LPKF *CircuitCAM* 3.2 Tutorial

Englisch-Version

Stand: 15.04.1999

Verwendete TrueType-Schriften in diesem Dokument:

- Arial
- Arial Narrow
- Symbol
- Wingdings
- Courier New

CircuitCAM

Version 3.2 (04/1999)

English Version

LPKF Laser & Electronics AG
Osteriede 7
30827 Garbsen
Germany

Tel.:   **49-(0)5131 7095 0**
Fax: **49-(0)5131 7095 90**
e-mail: **lpkf@lpkf.de**
Internet:http://www.lpkf.de

As at: 15.04.1999

Table of contents

About this manual

This manual is an introduction into the operation of *CircuitCAM 3.2* 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 learning about the installation of the *CircuitCAM* software and receiving a general introduction into the *CircuitCAM* user interface, you will be able to follow exercises introducing you to the basic operations and concept of *CircuitCAM*.

Conventions

The following conventions have been followed to assist your orientation within this manual:

SMALL CAPITALS	identify the name of a menu term, a dialogue box or an input field
BOLD SMALL CAPITALS	identify a list field entry or button-name
italic script	identifies a technical term or a proper name
bold italic script	identifies a keyboard input
<...>	identifies a key on the keyboard, eg **<F4>** is the label for function key F4

Note for newcomers to Windows:

This tutorial does not cover the basic operation of the Windows interface. Please first refer to your Windows documentation if you are not familiar with the techniques of handling Windows objects.

Note on the illustrations:

The illustrations are sometimes shown here in different colours from the way they appear on your screen to improve reproduction in this tutorial.

Installation

System requirements

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

IBM PC or compatible system with

- Pentium processor (Pentium or equivalent processor recommended)
- 64 MB RAM (128 MB recommended)
- VGA colour monitor (recommended resolution 1024 x 768 pixels)
- Microsoft Windows 95/98 or Windows NT 4.0 operating system
- 3.5" floppy drive
- (4x speed) CD-ROM drive

CircuitCAM requires approximately 5 MB free hard disk space for its installation.

Materials supplied

The materials supplied with *CircuitCAM* include:

- 4x 3.5" program disks or 1x CD-ROM
- this tutorial

Installation

CircuitCAM is installed from 4x 3.5" floppy disks.

Note:	Make sure that you have made a copy of all program disks and that you install from these copies.

How to install CircuitCAM:

1. Insert **program disk 1** or the CD_ROM in your floppy drive/CD drive.
2. Select RUN... from the WINDOWS START menu.
3. Enter the ***drive letter*** of your floppy drive followed by ***:setup*** in the text field of the RUN dialogue box
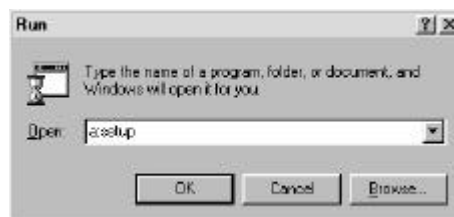
4. Click on "**OK**" to start the installation routine.

Note:	Clicking on "**CANCEL**" allows you to close the dialogue box and cancel the installation.

5. Follow the instructions of the installation program.

Folders created during installation

By default, the installation program creates a "LPKF 32" subfolder in your program folder; the programs such as *CircuitCAM* and *BoardMaster* are saved here.

The following subfolders are created in the "LPKF 32" folder for *CircuitCAM*:

- **CCAM**
  contains the program files for *CircuitCAM*.

- **Data**
  contains the data files for *CircuitCAM*. This is the default folder 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 folders for the conversion and template files to your requirements as described in the following section.

User preferences

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

Please enter the data required in this dialogue box.

Note: The **Product ID** and the **Release number** can be found on the
                   license card.

                   You may call up this GENERAL PREFERENCES dialogue box in
                   CircuitCAM at any time by selecting the GENERAL... line from the
                   CONFIGURATION menu.

