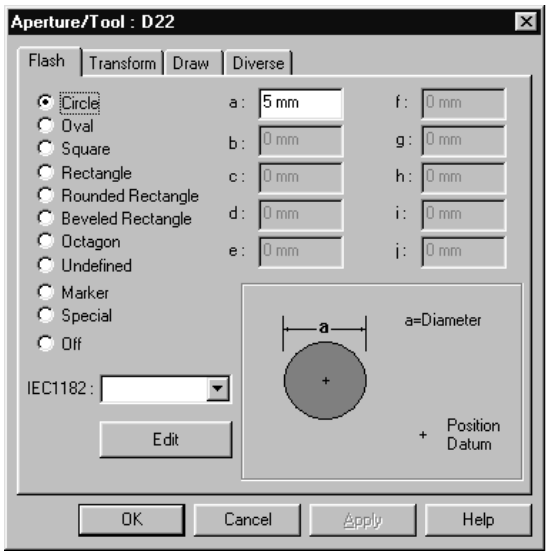
Figure 32: Undefined tools



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 <u>not</u> 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, defintion 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.

An additional program for simple creation of new translation files for apertures and tool lists can be opened by clicking the button “Template Editor” on the Import dialog box.

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.

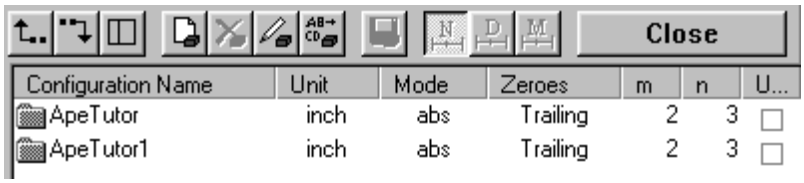
› Click on **IMPORT** to import the aperture list.

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.

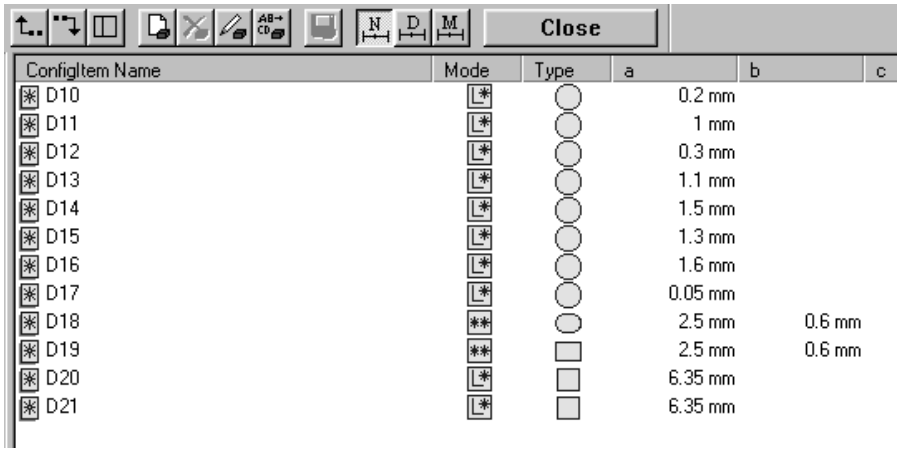
CircuitCAM shows the imported Gerber aperture lists in the list window:

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** dialogue box is displayed.

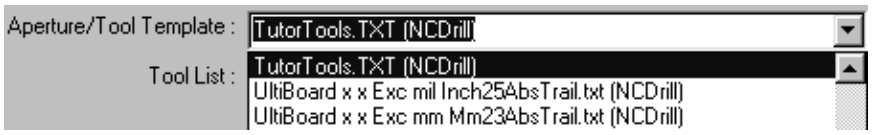
› Select the file **“TUTOR.NCT“** and click on **OPEN** to import the tool list.

The **IMPORT** *dialogue 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.

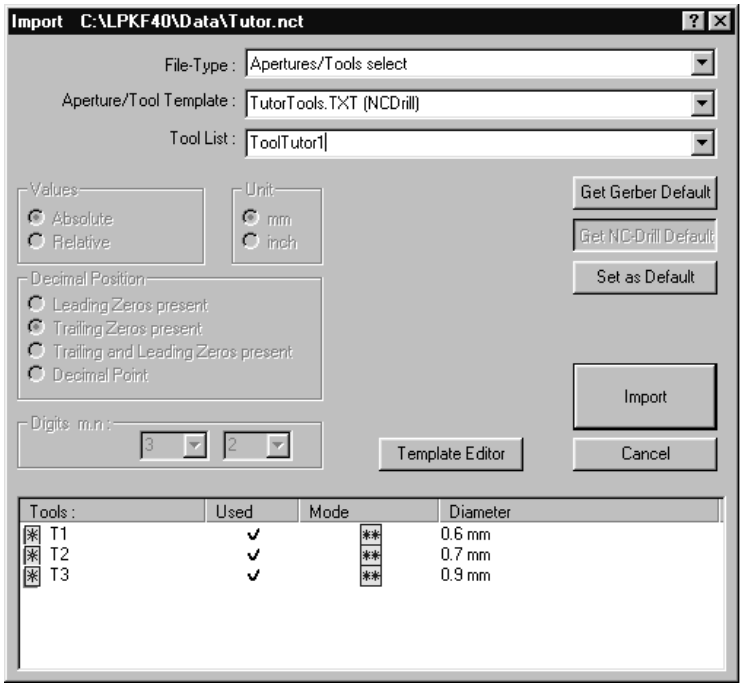
› Select the “TutorTools.txt“ translation file from the **APERTURE/ TOOL** translation list box.

Figure 36:

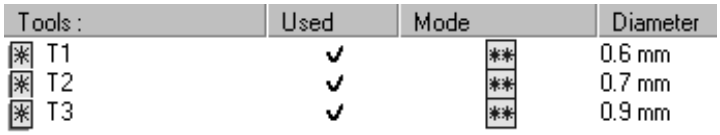


› Enter **“TOOLTUTOR1“** as the name for the tool list in the **TOOL LIST** edit field.

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** dialogue box.



› 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.

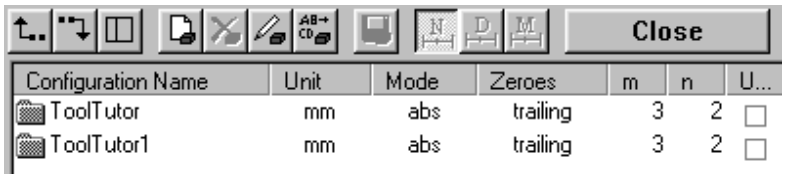
› Click on **IMPORT** to import the tool list.

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.

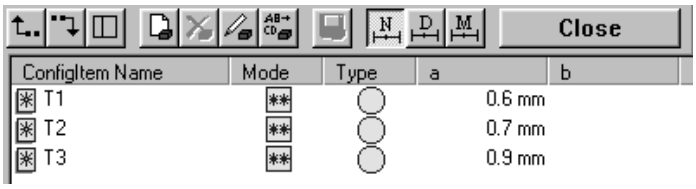
CircuitCAM shows the imported Excellon tool lists in the list window:

Figure 38:



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

Figure 39:



› 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** *dialogue 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** *dialogue 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)

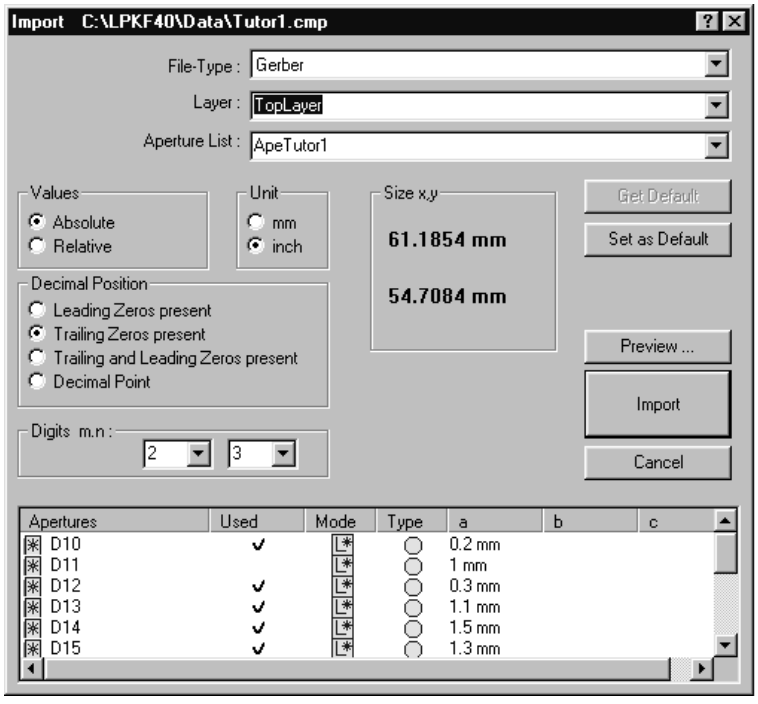
Note: Select the Gerber option under File-Type if *CircuitCAM* does not automatically interpret the file to be imported as a Gerber file.

› 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 **IMPORT** to import the file and to open the **IMPORT** dialogue 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.

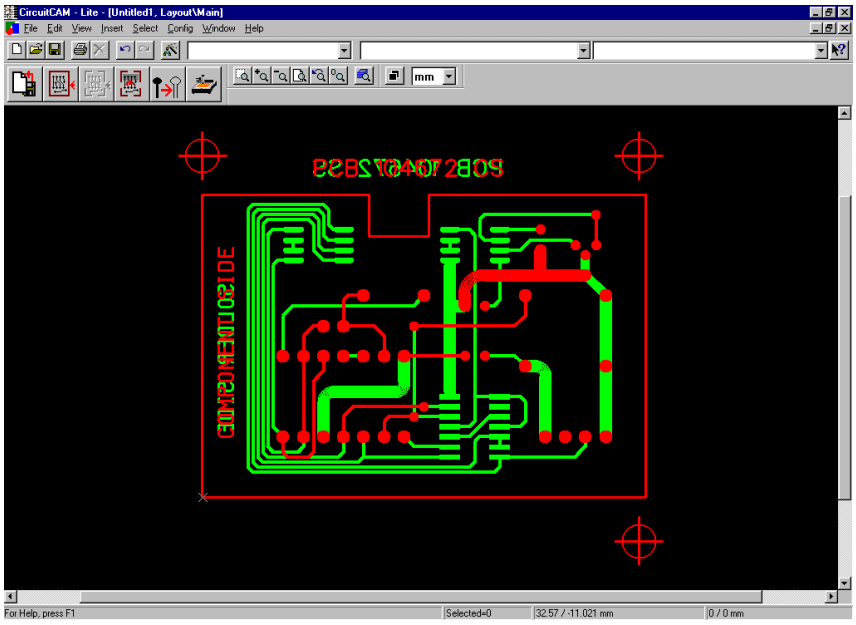
› Select in the list field **APERTURELIST** previously imorted aperture list „**APETUTOR1“**.

› 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 dialogue 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:

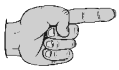
› Select **IMPORT...** from the **FILE** menu, or click on in the **FRONT-TO-END** toolbar.

The **OPEN** *dialogue box is displayed.*

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

The **IMPORT** *dialogue 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:



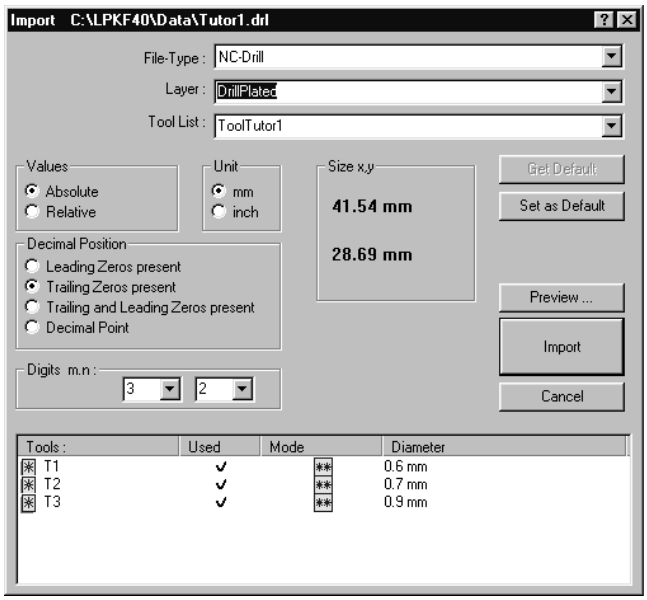
Note: Select Excellon from the file type list box if *CircuitCAM* **does not automatically interpret the file to be imported as an Excellon file.**

› 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.

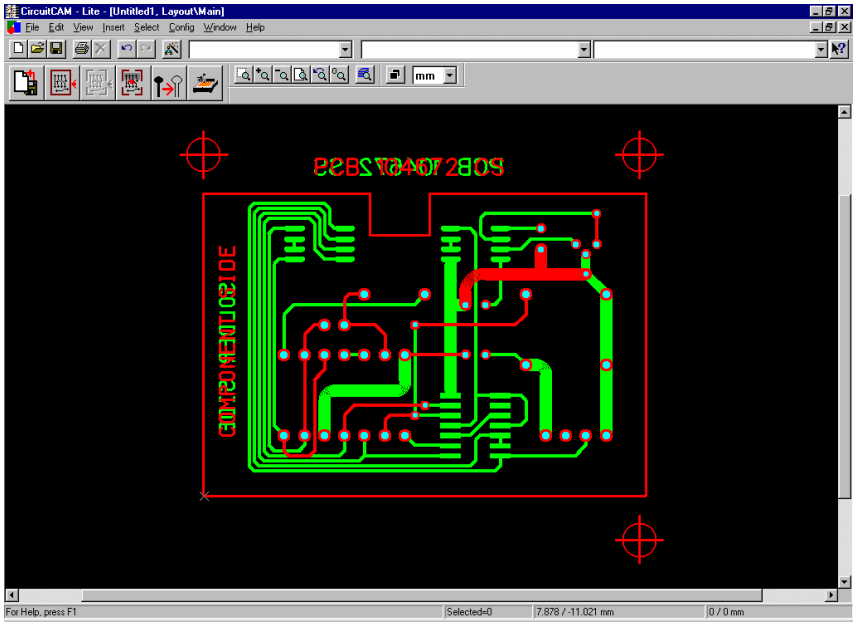
Figure 44:



› 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

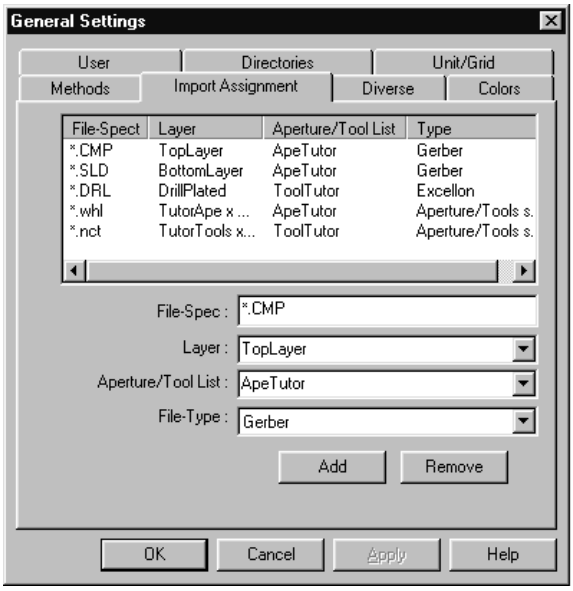
Importing data is significantly easier and quicker if you work with the aid of import assignment. Before importing, the parameters that were assigned manually previously in this exercise are predefined in a dialogue box. This means that the data is displayed on the screen as soon as it has been selected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

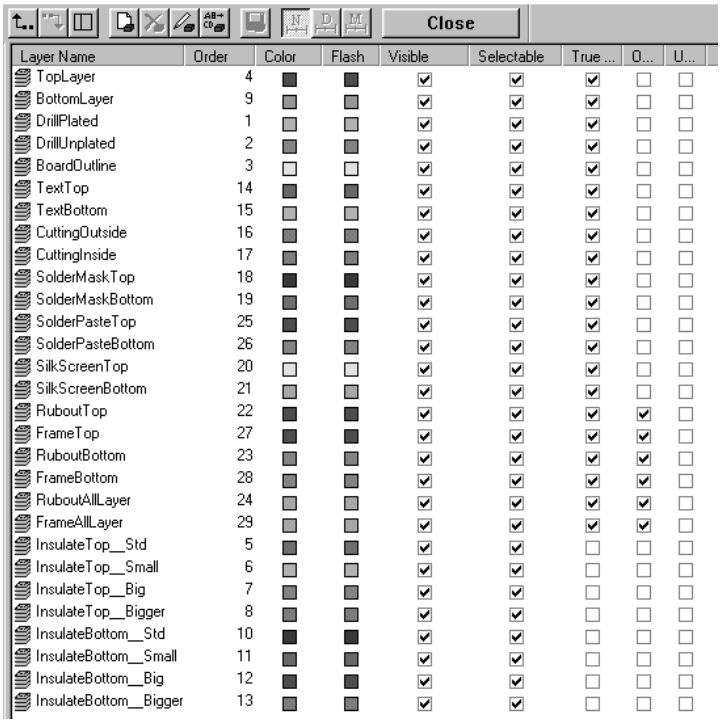
Use the **CURSOR KEYS**on the keyboard and the scrollbars to move the selection area.

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:



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

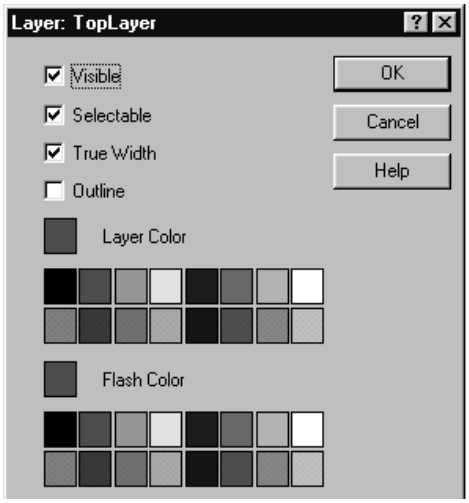
NAME LAYER

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 dialogue box by selecting a layer and clicking on .

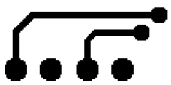
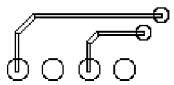
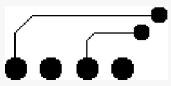
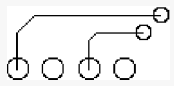
Figure 48:



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

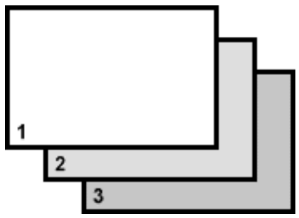
The following overview will clarify for you the effects of the True width and Outline illustrationsettings:

Figure 49:

	True width ☐	Outline-illustration ☑
TRUE WIDTH ☑		
TRUE WIDTH ☐		

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 4.0* - 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.

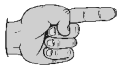
› Click on the right mouse button and click now on **MEASURE** in the just opened menu.