Clicking on the FOLDERS tab allows you customize the *CircuitCAM* folders for the translation
and template files to your requirements:
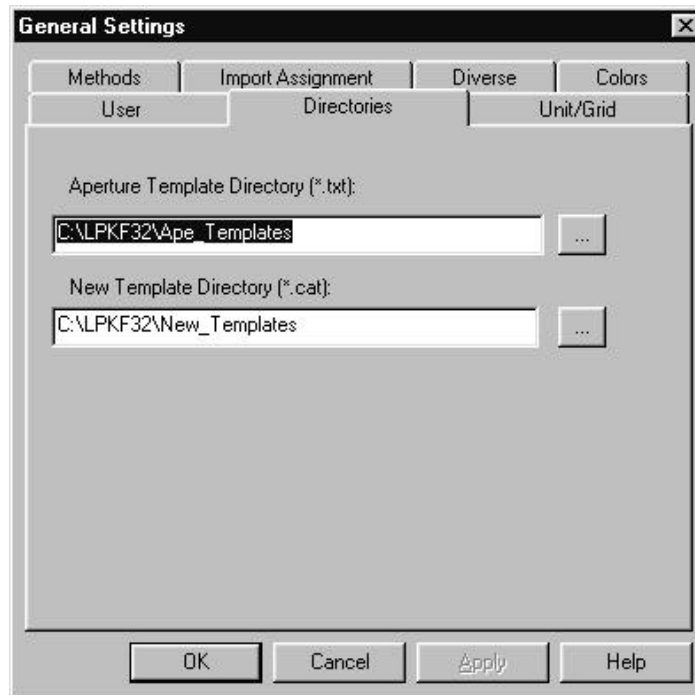
This settings will be saved in the Windows registry.

Getting started

In this chapter, we should 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 folders created.

Starting *CircuitCAM*

Once installation of *CircuitCAM* is complete, a new LPKF32 program group will be found under PROGRAMS in the Windows 95/98/NT START MENU; it will contain the following icons:

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 3.2 shortcut from the LPKF 32 program group in the START MENU.

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.

How to obtain help on the current function:

Press function key **<F1>**.

How to obtain help on any function:

1. Click on in the DEFAULT toolbar.
2. 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.

Show program information

You may display the *CircuitCAM* version number by selecting ABOUT *CIRCUITCAM*... from the HELP menu.
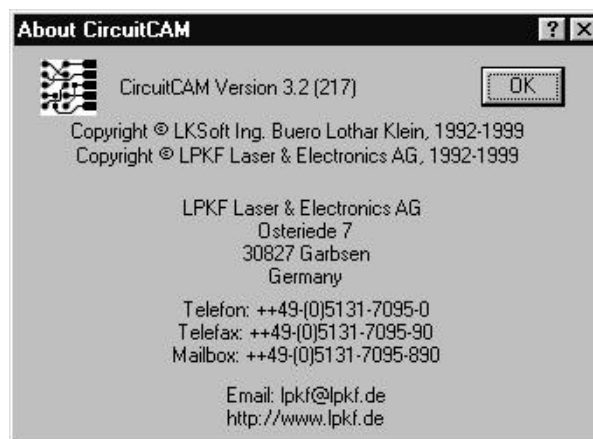
This dialogue box also contains information of how you may contact us.

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:

1. Select OPEN from the FILE menu.

2. Select the **TUTOR** file from the OPEN dialogue box.

Click on "**OPEN**" to open the file "Tutor.cam" or
click on "**CANCEL**" to cancel the process.

Tips & tricks:	If you have not yet started *CircuitCAM* you can open the sample circuit board together with *CircuitCAM* by double-clicking on the TUTOR icon in the LPKF 32 program group. The same is true for the TutorCPU" sample circuit board.

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.
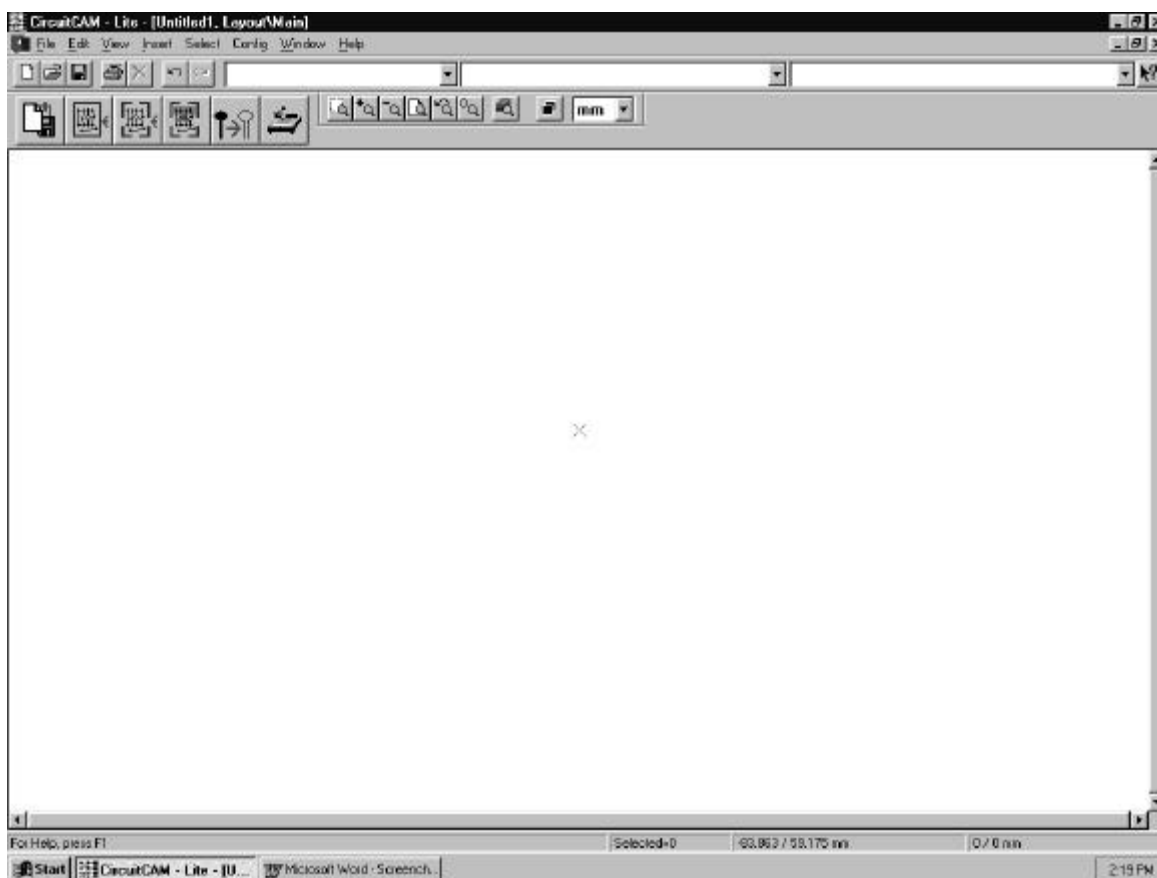
CircuitCAM has three types of document window, GRAPHIC WINDOWS, LIST WINDOWS and REPORT WINDOWS. When you start the program, the GRAPHIC WINDOW for the "empty" specification file is opened in full-screen mode.

> **Tips & tricks:** 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.