› 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:

LAYER

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.

APERTURES/TOOLS

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.

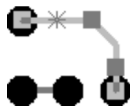
FLASH SELECTABLE
Pads (flashes) and drillings may be selected if this button is enabled.
All switches are enabled in the default setting (everything may be selected).

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.

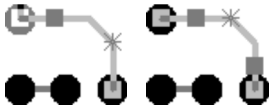
If just one object is selected, the handles for the selected object will be displayed:

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:

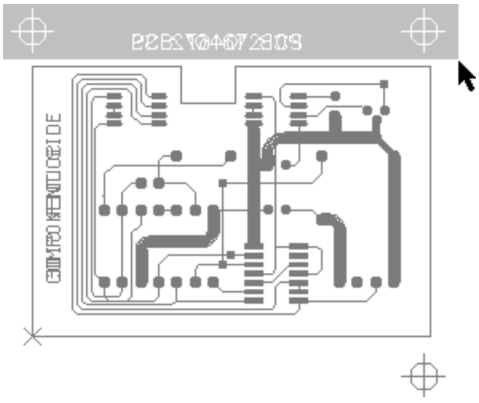
› Make sure that all selection switches are enabled in the Select toolbar so that it is possible to mark all graphical objects:

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:

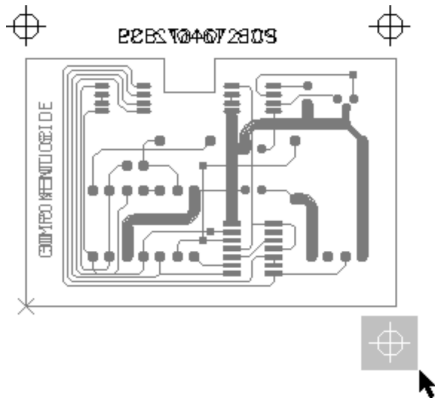
Figure 54:



› 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:

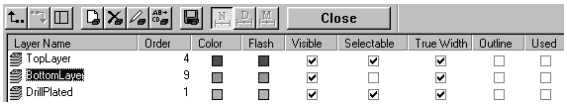
Note: As the circuit board outline on our sample circuit board is located on two different layers, it is useful to assign the property not selectable to one of the two layers to make it easier to select the other layer.

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:



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.

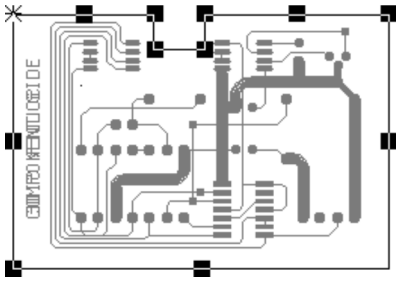
› Click on the upper left-hand corner of the circuit board outline in the graphic window to select it.

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**:

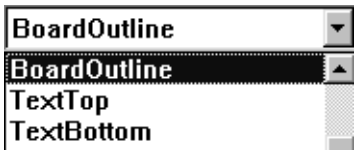
Figure 58:



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 Improved 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:

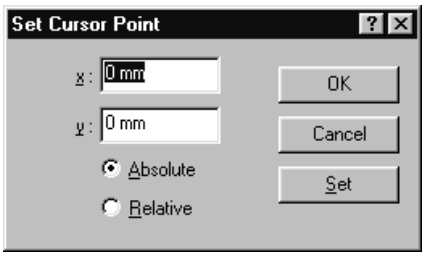
› Select a layer.

› Select the **POLYGON** tool or **CLOSED LINE**.

› Select **PROCESSING**, **SET CURSOR** in the main menu bar.

The following window appears:

Figure 60:



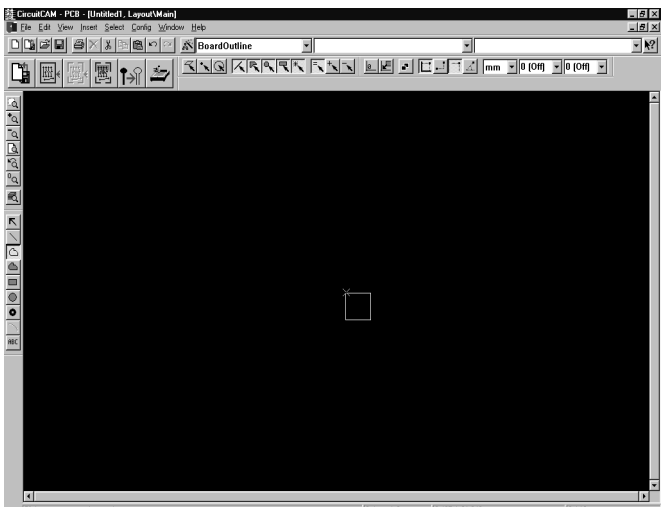
› Make the following entries in the **SET CURSOR POINT** window:

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

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

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

Figure 61:



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:

Text on separate layer

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

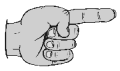
CircuitCAM offers default jobs which may be used for most circuit boards without modification of the isolation parameters to calculate the isolation channels. Later you will learn how you can customize the defaults for calculating the isolation channels individually.

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.



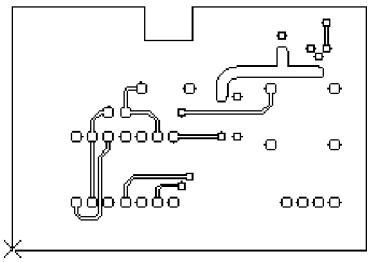
Note: Calculation of the isolation channels may take some time depending on the number of graphical objects and the parameters set such as the isolation grid.
In addition, the processor performance and the amount of memory in your computer are important factors for the time requirement.

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 Chapter “Color assignment for isolating tools” on Page 93).

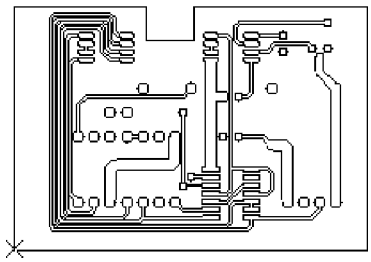
The isolation channels are illustrated graphically after calculation. The isolation channels on the component side are located in the **"INSULATETOP"**layer, those on the solder side on the **"INSULATEBOTTOM"** layer:

Figure 62:

Component side ("InsulateTop")



Solder side ("InsulateBottom")



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 4.0 *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 list 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 go to the next chapter “Checking the Tool Lists” on page 59.

5.3.3.1 Checking The Tool Lists

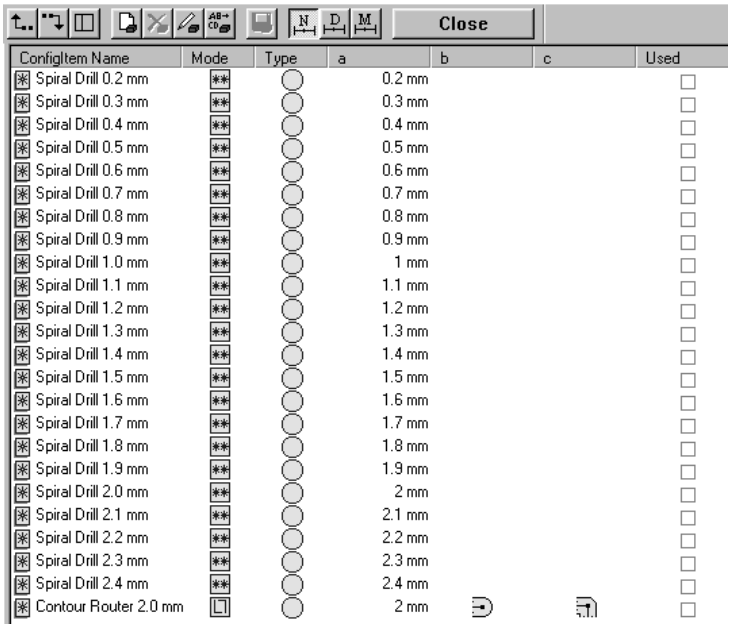
Check the tool list as described below if required:

› 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.

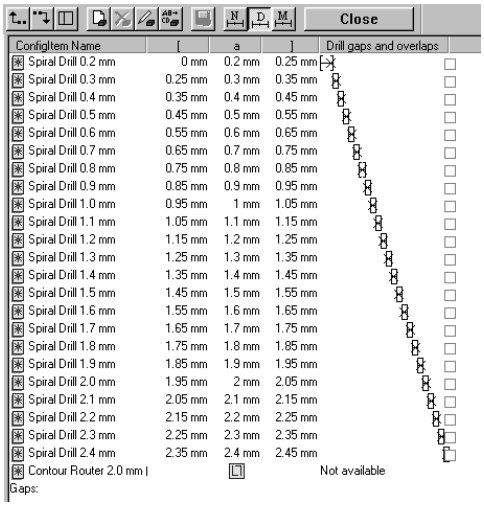
Figure 63: Tool list



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

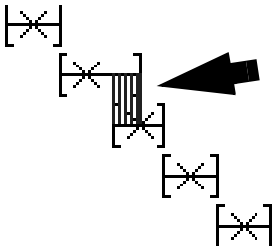
Here the drilling dimensions are displayed graphically in relation to one another on the right side of the tools (see figure below)

Figure 64:Drilling tool list



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

Figure 65: Tool overlapping

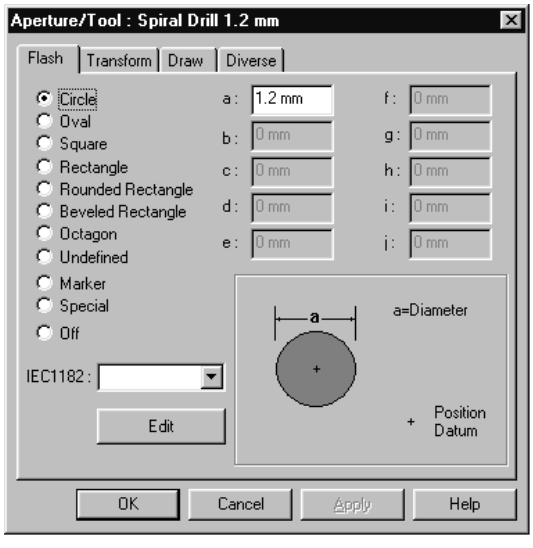


In this case the tool to be corrected should be changed so that the overlap is corrected or eliminated.

› 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 66: Flash

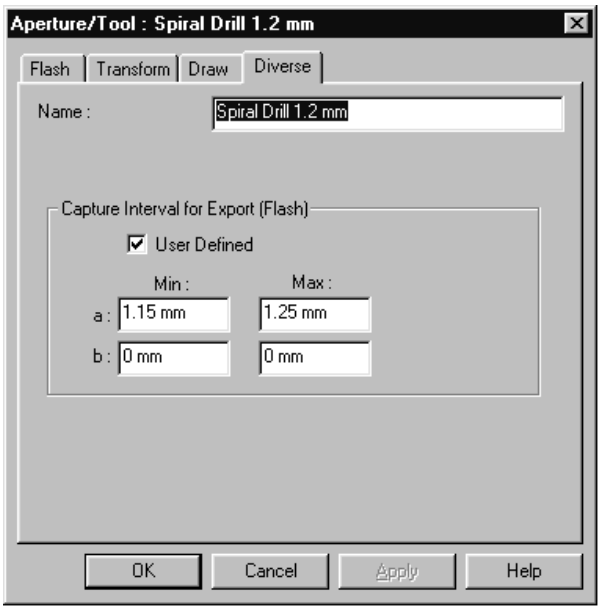


Here you can set the diameter of the tool as required.

› Click on **DIVERSE**.

The following illustration appears:

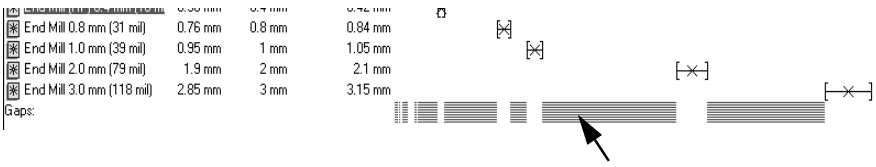
Figure 67: Diverse



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 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 68: Maßfehler bei Fräswerkzeugen



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:

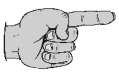
  With this job the production phases for the *AutoContac* system are not output. This has the advantage, that superfluous error messages regarding the *AutoContac* system are avoided.

- 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".

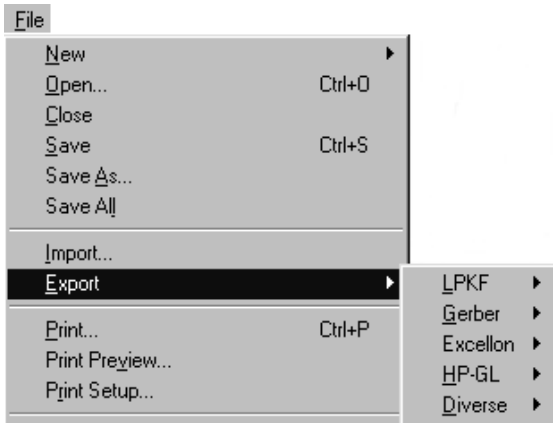
Following the export, the **REPORT WINDOW** displays information regarding the data exported, with any error messages.



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 69:



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.

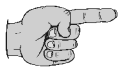
6.1 Using contour routing

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**.

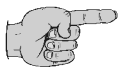


Note: Once you have opened the file "TUTOR1.CAM" in CircuitCAM, you should save this document to a different file name to ensure that the original data remains available to you for further exercises.

› 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.

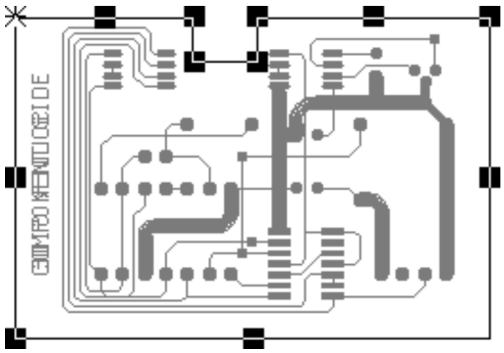


Note: If you are having the circuit board milled out by the circuit board plotter, it is no longer necessary to place the circuit board outline on a separate layer, as was the case in exercise 2.

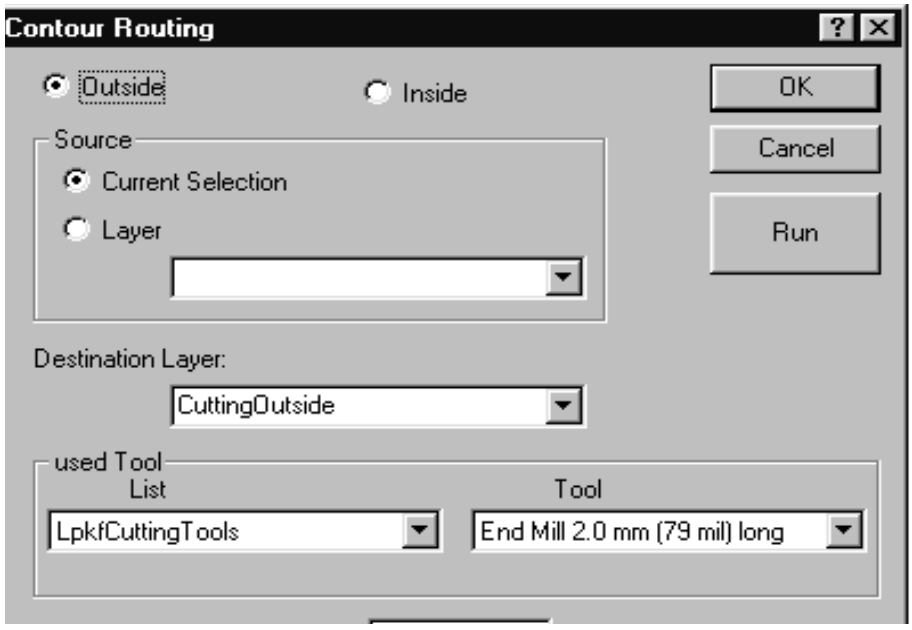
How to create a milling contour:

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

Figure 70:



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

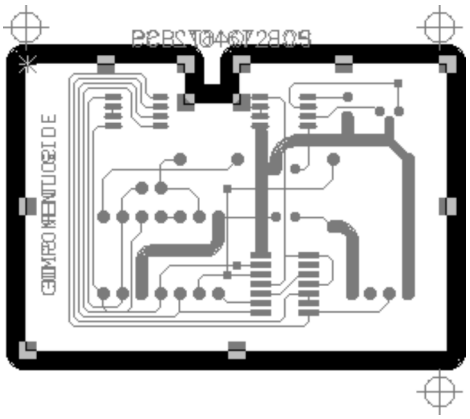


Note: If the circuit board outline for the circuit board is already placed on a separate layer, e.gon the "BOARDOUTLINE" layer, you may use this layer as the source for calculating the milling contour by clicking on the LAYER radio button and selecting BOARDOUTLINE from the LAYER list box.

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

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

Figure 71:



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.

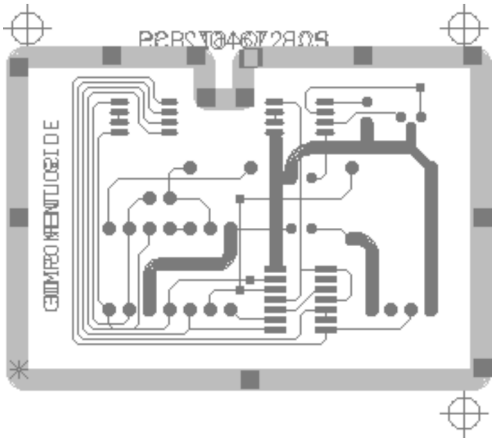
Caution! The circuit board would be completely milled out of the board material with the current contour and could be thrown about by the rotating milling head.

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:

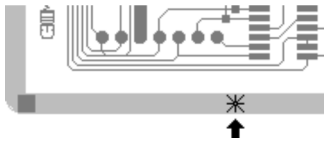
› Select the contour generated on the **"CUTTINGOUTSIDE"** layer

Figure 72:



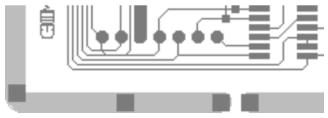
› Use the **+** and **-** keys to move the anchor point ✳ from handle to handle on the contour line.