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. **The view is always of the upper side of the circuit board, so that the lower side of the board is illustrated as a mirror image.**
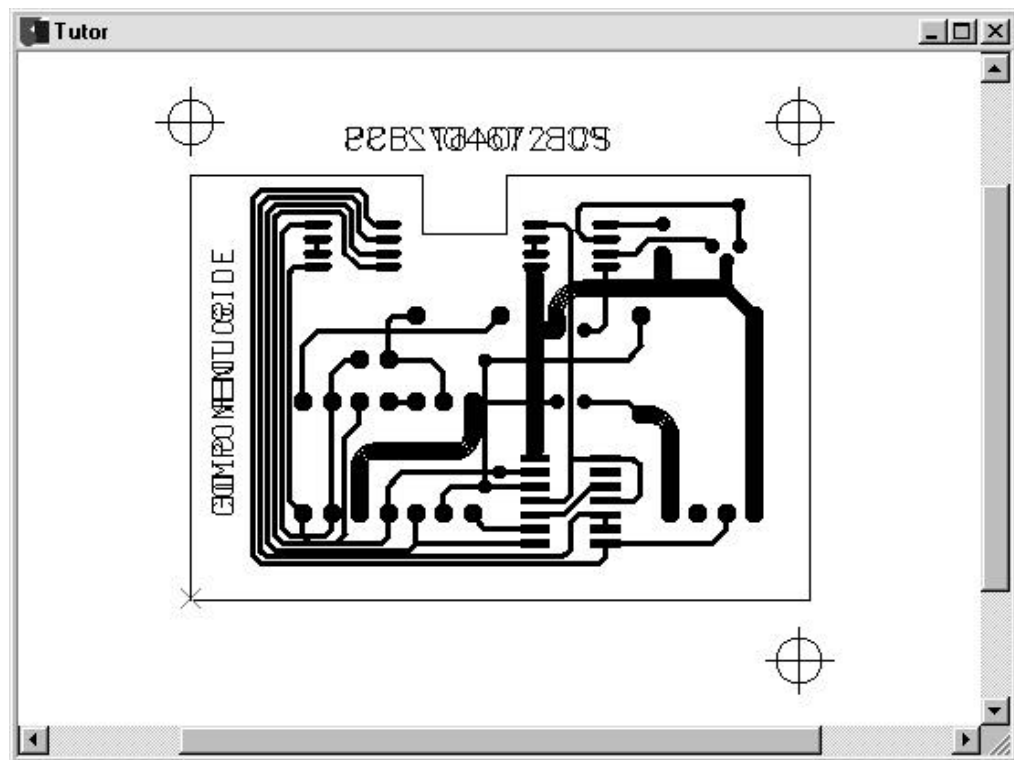
In addition, the graphic window gives you access to functions for editing the production data.

> **Tips & tricks:** When the mouse pointer is in the graphic window, clicking the right mouse button will access a context menu for editing selected objects:

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 or delete entries, or add new entries.
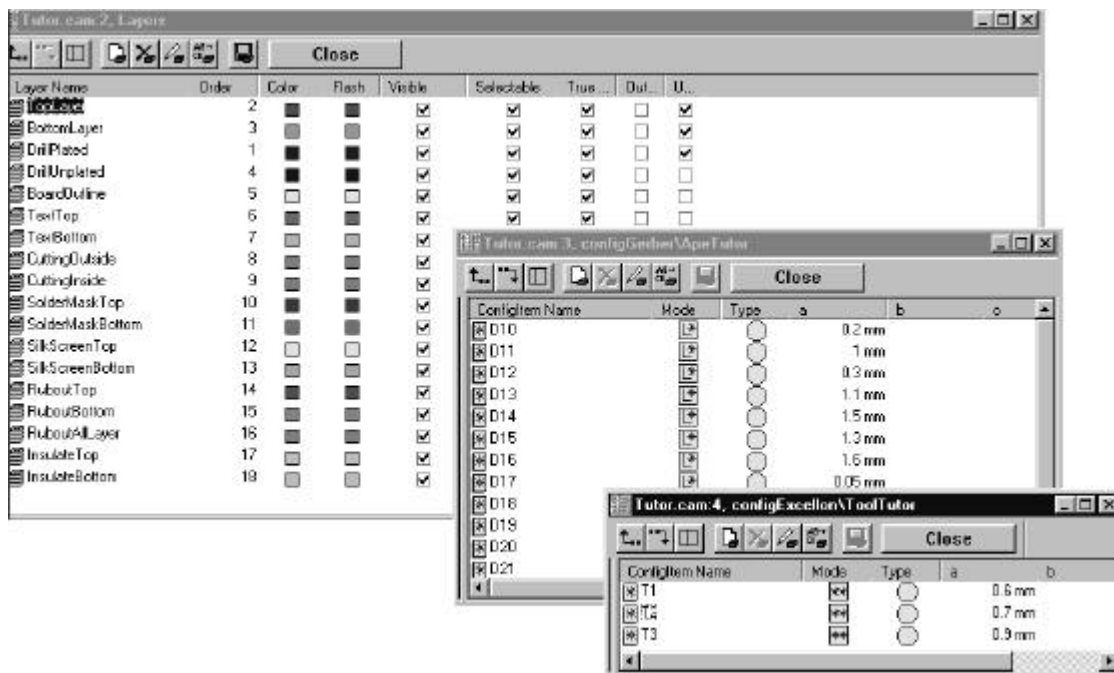
Report window