Figure 73:



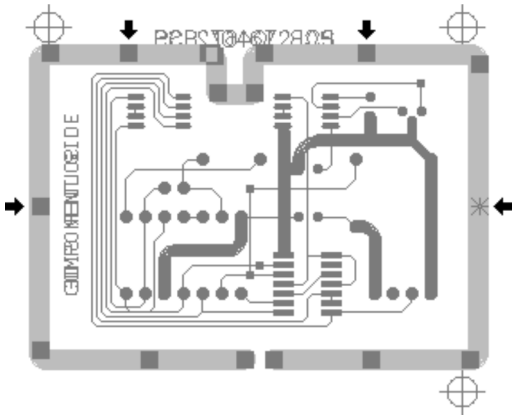
› Press **CTRL** + **G** or click on in the **FRONT TO END** toolbar to insert a tab at the anchor point position:

Figure 74:



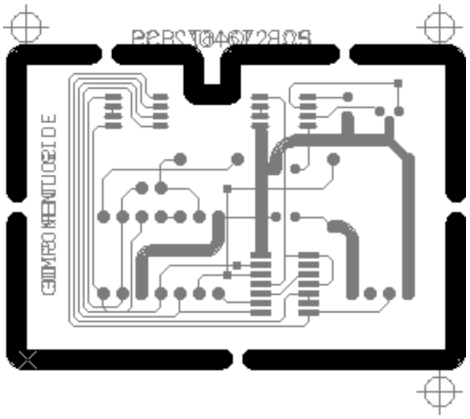
› Repeat steps 2 and 3 to insert break-out-tabs at subsequent positions:

Figure 75:



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

Figure 76:



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 buttom.

› Select **EDIT**, **SWAP ...** in the main menu bar.

The following menu appears:

Figure 77: Vertauschen

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

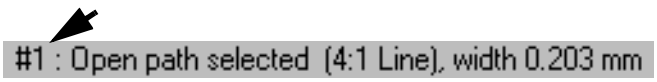
Move to first: This conductor is cut first.

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).

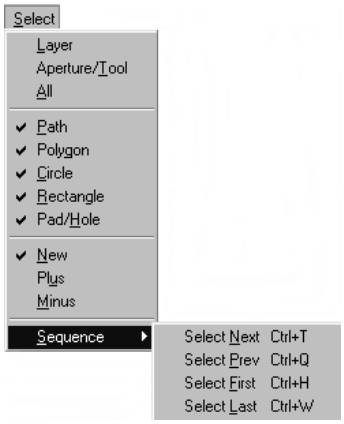
Figure 78:



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

To check the cutting path sequence, use the function in the menu **SELECT - SEQUENCE** (see menu below).

Figure 79:



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:

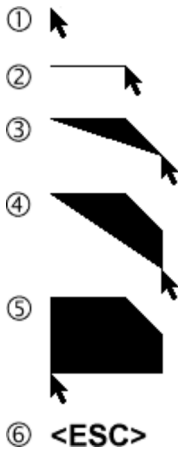
RUBOUT AREA ALL LAYERS

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

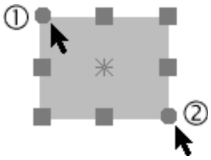
You can define the range of the Rubout area using the following drawing functions which can be found in the **GRAPHICS** toolbar:

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.



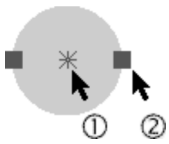
RECTANGLE
(default setting for Rubout area)
dined by two mouse clicks:



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

CIRCLE
defined from the centre point by two mouse clicks:

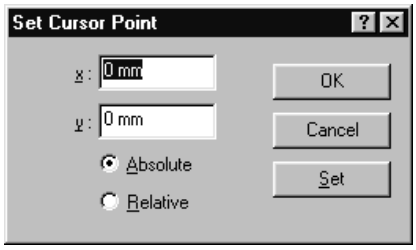
Figure 80:



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 dialogue 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.

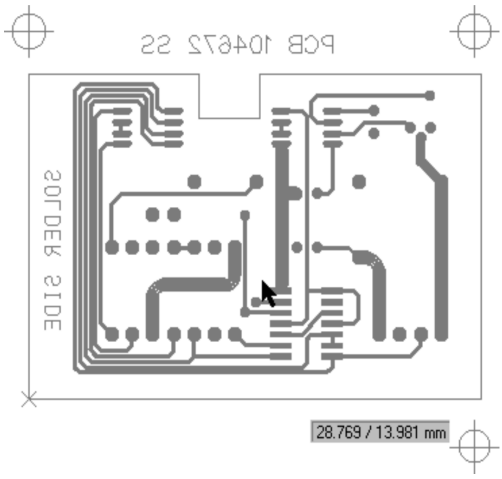
The **RUBOUT ALL LAYER** *layer is now active, the default for the rubout area shape is* **RECTANGLE***.*

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.

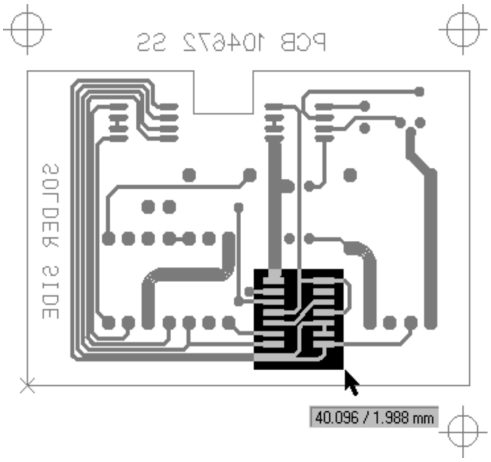
› Click above and to the left of the 14 pin SMD IC to set the starting point of the rectangle:

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.

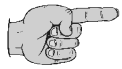
Note: Since a rubout area is a graphical object just like any other, you can also select rubout areas resize, reshape, move and delete them.

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.

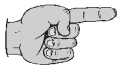
Note: Any text objects over which the rubout area is positioned must be located on an electrical layer, so that they are also taken into consideration in the calculation of the isolation channels for the rubout area.

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.

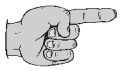


Note: At the end of the exercise 3 you can close the document "TUTOR3" by clicking on CLOSE in the menu FILE.

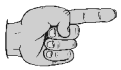
Note: You are now familiar with the basis functions of *CircuitCAM* for preparing production data for circuit board prototyping.



Note: A list of the predefined layers and insolating 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 **“INSULATEDEFAULTTOP...”** and **“INSULATEDEFAULTBOTTOM...”** are assigned by default to the button with the functions **INSULATE ALL LAYERS** .

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

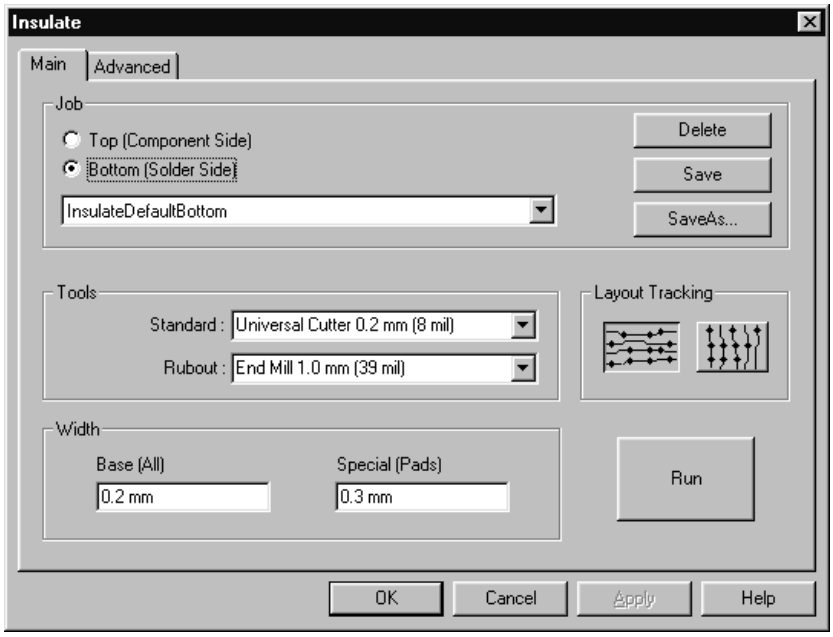
Open the **ISOLATE** dialogue box by selecting **INSULATE...** from the **EDIT** menu. The dialogue box is divided up into two pages to which you can jump by clicking on the tab.

The illustration shows the configuration for the **“INSULATEDEFAULTBOTTOM”** default job.

7.1.1 CircuitCAM - Lite

Settings on the Start tab

Figure 84:



JOB:

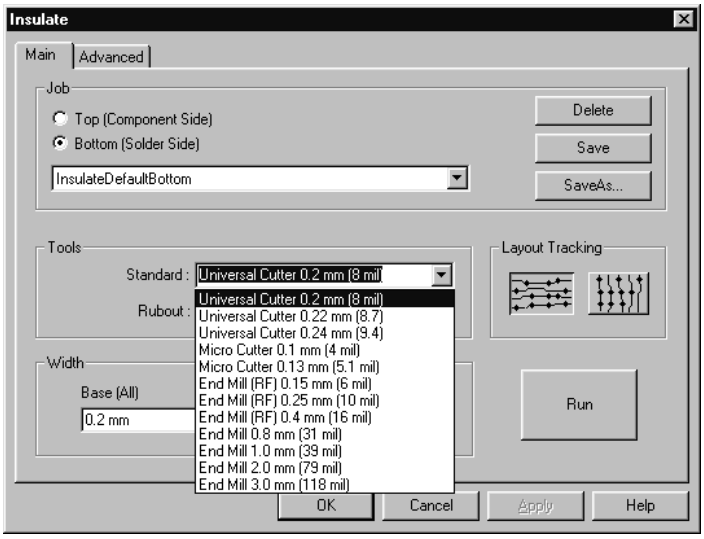
Select the side of the circuit board for which the job is to be executed. The corresponding layer, **“TOP”** or **“BOTTOM”**, will then be used automatically.

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:

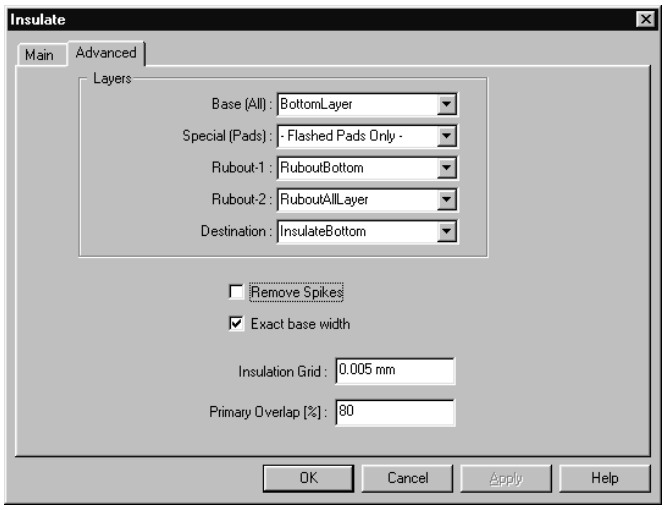
Clicking on the **SAVE** button saves the modified Isolation Job. The modification will also be saved to the underlying template file (*.CAT).

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:

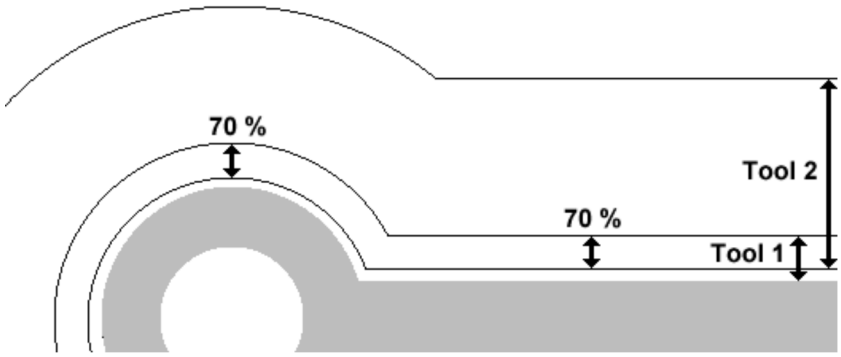
This value specifies the precision to which the isolation tracks are calculated. A smaller value means higher precision which means a longer calculation time. A value of 0.005 mm should be selected for structures with a conductor track width and minimum distance of 0.2 mm. Higher values (0.05 mm) can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



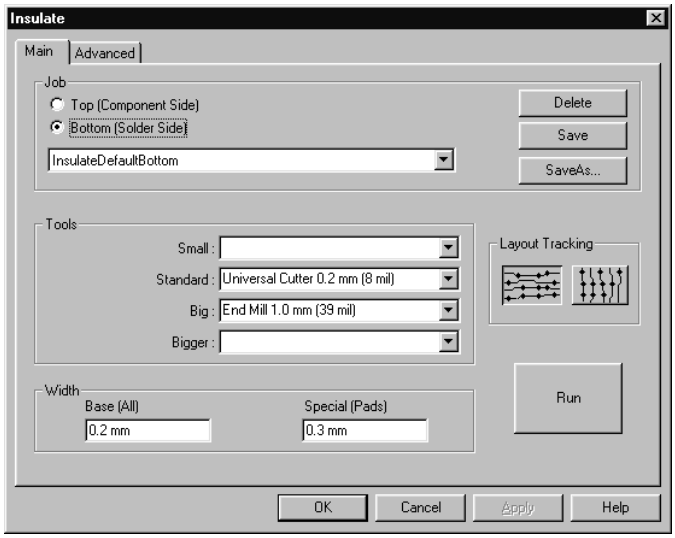
Note: The next larger milling path should overlap the smaller milling path by between 50% and 90%.

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:

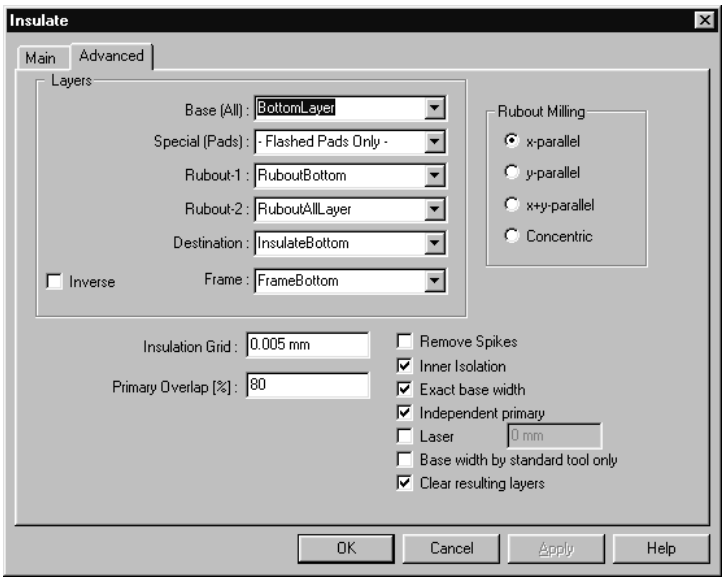


TOOLS:

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 4.0* . 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

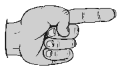
When this box is checked, the milling paths for the resulting layers from insulating operations performed previously are deleted (default setting). If the checkbox is switched off, the new result is added to the previous.

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 millling 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 (PCB's only)

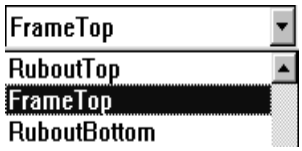
If you want to create circuit boards to be used in the high-voltage range or high-frequency range, you can increase or reduce the insulation intervals separately in certain areas with this function. Moreover, this function can be helpful for optimizing the milling times and tool costs.

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.

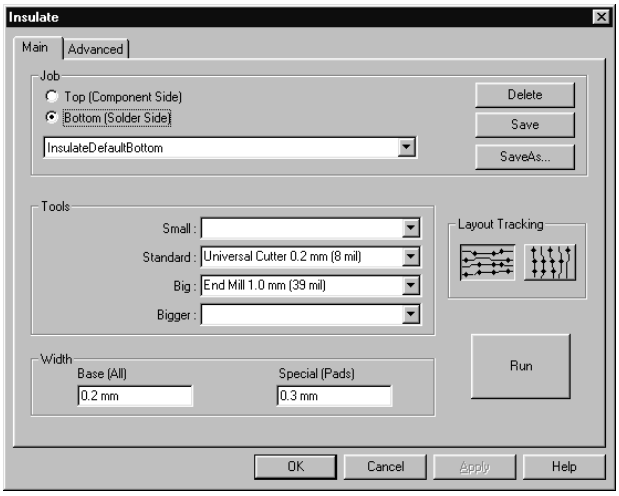
› Press the **POS 1** key on your keyboard to refresh the display.

› Select the menu **EDIT, INSULATE**.

The following window appears:

Settings on Main Index Card

Figure 91: Isolieren - Start



Select the radio button “Component side (Top)”.

Explanation on Windows and Functions:

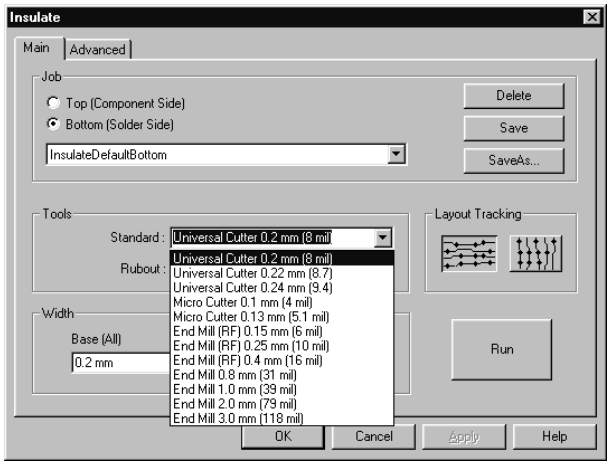
JOB:

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:

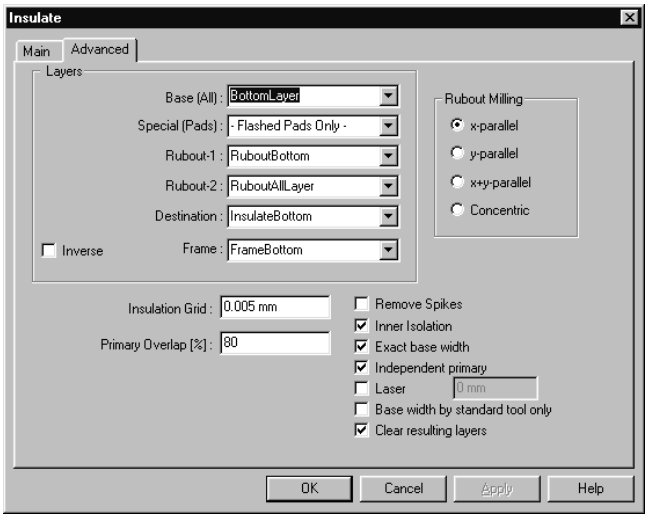
You can save the modified insulating job by clicking the **SAVE** button. Changes are also stored in the basic template file (*.CAT).

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: Isolation



Explanations on Windows and Functions:

LAYERS:

In the group box layers, you can specify which layer 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 miling 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.

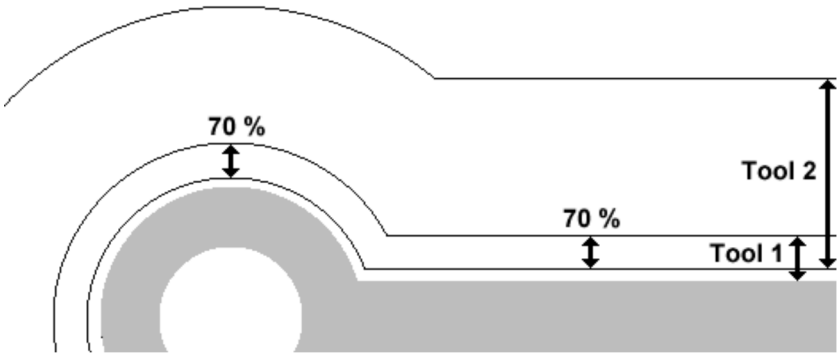
Note: Since the insulating operation is grid-based, the calculation time is linear to the number of grid points to be calculated. Therefore, the finer the insulation grid, the longer the calculation operation requires. Recommended insulation grid: 0.005mm

(For extremely slow computers: 0.01mm)

PRIMÄRY 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 be 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 %.

› Enter the tools (see figure 94) and layer (*FrameLayer*) as in Figure Figure 95: and Figure Figure 96:.

› Switch on the checkbox “Clear resulting layers” and switch off the “Inverse“ checkbox.

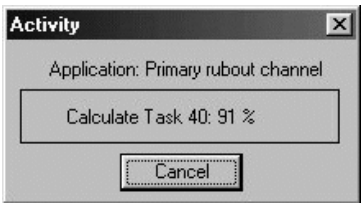
› Change back to the **MAIN** index card.

› Enter the value of 1.5 mm for the base insulation width.

› Click **RUN**.

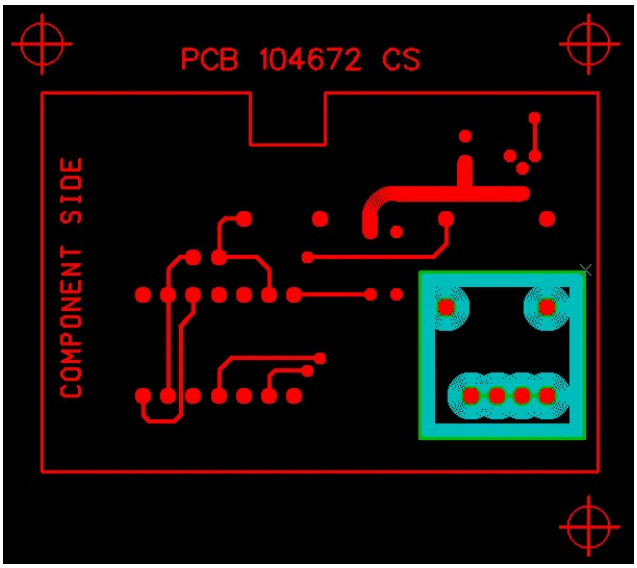
The following window appears:

Figure 95:



When the insulating process is completed, the screen could look by this:

Figure 96:



› Select **INSULATE** in the menu **EDIT**.

› Select the radio button “Component side (Top)”.

› Enter 1.0 mm for the base insulation width and 0.2 mm for special.

› Change back to the **ADVANCED** index card.

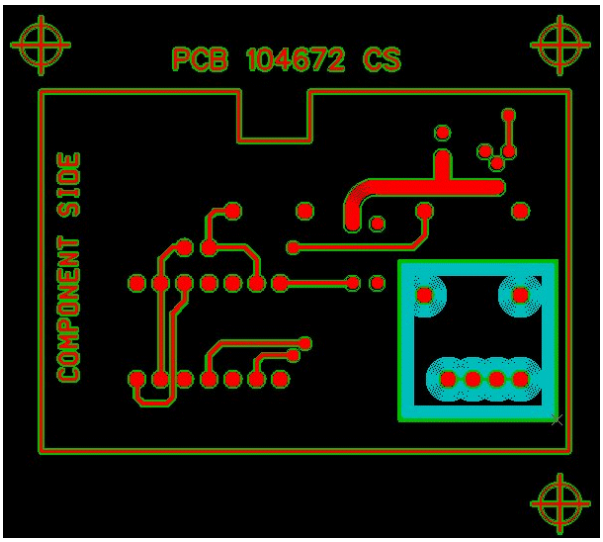
› Switch the “Inverse” checkbox on by clicking and the “Clear resulting layers” checkbox off.

› Change back to the **MAIN** index card.

› Click **RUN**.

The display could now look like this:

Figure 97:



A number of insulation paths were generated within the frame around the conductors and solder eyes to obtain a greater insulation. Moreover, an insulation channel was generated around the conductors and solder eyes.

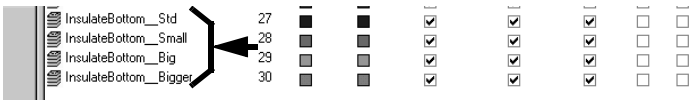
7.1.4 Color Assignment for Insulating Tools

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 98:Layer Liste



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 99:

› Click the desired color with the left mouse key.

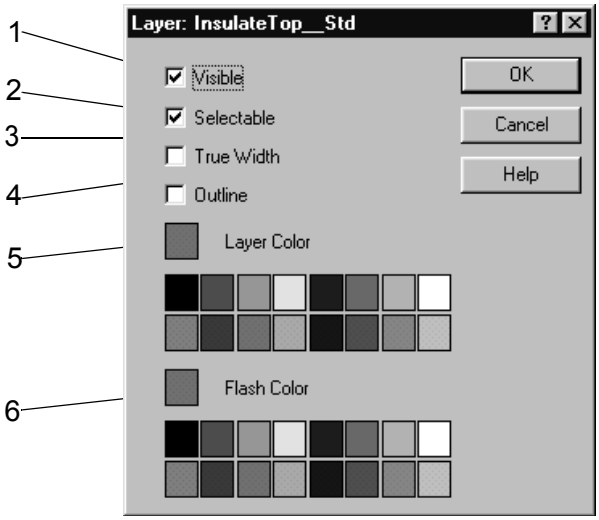
› The corresponding color is assigned and the window closes automatically.

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 100: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.

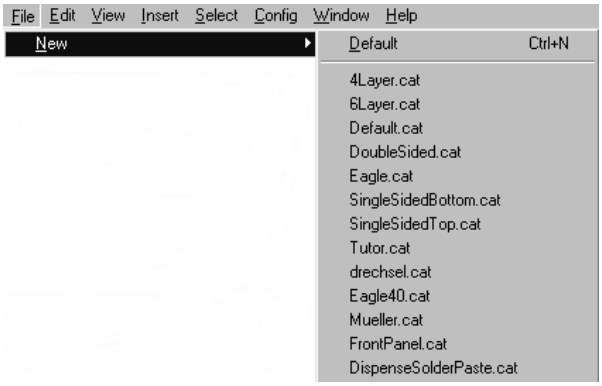
If, for example, you use the same aperture and tool list again and again, you can create your own template file which already contains both these lists. You will then not need to import these aperture and tool lists again for new circuit boards.

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 101:

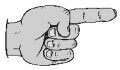


Various template files are saved to the templates directory when you install *CircuitCAM*:

Template file	Intended Purpose
Default.cat	Standard template
SingleSidedTop.cat	Single-sided PCB (Only component side)
SingleSidedBottom.cat	Single-sided PCB (Only solder side)
DoubleSided.cat	Double-sided PCB
4Layer.cat	Multilayer circuit board, 4 layers
6Layer.cat	Multilayer circuit board, 6 layers
Tutor.cat	Double-sided PCB (with import assignments for Tutor data)
EagleTutor.cat	Double-sided PCB (with import assignments for Eagle data)

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** dialogue box. This directory is located in the **"LPKF 40"** 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:

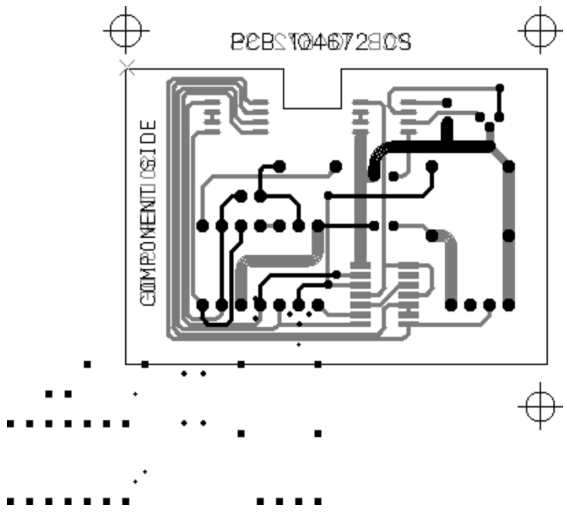
› Select **TUTOR.CAT** from the **NEW** submenu of the **FILE** menu.

› 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 102:



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.

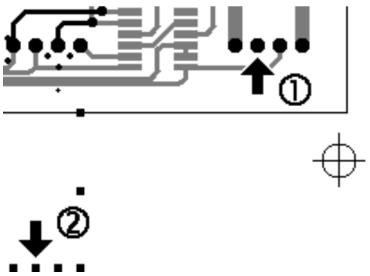
CircuitCAM includes the following function to make the layers coincide again:

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 103:



Note: A significant object can be a drill hole and it`s solder pad, as long as at 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.

› 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 datawith 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 a 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**.

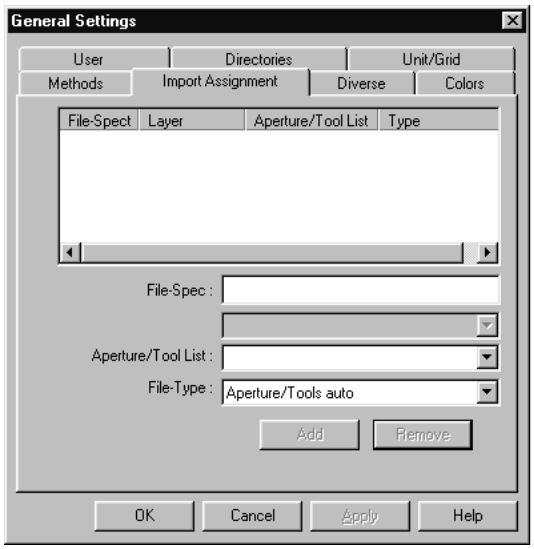
Save this new document as template file under a new name (Test.CAT) in the table **"NEW TEMPLATES"**. Now the import-allocations are saved in a new CAT-file and not in default.CAT.

How to specify import assignments:

› Open the **CONFIGURATION** menu and select the **GENERAL...** line to open the **GENERAL SETTINGS** dialogue box.

› Click on the **IMPORT ASSIGNMENTS** tab.

Figure 104:



› 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 dialogue box.

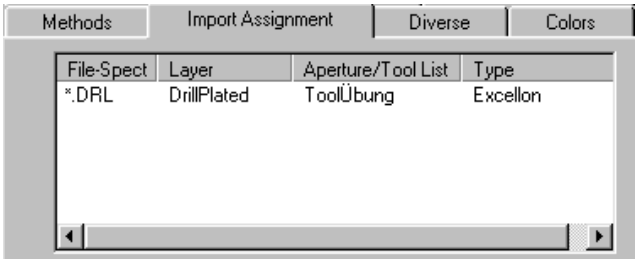
› Select **TOOLEXERCISE** from the **APERTURE/TOOL LISTS** list box to use this tool list for the Excellon production data to be imported.

› 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 105:



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**.

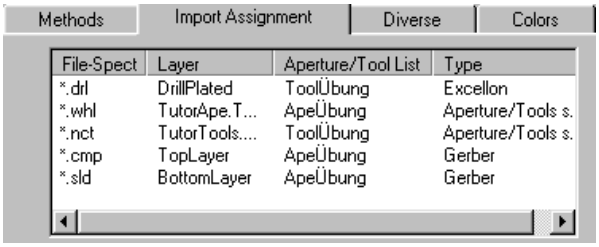
Repeat the works steps on the previous page, to define import instructions for the following production data:

	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 106:



› 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 Listbox LAYER or the Listbox TEMPLATE is indicated in the Dialogbox IMPORT ASSIGNMENTS.

9.2 Rotate layer

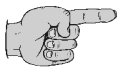
In exercise 5 you learned how to correct production data in which the layers did not coincide. This exercise include Excellon data which are offset against the Gerber data and are additionally rotated by 90°.

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:

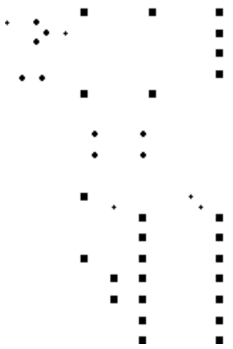
- „Tutor3.cmp"
- „Tutor3.sld"
- „Tutor3.drl"

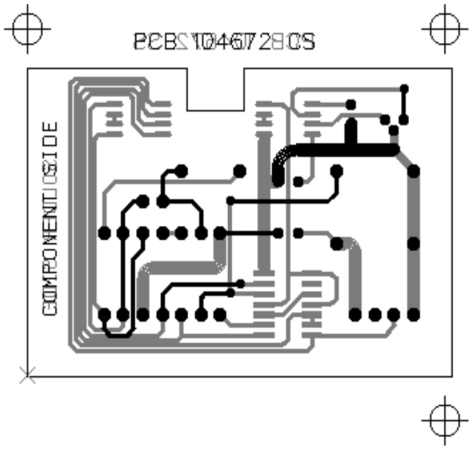


Note: Import the three files in a single operation. Thanks to the definition of the import assignments it is not necessary again to assign layers or tool and aperture lists in the IMPORT dialogue box.

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

Figure 107:





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 108:

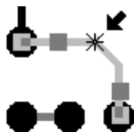
The effects of the various functions are illustrated by this example object

:

0°	MIRROR X
90°	MIRROR X+90°
180°	MIRROR Y
270°	MIRROR Y+90°

A selected object always rotates around the anchor point:

Figure 109:



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:

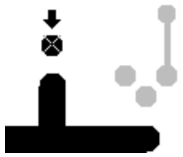
Figure 110:

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

Figure 111:

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 112:

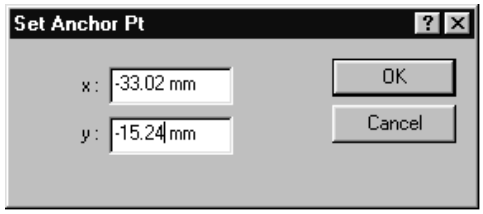


You can also position this anchor point completely independently of any object:

› Select **SET ANCHOR POINT...** from the **EDIT** menu.

The **SET ANCHOR POINT** *dialogue box showing the current coordinates of the anchor point will be displayed:*

Figure 113:



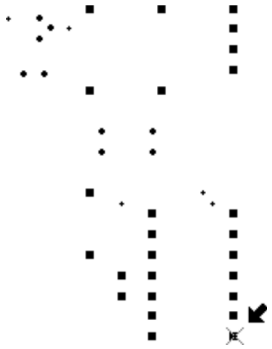
› Enter the new coordinates for the anchor point in the text boxes and click on **OK**.

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 114:



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:*

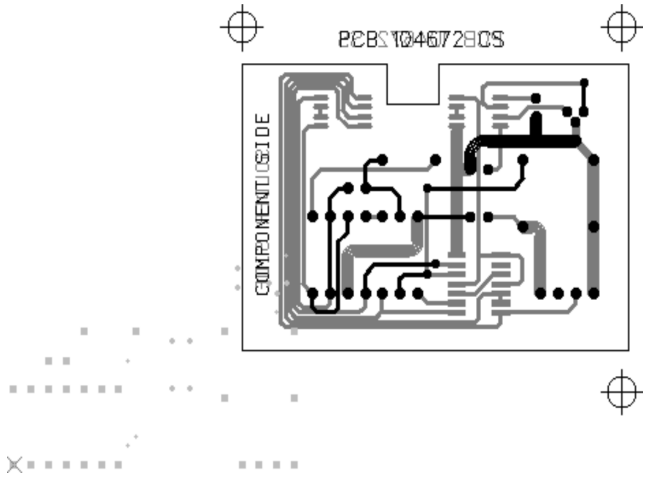
Figure 115:

› 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 116:



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.

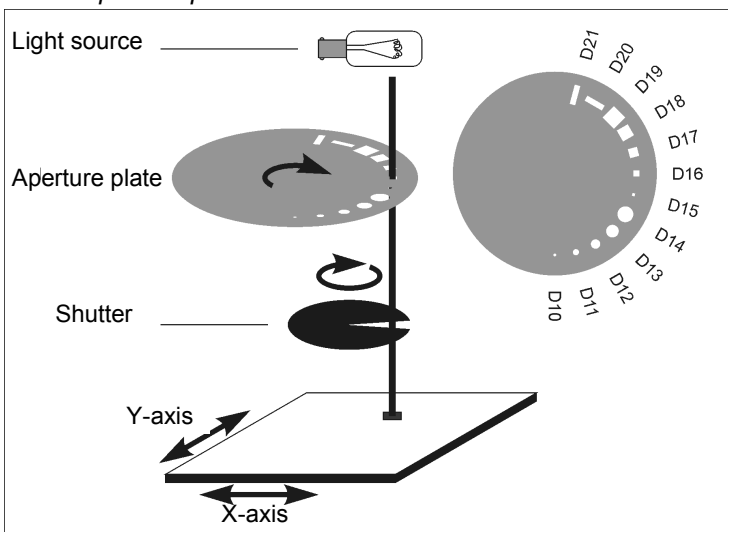
10.1.2 Structure of a Photoplotter

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 117:Schematic layout of photoplotter

Light source Aperture plate Shutter Y axis X axis



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 consists 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,0.10000X0.10000***%
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

0.10000X0.10000 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

		Template file				
Layer name	**Use**	**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	*Rubout* surface Component side	X		X	X	X
RUBOUTBOTTOM	*Rubout* surface Component side	X	X	X	X	X
RUBOUTALLLAYER	*Rubout* 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

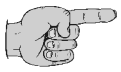
When importing DXF files, the problem frequently arises that the data does not contain surface information, but rather only outline data which may also be split up into individual line segments. In *CircuitCAM* such data can be converted quickly and easily into surface data with the aid of the “Combine 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 Y 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 118:

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

Figure 119:

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

Figure 120:

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

Figure 121:

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 TextFunction

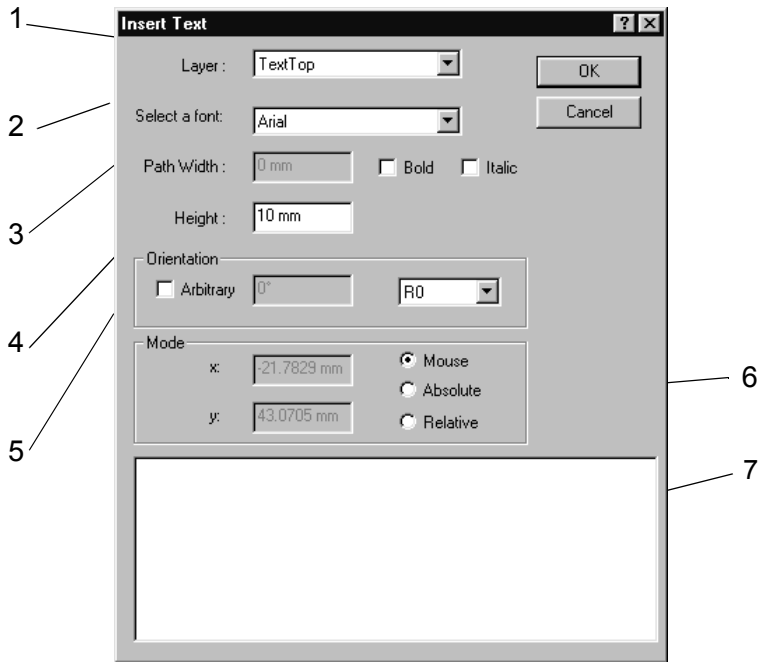
With *CircuitCAM* 4.0 it is possible to place texts on any desired layer. In practical applications, this is usually meaningful only on the top or bottom layer.