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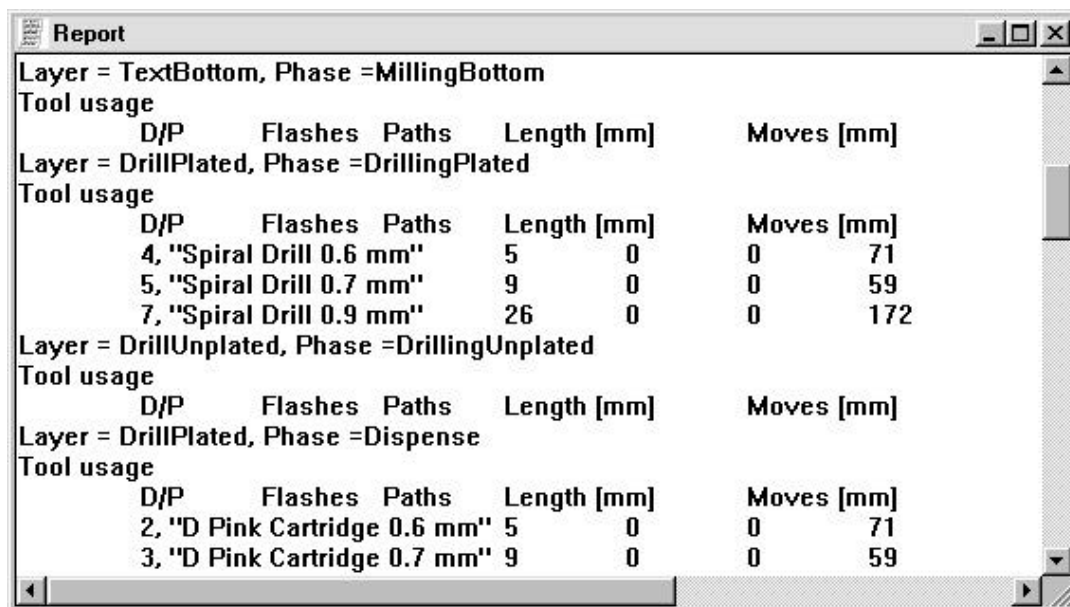
Toolbars

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

DEFAULT

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

Front to End
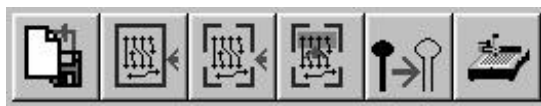
           All functions to the generating of milling- and drilling data
like importing, contour routing, webs insert,
Rubout milling, Insulate all Layer, exporting LMD - file

VIEW

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

GRID+UNIT

           Functions for customizing the coordinates display and list boxes for
unit, viewing grid and cursor grid

By selecting the toolbar VIEW\TOOLBARS the following toolbars can be switched on:

SELECT

           Functions for marking graphical objects

GRAPHICS

           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 of the VIEW menu.

How to move a toolbar:

1. Place the mouse pointer on the area between the buttons on the toolbar or on the title bar of a floating toolbar.

2. 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.

Status bar

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

Few objects are selected	Selected=16	50.8 / 25.4 mm	33.02 / 24.13 mm
①	②	③	④
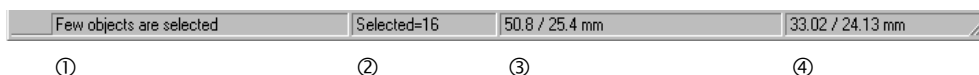
The status bar is divided into four information areas:

    ① Textual information

    ② Number of objects selected

    ③ Current coordinates of the cursor

    ④ 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.

Closing *CircuitCAM*

How to exit CircuitCAM:

using the keyboard:

Press <ALT><F4>.

using the mouse:

Select EXIT from the FILE menu
or
click on ☒ in the top right-hand corner of the application window.