All true type fonts available on the PC can be used and selected in the pulldown menu **SELECT FONT**.

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

The following window appears:

Figure 122:Text window



1- **Layer**: List box for selection of layer on which text is to be located.

2- **Select a font**: Here; it is possible to select the font for the text.

3- **Path width**: Here, the path width for the text can be specified in mm.

4- **Height**: Here, the height of the text can be specified in mm.

5- **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.

6- **Mode**: Radio buttons for setting the text position with the mouse (absolute or relative)

7- **Text entry window**: Here, you can enter the text to be positioned on the layer.

› 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.

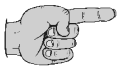
The entry cursor is activated and flashes in the text entry box.

› 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.

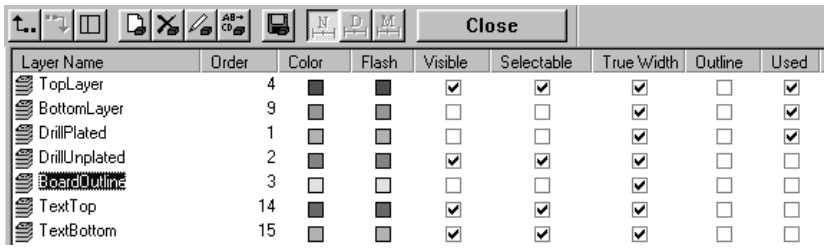
The following instructions are an example. In this case, the same file was used which was used for the main exercises. If other files are used, the circuit board representations naturally look different.

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 123:Layer for ground surface

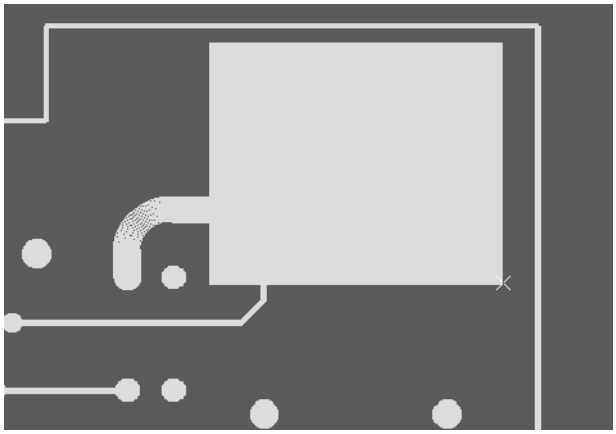


› Select the **TOPLAYER** as the active layer (See illustration below).

Figure 124:Top layer

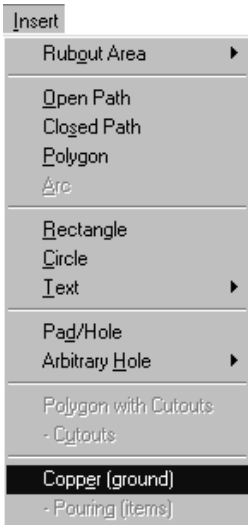
› Construct the desired ground surface with the **RECTANGLE** or **POLYGON** function. Then press **ESC** to quit this function.

Figure 125:



› 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.

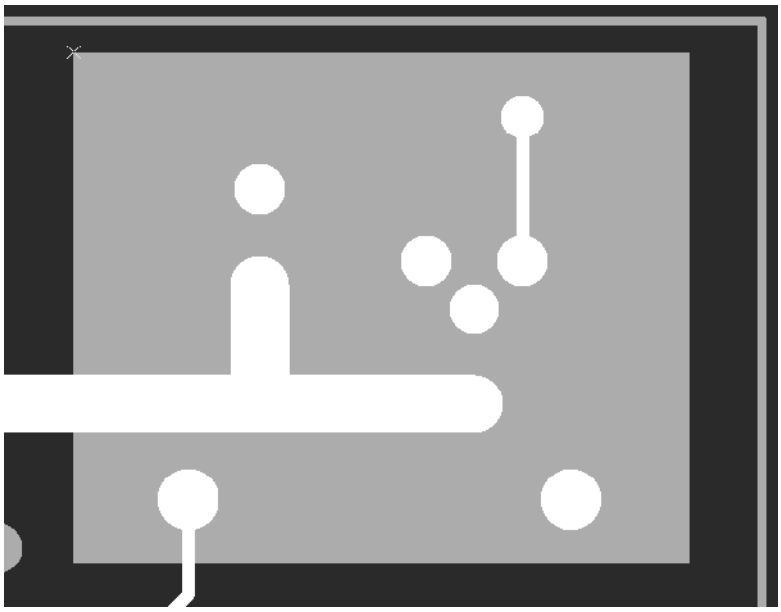
Figure 126:



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

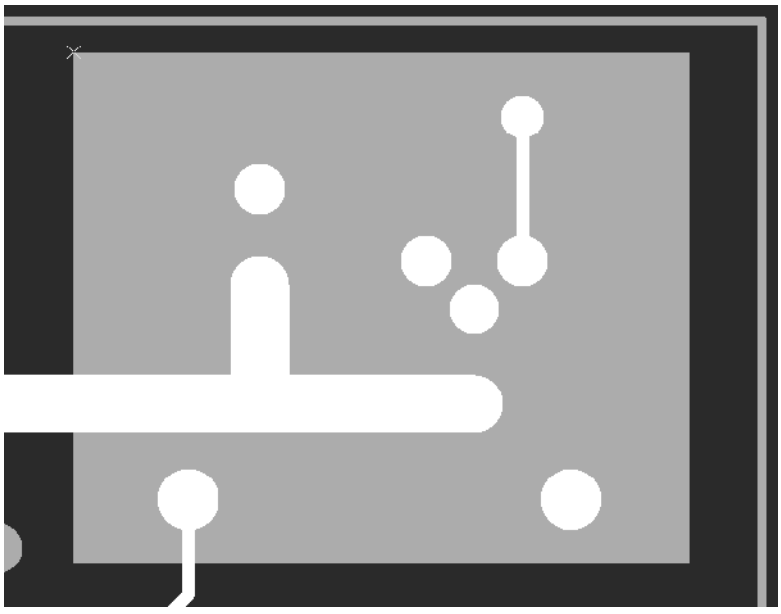
In this manner, all elements which are to be released from the ground surface are marked.

Figure 127:



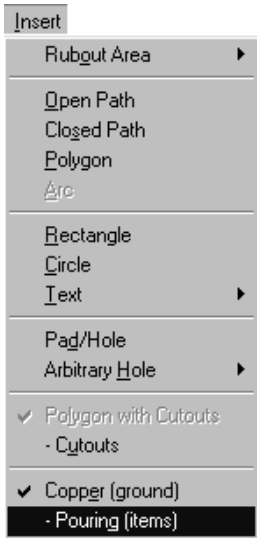
› Now deselect all elements not removed, i.e. which are to be connected to ground. For this purpose, press the **SHIFT** key and click the left-mouse key in the vicinity of the element in question.

Figure 128:



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

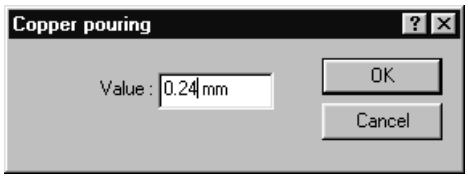
Figure 129:



› 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.

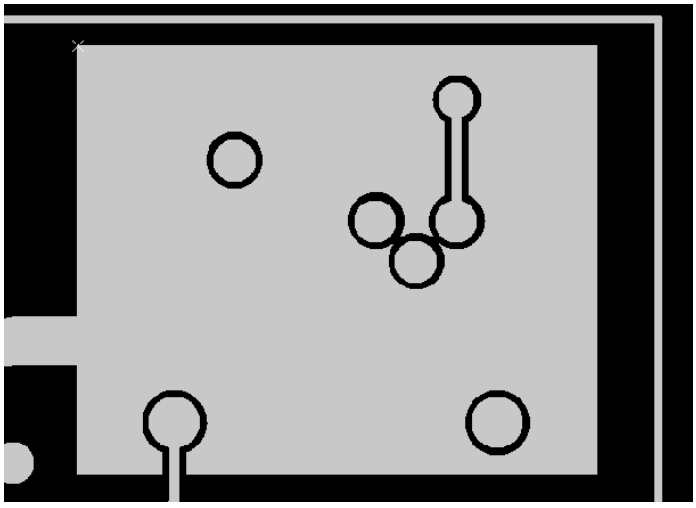
Note: This value should be adapted to the insulating tool to be used, e.g. for the universal cutter 0.2mm, the value should be set to 0.24 mm.

Figure 130: Copper pouring



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

Figure 131:



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.

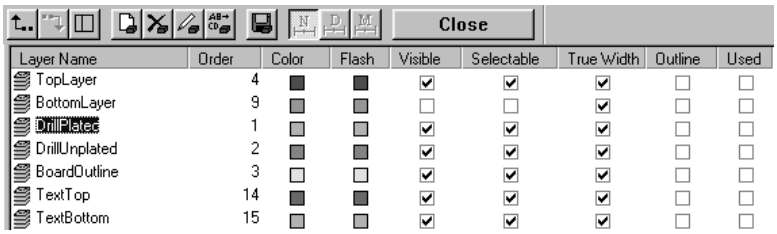
Proceed as follows to familiarize yourself with this function.

› 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 132:



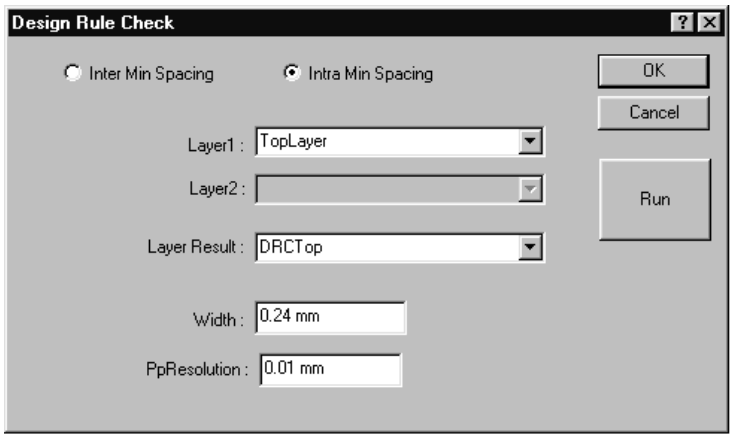
1. Checking the interval within a layer:

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 133: 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**.

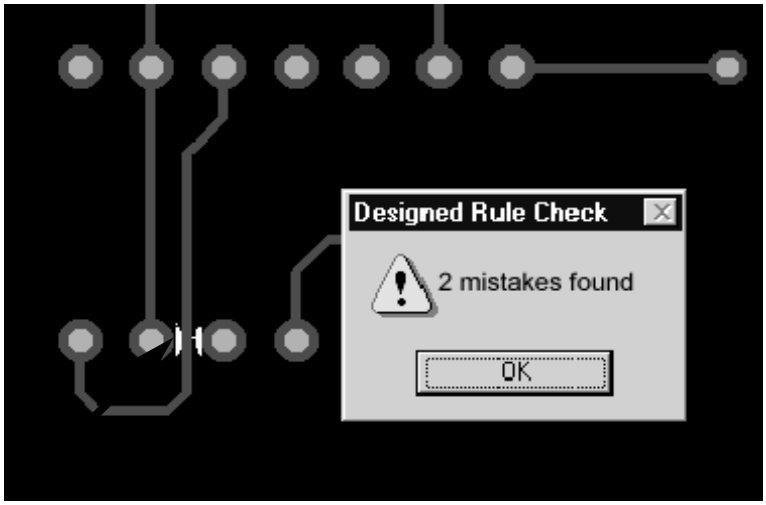
If the possible use of a milling tool is to be checked, enter a value approx. 0.02 mm larger than the diameter of the tool in order to avoid later flaws resulting from rounding errors when insulating.

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

› Click **Run**

The following message appears:

Figure 134: 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).

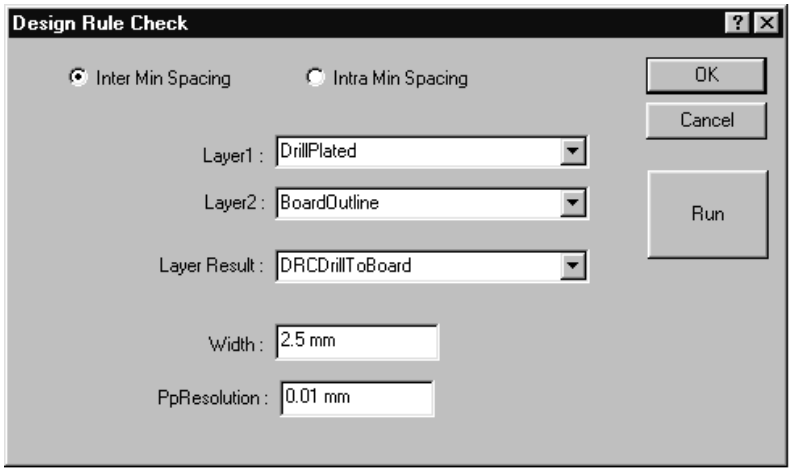
2. Checking the interval between 2 layers:

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 135: 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.

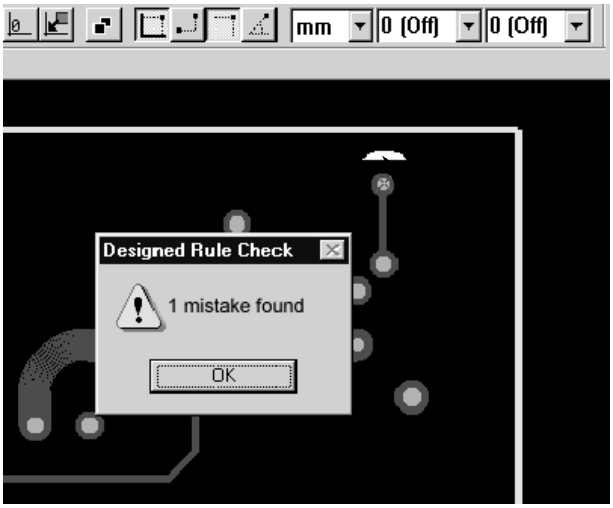
› Enter the minimum interval to be checked under **WIDTH**.

› Select the value **2.5 mm** here to provoke errors.

› Click **RUN**.

The following message appears:

Figure 136: 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“**.

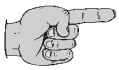
Template file	**Purpose**
Default.cat	Default template file
SingleSidedTop.cat	Single-sided PCB (component side only)
SingleSidedBottom.cat	Single-sided PCB (solder side only)
DoubleSided.cat	Double-sided PCB
4Layer.cat	Multilayer circuit board, 4 layers
6Layer.cat	Multilayer circuit board, 6 layers
Tutor.cat	Double-sided PCB With import assignment for Tutor data
Eagle.cat	Double-sided PCB With import assignment for Eagle data
Eagle40.cat	Doppelseitige Leiterplatte Mit Import-Zuweisung für Eagle 4.0 Daten

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

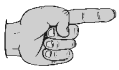
Figure 137:

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 dialogue 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.

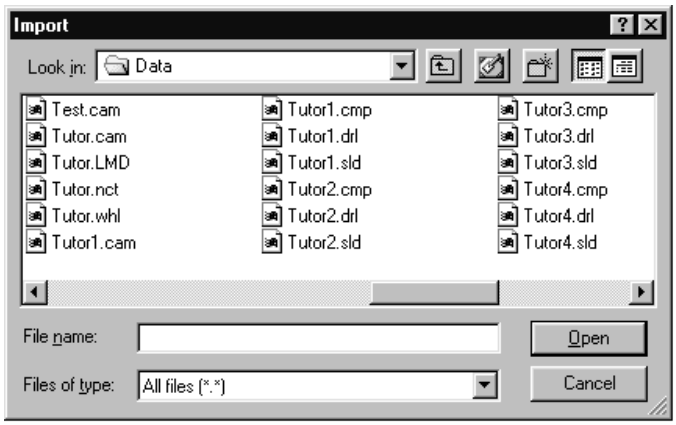
All subsequent changes will, however, be saved to your own.cat file. This .cat file is available at the start of a new project in the File, New menu.

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 138:



› 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.

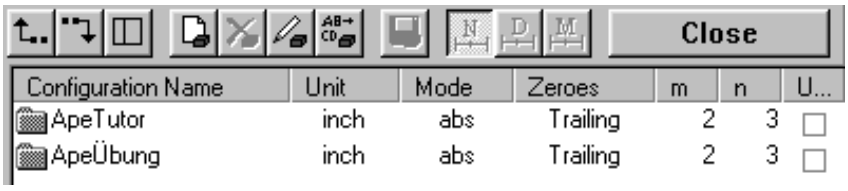
The file **TUTOR.WHL** is only an example in this case. You should carry out the same procedure with your own aperture list in future.

› To import the aperture list **TUTOR.WHL**, make all the required settings in the **IMPORT** dialogue 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 139:

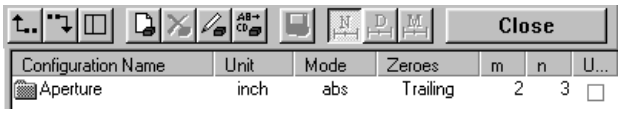


Let us assume that you only want to use your **APERTURES** list in future. This would mean that all other lists can be deleted.

› Left click to select **"APETUTOR"** and click on .

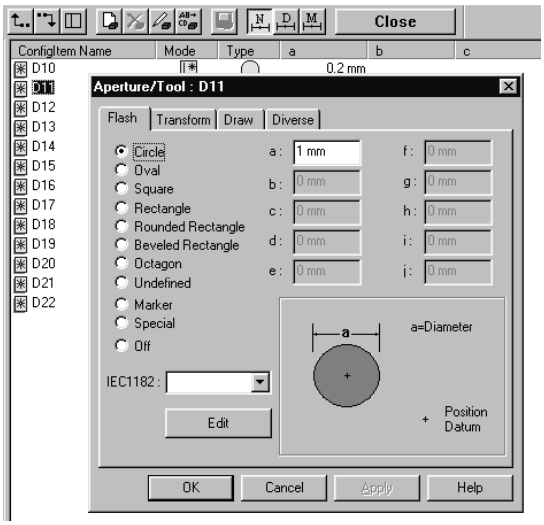
This action will delete this aperture list.

Figure 140:



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 141:



By clicking **OK** you quit this dialogue 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.

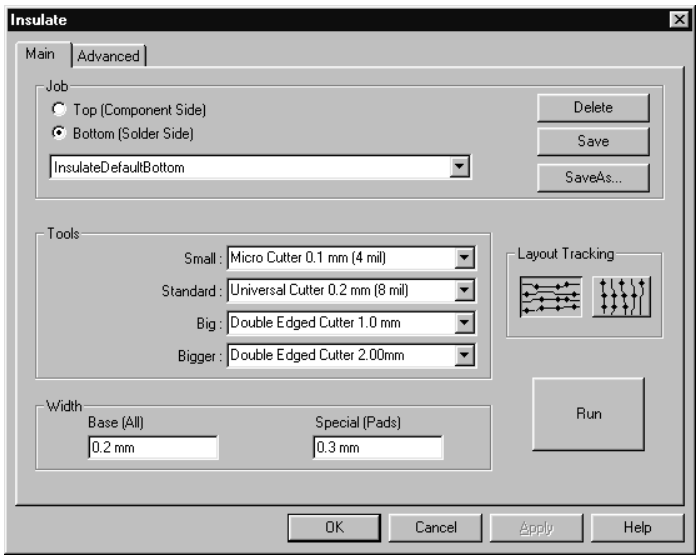
The list of all format configurations appears.

› 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 142:



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 **“INSULATEDEFAULTBOTTOM“** 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.

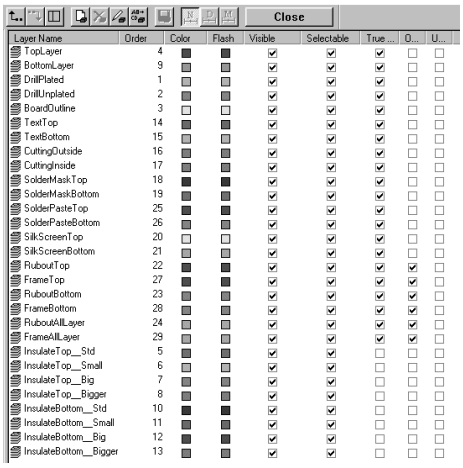
Clicking **OK** quits this dialogue box.

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.

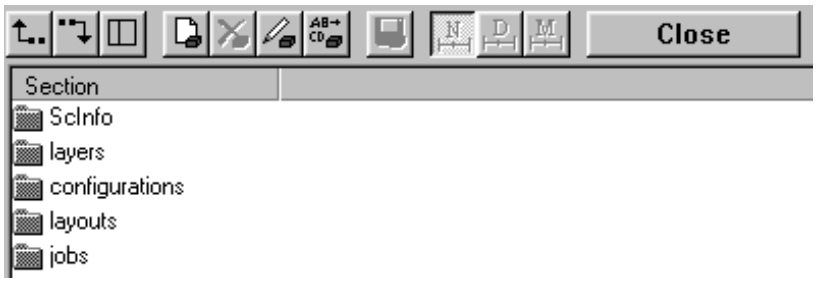
Figure 143:



In this list you may now modify the various parameters by clicking in the checkboxes, 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 144:



› 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

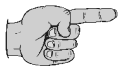
The settings for the parameters under General Settings are very personal and depend on the one hand on your individual preferences and on the other hand on the characteristics of the layout to be processed.

Open the **GENERAL SETTINGS** dialogue 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 dialogue 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	**InsulateBuildTop**
Isolation width		
Base	0.2 mm	0.2 mm
Special	0.3 mm	0.3 mm
Tool		
Standard	Universal Cutter 0.2 mm	Universal Cutter 0.2 mm
Big	End Mill Cutter 1.0 mm	End Mill Cutter 1.0 mm
Layer		
Base	BottomLayer	TopLayer
Special	- Flashed Pads Only -	- Flashed Pads Only -
Rubout1	RuboutBottom	RuboutTop
Rubout2	RuboutAll	RuboutAll
Summery	InsulateBottom	InsulatTop
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.

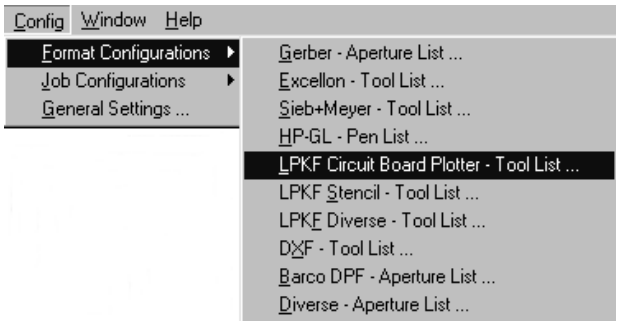
As a matter of principle, we differentiate between two types of tools: Drilling tools and milling tools for routing the contours of the circuit board or structuring the copper laminate.

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**:

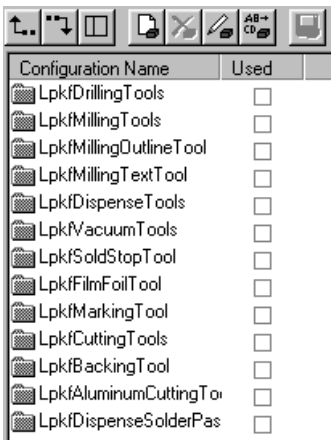
› Click on **FORMAT CONFIGURATIONS ..._LPKF CIRCUIT BOARD PLOTTER TOOLS LISTS** in the **CONFIGURATION** menu (see illustration below)**.**

Figure 145:



The following list window appears:

Figure 146:



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

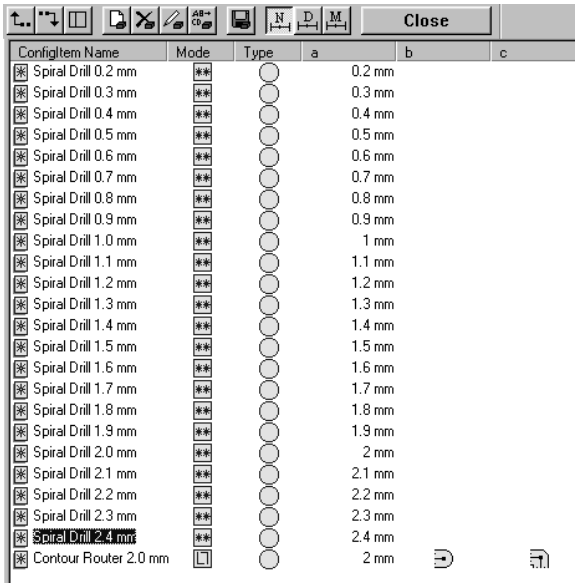
LpkfDrilingTools	contains all tools for drilling the circuit board
LpkfMillingTools	contains all tools which can be used for insulating
LpkfCuttingTools	contains all tools for circuit board contour routing

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 theentry *LpkfDrillingTools.*

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

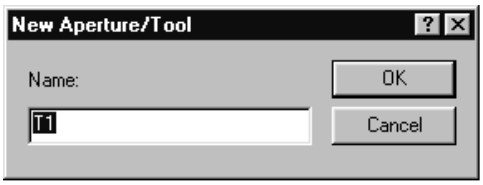


A drilling tool with a diameter of 2.5 mm is to be added to the list as an example.

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

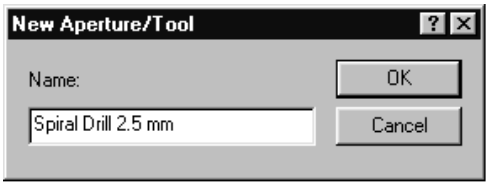
The following window appears:

Figure 147:



› Enter the name of the new tool:

Figure 148:

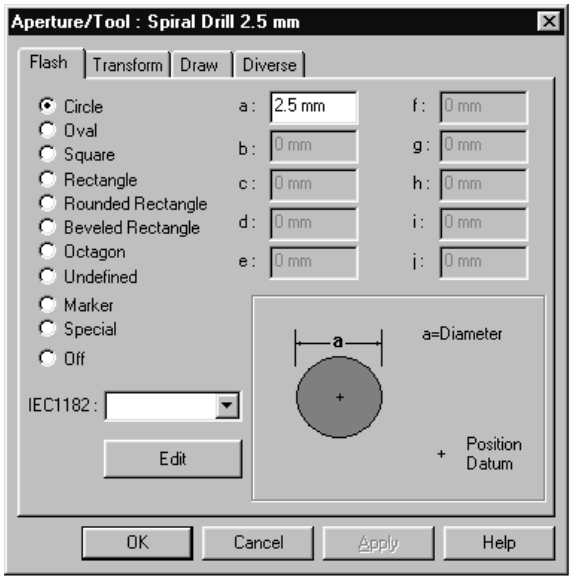


› Confirm the entry with **OK**

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 149: 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).

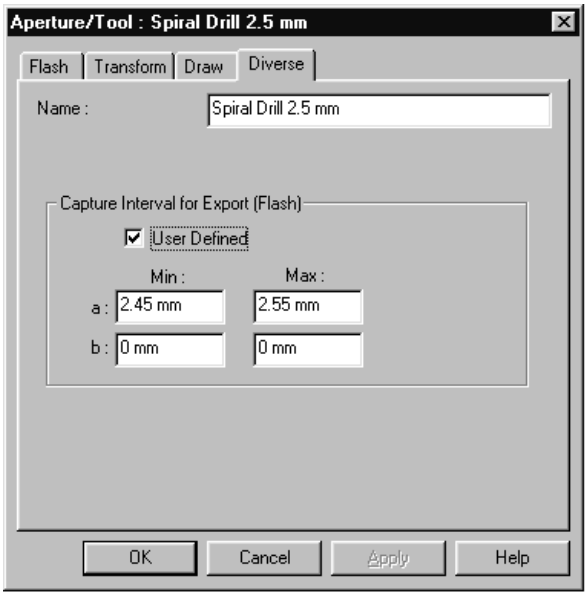
Note: This dialog window is also used for definition of highly varying apertures in Gerber format; for this reason, a large number of different shapes are specified here.

› Enter the desired drill diameter under a:

› Click the **DIVERSE** tab and enter the rest of the parameters.

The following window appears:

Figure 150:



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).

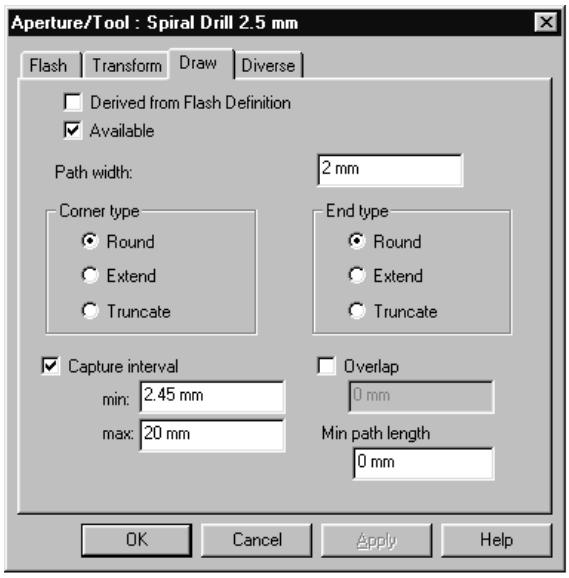
A capture interval of 2.45 to 2.55 mm is defined for the 2.5 mm drill. This means that all circles with a diameter of 2.45 mm (inclusive) up to 2.55 mm (exclusive) are assigned to this tool.

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 widow for the *2 mm Contour router* with a right-mouse click on this entry in the list.

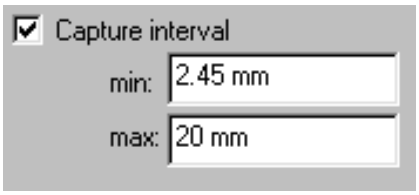
› Select the index card Draw in the dialog window.

Figure 151:



Change the values under capture interval according to the definition of the new drilling tool:

Figure 152:



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**.

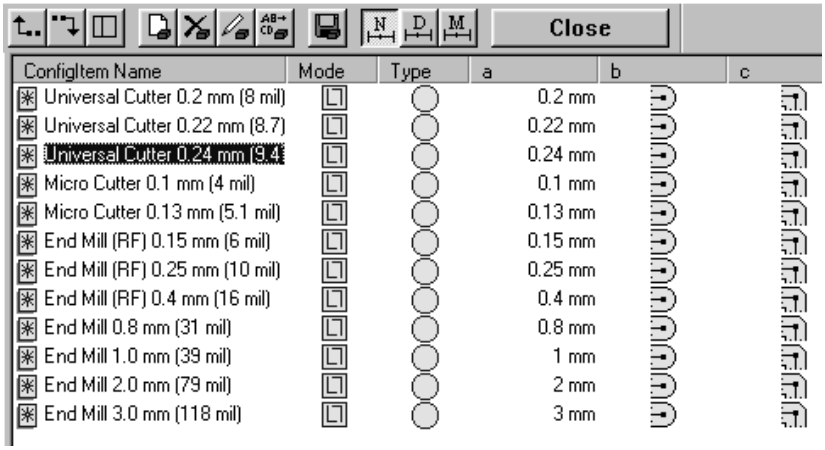
If the change is to remain stored for future layouts, save the new tool list as template file (see “Creating CircuitCAM template file” on Page 91).

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 153:

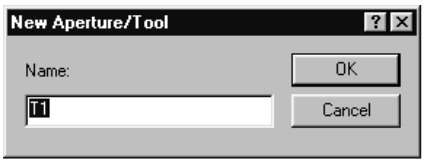


As an example, a new milling tool with diameter of 0.26 mm is to be added to the list.

Mark the milling tool, **below** which the new tool is to be added and click the button .

The following window appears:

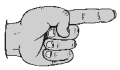
Figure 154:



› Enter the name of the new tool:

› Confirm the entry with **OK.**

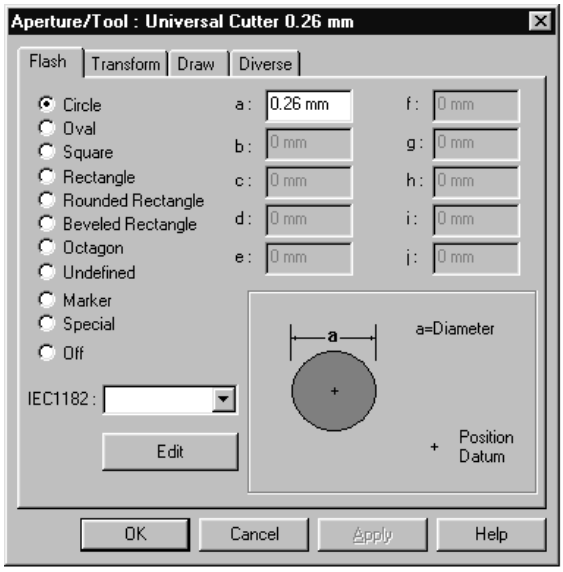
The new tool appears in the list.



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 155:



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 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 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 156:

This symbol in the tool list describes the tool as a drilling (Flash) tool as well as milling (Draw) tool.

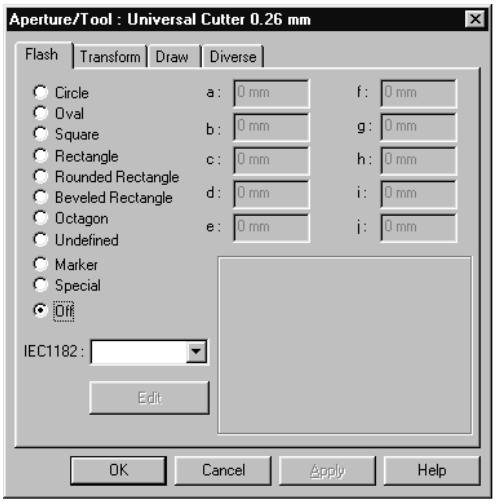
This symbol in the tool list describes the tool as a drilling (Flash) tool.

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.

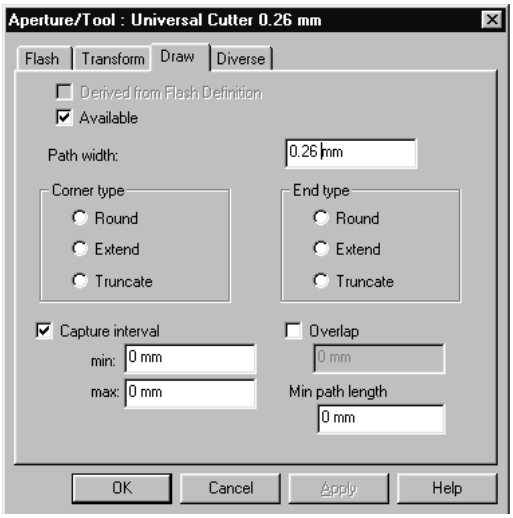
Figure 157:



› Click the radio button **OFF** on the *Flash* index card. This switches off the drilling (Flash) characteristics for the tool.

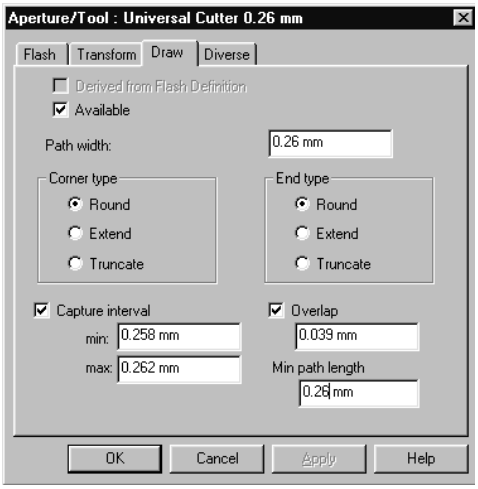
› Now click the *Draw* index card. The following box appears:

Figure 158:



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 159:



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.

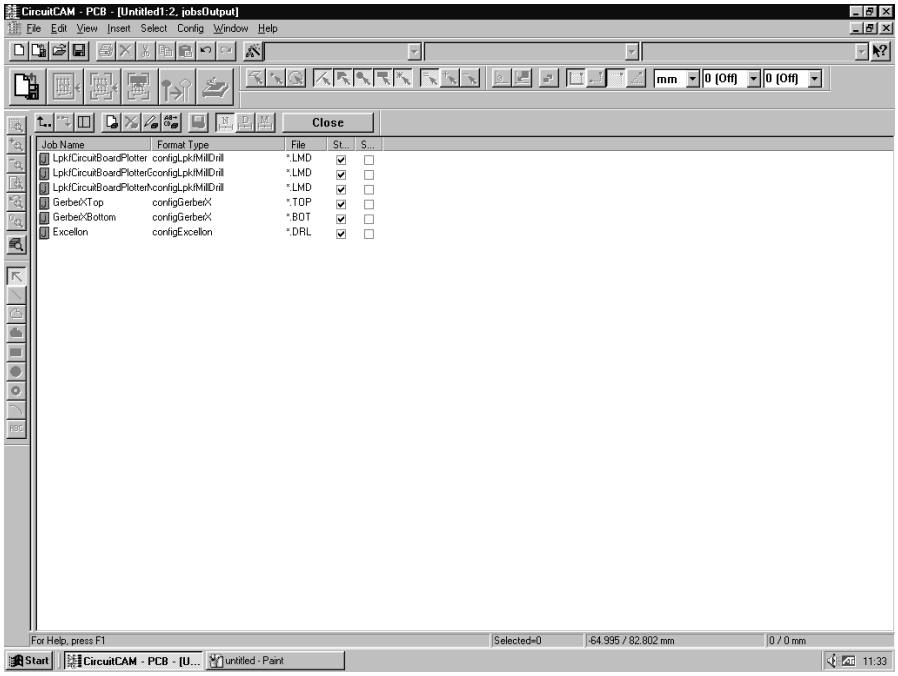
Use one of these jobs to output the production data you have generated and provide it for use by LPKF *BoardMaster* or other software.

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.

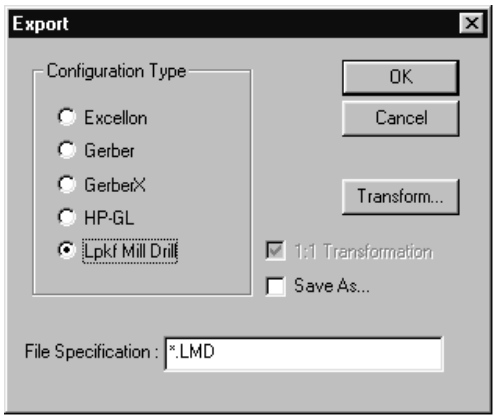
Figure 160:GerberXTOP,- Bottom + Excellon Jobs (only in the PCB-Version)



› Click the entry **Lpkf Circuit Board Plotter** with the right-mouse key.

The following window appears:

Figure 161: Export Job

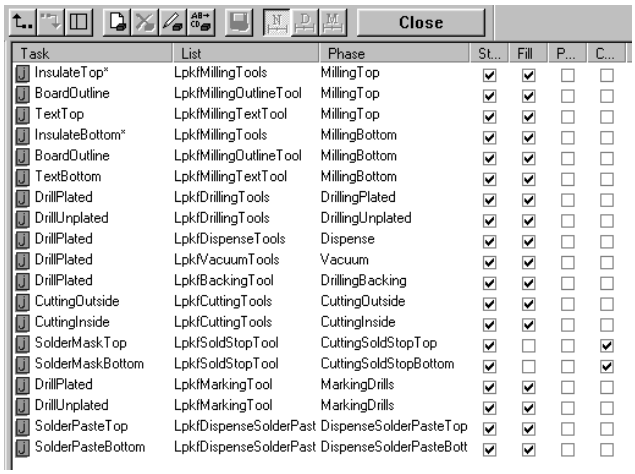


The format in which the data is to be exported, the name of the exported file and whether the data is to be turned, mirrored or shifted (transformed) during export are defined in this window.

› Click **OK** to close the window.

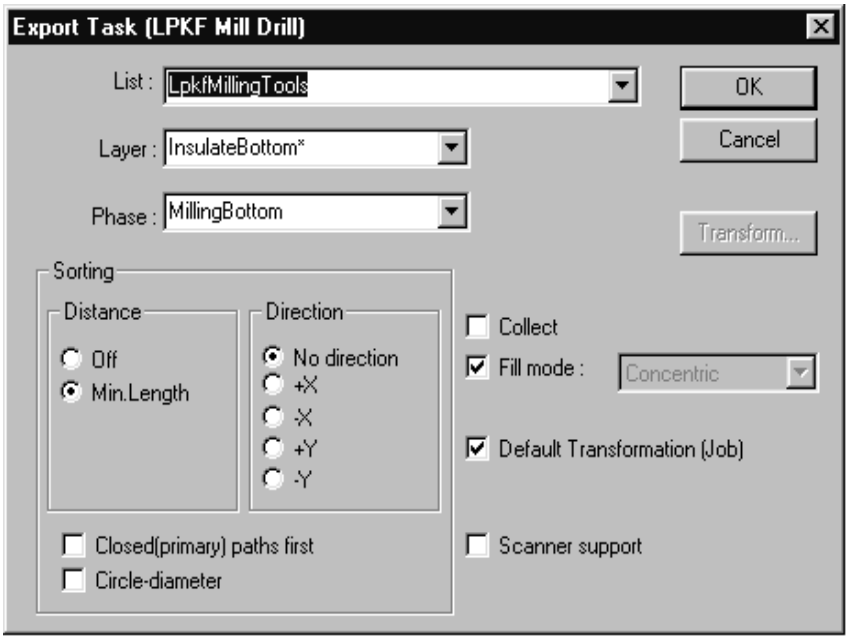
› Double click the entry **“LPKF CIRCUIT BOARD PLOTTER”** to display the various entries (tasks) with the assigned tool list, layer and production phase:

Figure 162:



Now open the following window by clicking the desired job with the right-mouse key:

Figure 163: Export Task (LPKF Mill Drill)



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 dialogue window where you can customize the export task to meet your requirements.

Note: You will find a description of the various production phases in the LPKF BoardMaster Manual.
Click on "Transform" to open the Transform dialogue box and modify the origin, scaling and alignment used to output the relevant layer.

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 saveed 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.

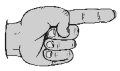
Another advantage: infrequent or less experienced users can use CircuitCAM with very little information, such as where and how to start the batch mode.

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:

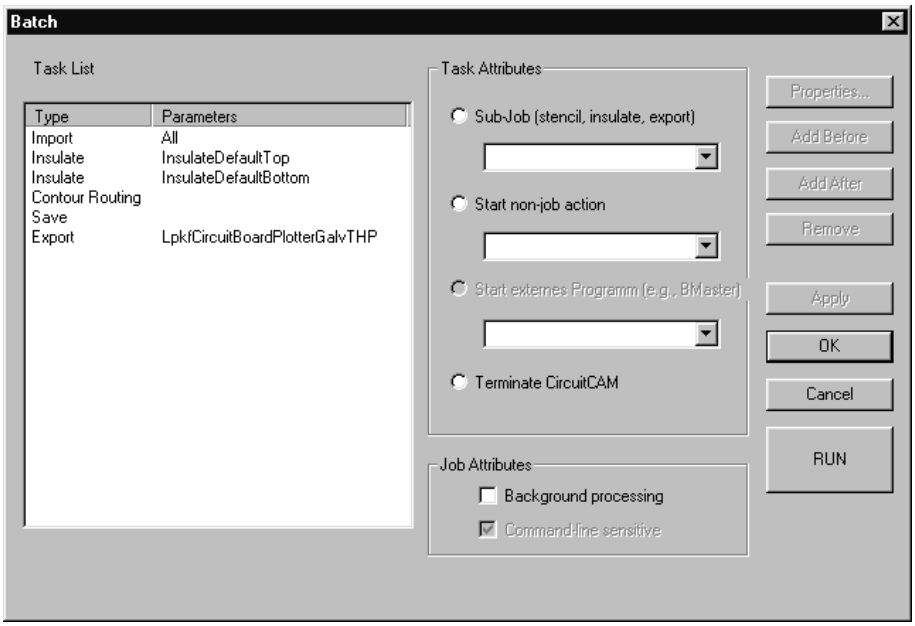
Figure 164:

The following list window is opened:

Figure 165:

› The batch dialog window can be opened by clicking the entry **Default** with the right-mouse key:

Figure 166:



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.

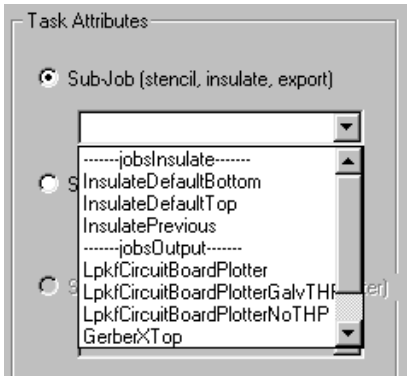
1. Sub-Job tasks:

› 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 167: 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

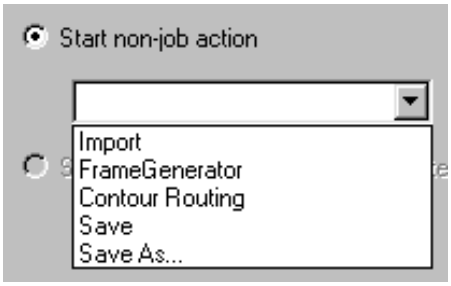
› Mark the task before or in back of which the new task is to be inserted.

› 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 168:

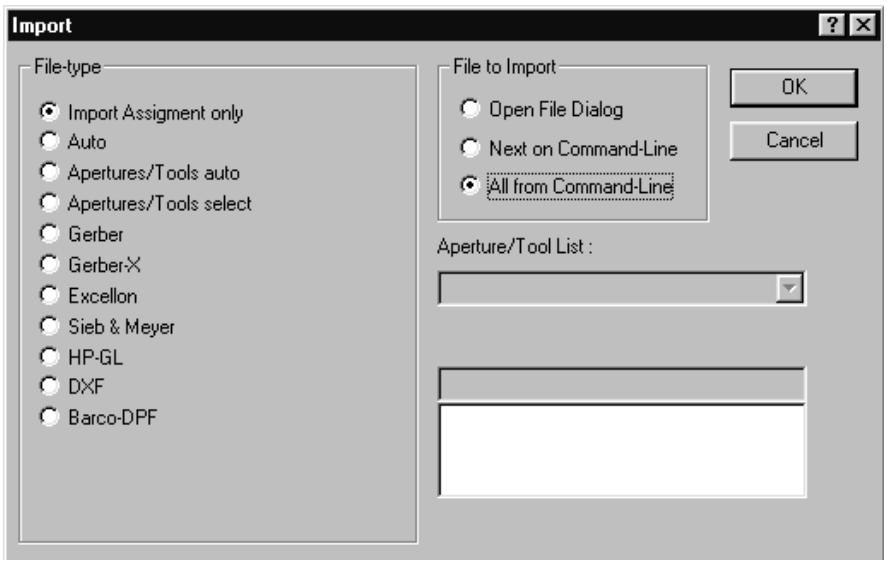


For **Non Jobs**, a dialog window is then opened automatically to define the properties of the job:

Import: Properties

After creating an import task, the following dialog window appears:

Figure 169: 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").

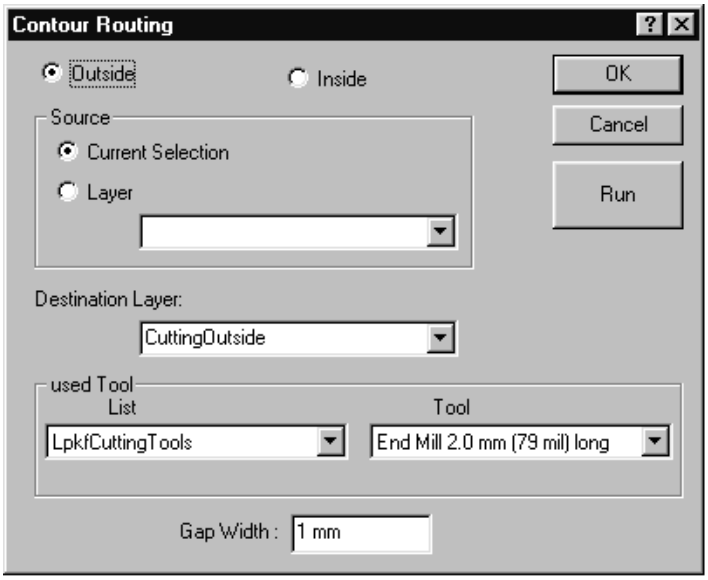
› Click **OK.**

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 170: Contour Routing

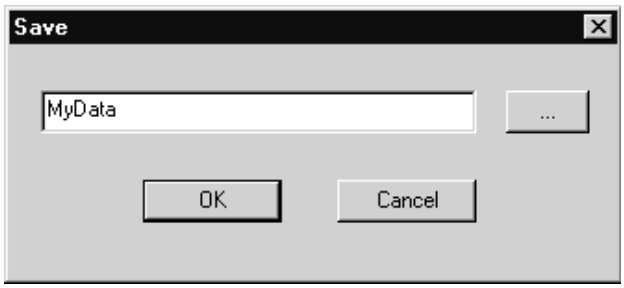


Here you can see the contour routing settings already familiar from manual work. You can make the settings according to your desires or requirements.

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



› 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 bottom on the desktop. The command line for the button must contain all required parameters such a file name and path.

e.g. : C:\LPKF40\CCAM\CircuitCAM.EXE C:\LPKF40\DATA\Sample.TOP C:\LPKF40\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.

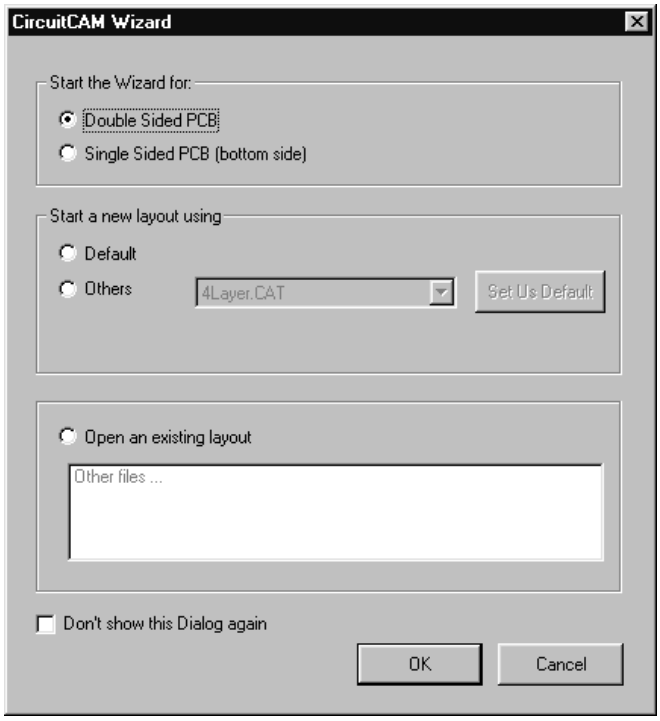
- The two-fluted end mill is a particularly suitable tool, given its good splinter removing properties.

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.

The following window appears:

fig. 171:



› Click on **DEFAULT** and then close the window by clicking on **OK**.

CircuitCAM is opened.

The sample file **FRONTPANEL.CAT** must be loaded in order to make the correct tools available during the processing.

› To do this, select the file **FRONTPANEL.CAT** from the files shown under **EDIT**, **NEW** in the menu toolbar.

10.12.1 Prepare front plate outline data

› Select the layer **RuboutAllLayer** from the **Main** toolbar.

fig. 172: Main toolbar

1- Layer
2- Aperture list
3- Aperture

› Click on the button ”Rectangle“ in the **Graphics** toolbar (figure 173).

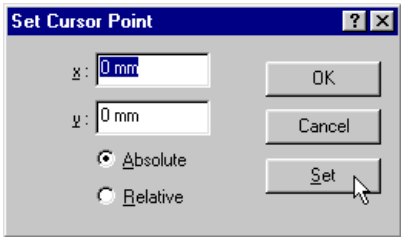
fig. 173: Graphics toolbar

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C**.

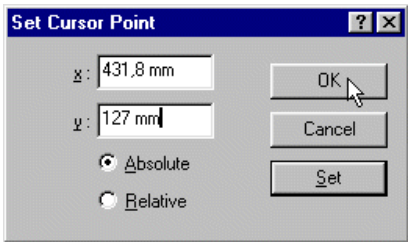
The following window appears:

fig. 174: Setting the cursor

1. Input



2. Input



› The following details have to be entered into the corresponding window:

x= 0 x= 431.8
y= 0 -> **SET** y= 127 -> **OK**

› Press **ESC** on your keyboard, to switch off the tool.

› Press the key **POS1** on your keyboard.

When the keys have been pressed, a zoomed overview of the data from the corresponding CamFile appears on the screen.
A dark yellow outline can now be seen on the screen.

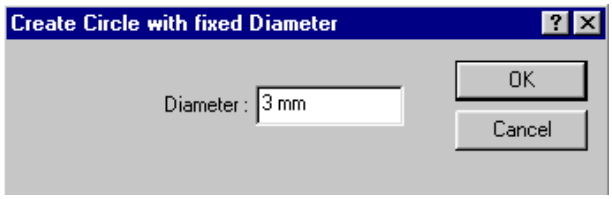
10.12.2 Preparing cutting and drilling data

› Select the layer **DrillUnplated** in the **Main** toolbar.

› Click on the button "Circle" in the **Graphics** toolbar.

› Enter a diameter of 3 mm (following figure) and close the window by clicking on **OK**.

fig. 175: Diameter



› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C** on the keyboard.

› Enter the following values in the **SET CURSOR** text field:

x= 7 x= 7 x= 424.8 x= 424.8
y= 3 -> **SET** y= 124 -> **SET** y= 124 -> **SET** y= 3 -> **OK**

› Press the **ESC** key on the keyboard.

› Click on the button "Circle" in the **Graphics** toolbar.

› Enter a diameter of 50 mm and close the window by clicking on **OK** (figure 175).

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C** on the keyboard.

› Enter the following values in the **SET CURSOR** text field:

x= 140
y= 63.5 -> **OK**

› Press the **ESC** key on the keyboard.

› Click on the button “Rectangle“ in the **Graphics** toolbar.

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C** on the keyboard.

› Enter the following values in the **SET CURSOR** text field:

x= 20 x= 40
y= 80 -> **SET** y= 105 -> **OK**

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C** on the keyboard.

› Enter the following values in the **SET CURSOR** text field:

x= 250 x= 300
y= 68.5 -> **SET** y= 108.5 -> **OK**

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C** on the keyboard.

› Enter the following values in the **SET CURSOR** text field:

x= 330 x= 380
y= 68.5 -> **SET** y= 108.5 -> **OK**

› Press **ESC**.

› Select the aperture list **Alutools** from the **Main** toolbar.

› Select the aperture **D10**.

› Click on the button “Pad / Drilling“ in the **Graphics** toolbar.

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C**.

› Enter the following values in the **SET CURSOR** text field:

x= 275
y= 25 -> **OK**

› Press **ESC**.

› Go to the **Grid Unit** toolbar and click on the button “Set zero point“.

› Select the layer **TextTop**.

› Select the aperture **D11**.

› Select again the tool “Pad / Drilling“.

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C**.

› Enter the following values in the **SET CURSOR** text field:

x= 0
y= 0 -> **OK**

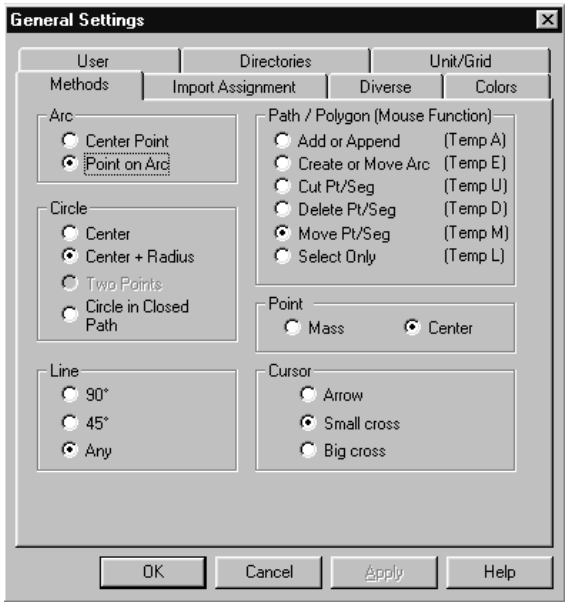
› Press **ESC**.

10.12.3 Preparing text and scale data

› Go to the menu toolbar **CONFIGURATION**, and select **GENERAL SETTINGS, METHODS**.

The following window appears:

fig. 176: General Settings



› Click on the value 45° in the field **LINE**, and then close the window by clicking on **OK**.

› Select the layer **TopLayer**.

› Select aperture **D12**.

› Click on the button “Open Path” in the **Graphics** toolbar.

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C**.

› Enter the following values in the **SET CURSOR** text field:

x= -15	x= 15	x= 0
y= 0 -> **SET**	y= 0 -> **SET**	y= 0 -> **OK**

To avoid having to alter the selection point manually, the last input value sets the cursor back to zero.

› Press **ESC** twice.

› Click on the button “Select layer objects“ in the **Main** toolbar.

› Go to the **EDIT** toolbar and select **DUPLICATE**, or press the keys **CTRL** + **SPACE**.

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C**.

› Enter the following values in the **SET CURSOR** text field:

x= 0
y= 0 -> **OK**

› Go to the menu toolbar **EDIT, MIRROR / TURN**, and select the value **90°**.

› Click on the button “Select layer objects“.

› Go to the menu toolbar **EDIT** and select **DUPLICATE**.

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C**.

› Enter the following values in the **SET CURSOR** text field:

x= 0
y= 0 -> **OK**

› Go to the menu toolbar **EDIT**, **MIRROR** / **ROTATE RELATIV, VALUE... ,** enter the value **45°** and close the window by clicking on **OK**.

› Click on the button “Select layer objects“.

› Go to the menu toolbar **EDIT, SET OPERATION** and select **MOVE TO STACK**.

› Select the layer **TextTop**.

› Click on the button “select layer objects“.

› Go to the menu toolbar **EDIT, SET OPERATION** and select **MOVE TO STACK**.

› Go to the menu toolbar **EDIT, SET OPERATION** and select **DIFFERENCE**.

› Go to the menu toolbar **EDIT, SET OPERATION** and select **PULL FROM STACK**.

› Use the mouse to drag a window over all the objects processed in the last drilling / scale operation, to select them.

› Go to **EDIT** in the menu toolbar and select **DUPLICATE**.

› Select **EDIT, SET CURSOR** from the menu toolbar (figure 174) or press the keys **SHIFT** + **C**.

› Enter the following values in the **SET CURSOR** text field:

x= 80
y= 0 -> **OK**

› Press **ESC**.

10.12.4 Setting text data

TrueType fonts can now be used from version 4.0 of *CircuitCAM*.

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).

fig. 177: Grid/Unit toolbar

1- Display Grid
2- Cursor Grid

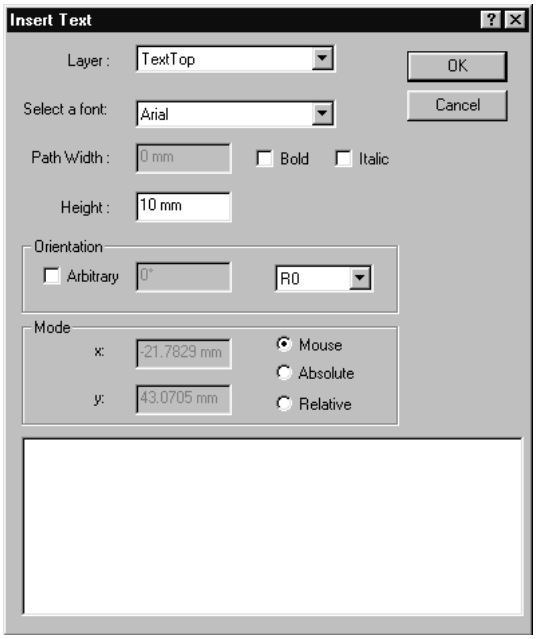
Note: The display grid and the cursor grid should be set to the same value.

› Select the layer **TextTop**.

› Select the text program by clicking on from the **Graphics** toolbar.

The following window appears:

fig. 178: Entering text



Enter the font type and the font height.

› Select “Arial“ for the font type and 5 mm for the font height.

› Enter the text (see figure 9 on page 8) and close the window by clicking on **OK**.

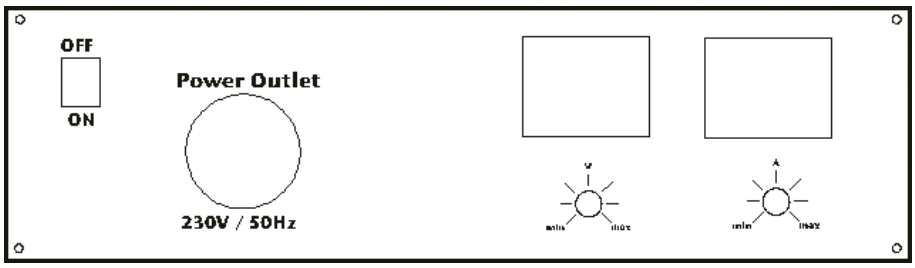
Use the mouse to position the text (use the keys **PAGE UP; PAGE DOWN** to enlarge the object, to help position the text correctly).

› Press **ESC**.

Repeat this procedure until all the text items have been entered.

The finished result might look like the following:

fig. 179: Result



If the outer contour on the panel is to be milled out, the contour generator must be used to set up a milling path for a cutting tool.

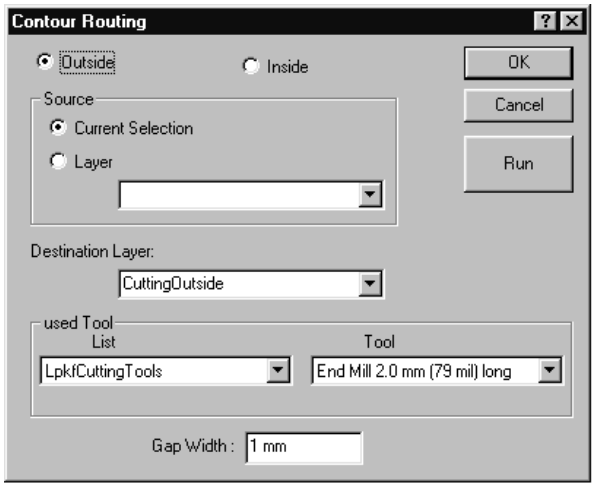
Note: Before doing this, the options “Display grid” and “Display cursor” in the Grid / Unit toolbar should be switched off again.

› Select the layer **RuboutAllLayer** and click on “Select layer objects“

› go to the **EDIT** menu toolbar and select **CONTOUR GENERATOR**.

The following window appears:

fig. 180: Milling an outline



› Make the selection as shown in the above figure.

› Click on **RUN**.

› Click on **ESC** to cancel the selection.

10.12.5 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.

10.12.6 Positioning the front panel on the ProtoMat

The prepared front plate is positioned with the long side against the alignment pin (Y axis) so as to ensure the correct angle of alignment. At least one drilling subplate must be beneath the front plate, since otherwise the milling would also affect the base plate. It is also advisable to fix all four sides of the front plate with adhesive tape.

› Start the *BoardMaster* software.

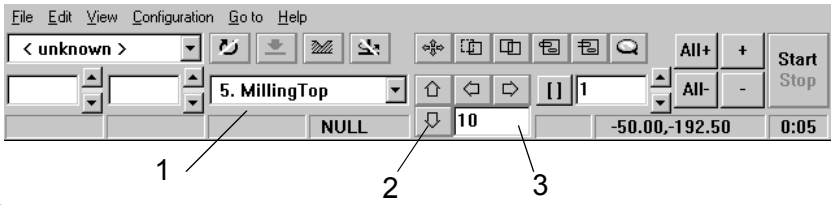
› Select **CONFIGURATION**, **TOOL LIBRARY**, **DRILLING/MILLING HEAD**.

› Click on the button **LOAD**.

› Go to **C:\LPKF32\BoardMaster** select the file **Alu-60.tol** and load it.

Note: The file Alu-60.tol must be loaded for the correct tool parameters to be available.

fig. 181:



1- Phase
2- Move buttons
3- Step width

› Use the move buttons in *BoardMaster* to move the middle point on the drilling spindle to the bottom right corner of the plate (as seen standing in front of the machine).

› Using the menu toolbar, select **CONFIGURATION**, **MATERIAL, SET X,Y-MIN**.

› Use the move buttons to move the spindle to the top left corner.

› Using the menu toolbar, select **CONFIGURATION**, **MATERIAL**, **SET X, Y-MAX**.

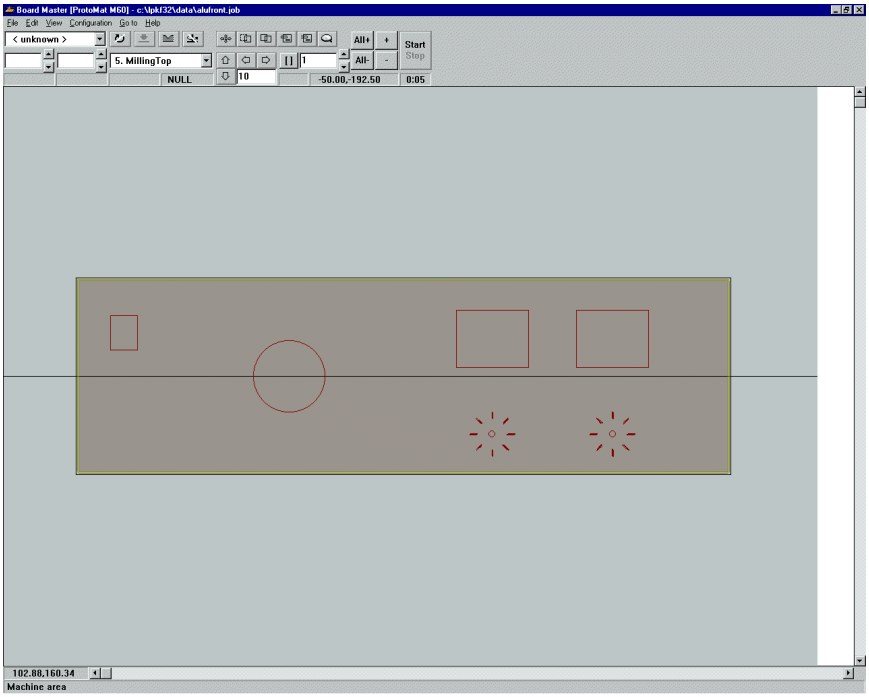
› Select **FILE, IMPORT** in the menu toolbar and then **LMD / LPR**.

› Select the **PHASE** (figure 181)**MILLING TOP**.

› Click on **VIEW, MACHINE** in the menu toolbar.

The screen should now look something like this:

fig. 182:



› Next click on the button **ALL+** and then on **START**.

The machine moves to the change tool position and asks for the tool "UniversalCutter 0.2 mm".

› Fit in the tool and then click on **STOP AFTER TOOL CHANGE FOR ADJUSTMENT**.

Use the working depth limiter to raise up the milling tool, so that it cannot penetrate the material.

› Click on **ALL+** in *BoardMaster* and then click again on **START**.

The milling head now moves to the first milling line, lowers down and starts to mill.

› The working depth limiter can now be used to set up the miller to an adequate milling depth.

Caution! The milling depth should only be half the diameter of the tool.

› First click on **STOP**, **ALL+** and then on **START**.

The machine now works along all the lines which have been assigned to this phase and tool; it only reports back again if another tool is required, if the standing time of the miller has been arrived at, or if all the lines have been processed.

The milling depth should be checked and set each time the tool has been changed.

10.13 CircuitCAM 4.0 Hotkeys

Key Combinations	Functions	Reference
General:		
F1	Help about current subject	
Ctrl + Tab	ToggeIn 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

10.14 Updapting 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.

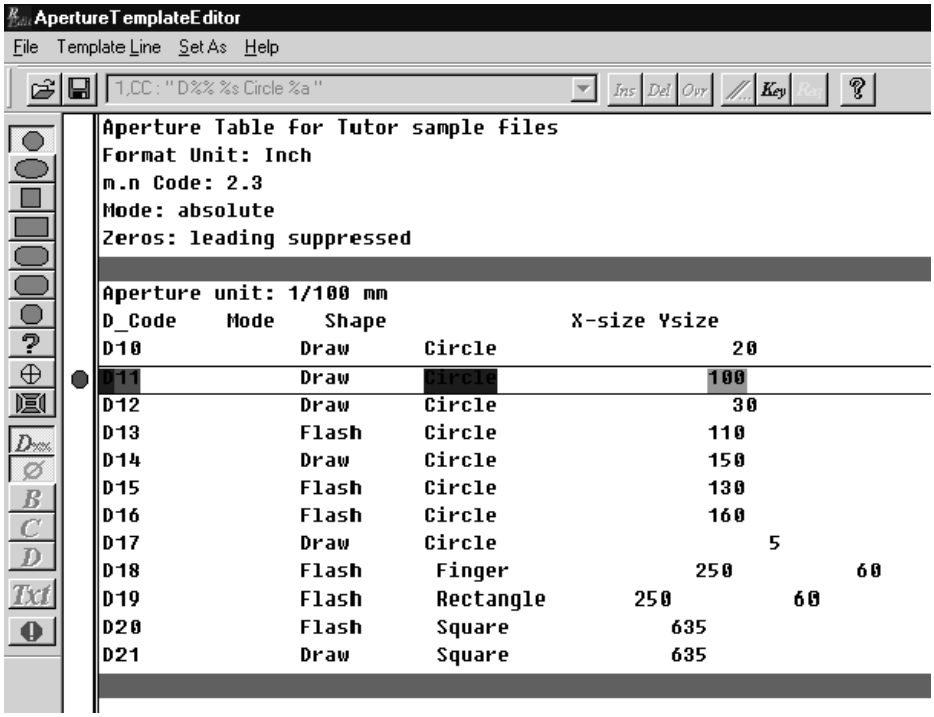
10.14.1 Structure of Translation Files

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 4.0 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 183: 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 *CircuitCAM*.	

Parameter <APE_FORMAT>

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.
.

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)

Fixed entries

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.

Tip: All entries after the last relevant parameter can be ignored. (There is no need to use the **%s** variable for this.)

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"
```

10.14.2 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.

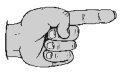
For example if conductors pads are displayed in the wrong shape or size, the aperture/tool definitions in the translation file do not correspond to the format of the aperture/tool list to be read in. As a remedy, the translation file can be adapted specifically to the aperture/tool list as described 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 15
Achtung 4
Action
  Cancel 53
Active layer 48
Aluminium front panel (creating) 152
Anchor point 98
  Move 52, 67
Ape_Templates 10
Aperture 21
Aperture List 106
Aperture list
  Edit 35
aperture list 128
ApeTutor 41, 91
Application window 19
ASCII file 31

B

Basic settings 126
Batch Function 146
Batch job (Executing) 151
BoardOutline 55
BottomLayer 40

C

CAD 44
CAM data 26
CAM format 3
Cancel function 28
CCAM 10
Check box 47
Circuit board
  Tutor 25
Circuit board CAD program 30
Circuit board outline 53
  Move to different layer 55
  Select 54
CircuitCAM 4
  Exit 24
  Start 13
  Version number 15
CircuitCAM - PCB 78
CircuitCAM Lite 74
Circular selection 51
Computer Aided Design (CAD) 3
Computer Aided Manufacturing (CAM) 3
Contacting LPKF 15
Context menu 20
Contour routing 64
Conventions 4
Coordinates display 29
Create new document 28

Cursor grid 30
Cursor keys 45

D

Data 10
  Import 30
Data export 62
Data select 31, 38
Data subfolder 26, 62
Default job 40, 58
Default settings 29
Design Rule Check 122
Display grid 30
Display unit 30
Document 28
Document window 19
DoubleSided.cat 91, 126
Draw 34
Drawing 104
Drilling Tool (Definition) 135
Drilling/Milling Tools (creating) 134
DrillPlated 41
DXF files importing 111

E

Error messages 21, 62
Exercise files 26
export jobs 143
Extended Gerber 35, 108

F

File type 32
Flash 34
Flash selection 51
Flashing 105
Format 107
Front plate positioning (ProtoMat) 161
FRONT TO END 22

G

General preferences 11
Gerber 102
  Extended 108
  Syntax 104
Gerber Format 102
Gerber format syntax 102
Gerber-X 35
Graphic mode 72
Graphic window 20
Graphical objects
  Mirror/Rotate 97
  Select 50
Grid 29
Ground plane generating 118

H

Hotkeys 164
HP-GL 62

I

Import 30
  Gerber data 38, 41
  Multiple files 38
Import assignments 94, 97
Incorrect display 173
InsulateBottom 58
InsulateDefault 74
InsulateTop 58
Isolate
  Isolation width 75, 76
  Primary overlap 77
Isolation grid 58
Isolation job 40
  Customize 74
Isolation jobs (prepared) 133

J

Job 133

K

Keyboard functions 44

L

Layer 21
  Active 48
  Colour 47, 90, 127
  Move 92
  Not selectable 54
  Properties 47, 54
  Registration 93
  Rotate 97
  Vordefinierte 108
Line selection 51
List window 21
  Aperture list 35, 37
  Layer 46
LPKF BoardMaster 58

M

Materials supplied 9
Milling contour 64
  Insert webs 66
Milling Tool-new (definition) 139
Milling-drilling plotter 62, 64
modified isolation jobs 130
Move graphical objects
  To different layer 48

N

Neuer Bildaufbau 45
New_Templates 10, 90, 126

O

Offset 100
On-line help 15
Order 48
Outline illustration 47
Overview 44

P

parameters ... 132
Personalize CircuitCAM ... 11
Photoplotter ... 103
POLYGON ... 69
Polygon selection ... 51
Predefined Layer ... 110
Preferences ... 11
  Folders ... 11
Prepared isolation jobs ... 133
Product ID ... 11
Production data ... 20, 28
  Checking ... 44
  File extensions ... 94
  Import ... 30
  Offset ... 92
  Output parameters ... 92
  Preparing for prototyping ... 50
Program group ... 13
Protolaser ... 80

R

Read Me file ... 13
Rectangle selection ... 51
Reference coordinates ... 93
Registration mark ... 50, 52
Release number ... 11
Report window ... 21, 62
Right mouse button ... 35, 47
RuboutBottom ... 71
RuboutTop ... 71

S

Saveing a new Layer ... 131
Scale ... 44
Selection ... 44
Selection mode ... 52, 72
Selection switches ... 51, 52
Selektieren ... 50
Set anchor point ... 99
Set cursor ... 70
SMD ... 25
Start Menu ... 13
Status bar ... 51
  Show/Hide ... 23
Subfolders ... 10
suppressed Zeros ... 107
System requirements ... 9

T

Template file ... 10, 28
  Create ... 90
template file ... 126
Template file creating ... 127
TextBottom ... 57
Textfunktion ... 116
TextTop ... 57
Tool ... 21
Toolbar
  Move ... 23

ToolTutor 36, 91
TopLayer 40
Translation file 10
Translation File (Updating) 166
Translation Files (Structure) 167
True width 47
Tutor.cat 91

U

Unit 49
Units 107
Untitled file 28
User interface 13
User preferences 11

V

Visual check 48

W

Wizzard 16

Z

Zoom in 44
Zoom out 44
Zoom window 44