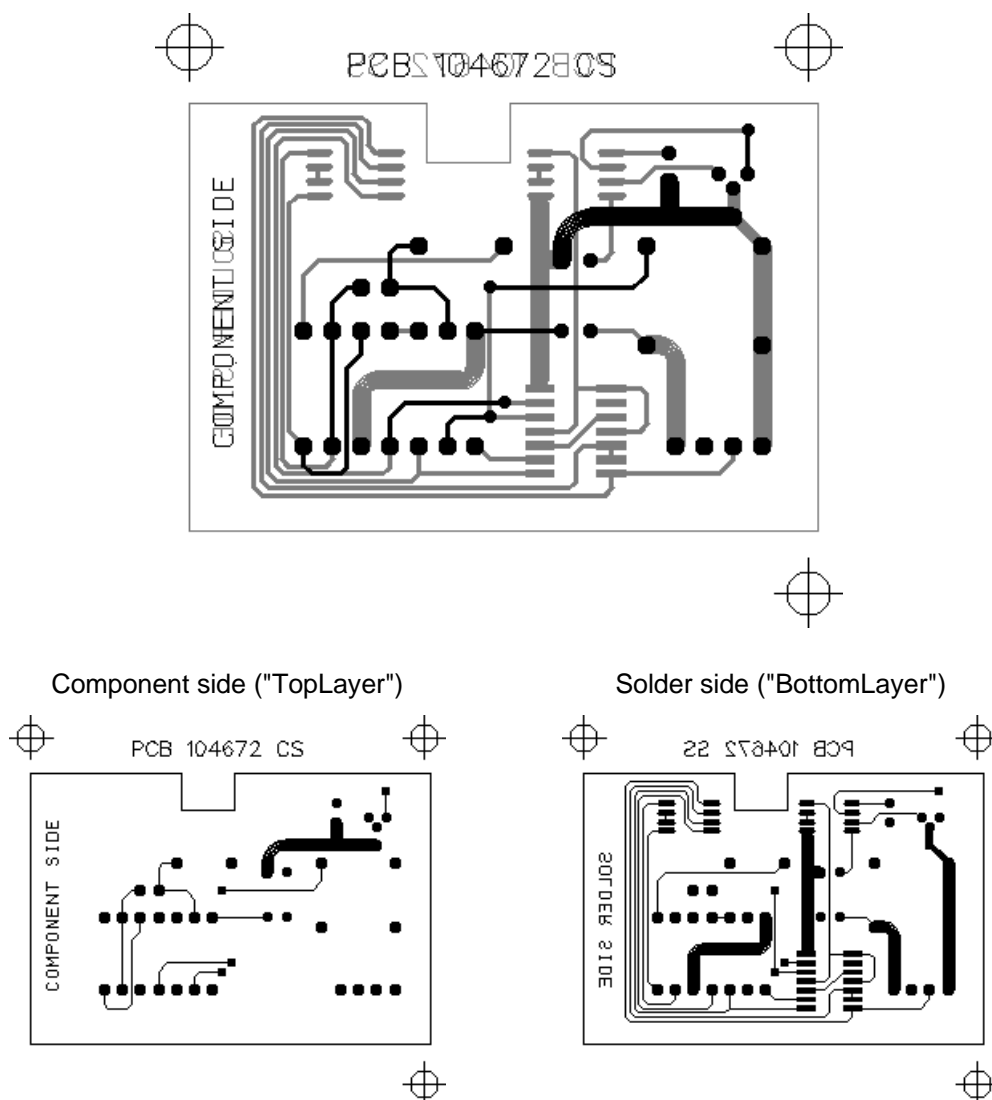# Preface to the exercises

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.

"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, with the SMD components located on the solder side, that is on the underside of the circuit board:

Component side ("TopLayer")   Solder side ("BottomLayer")

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* subfolder during installation.

Production data for the "Tutor" sample circuit board:

TUTORAPE.TXT	Translation file for the aperture lists (Ape_Templates subfolder)
TUTORTOOLS.TXT	Translation file for the tool lists (Ape_Templates subfolder)
TUTOR1.CAM	CAM data, *CircuitCAM* version 3.2 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
TUTOR4.CMP	Gerber data for the component side
TUTOR4.SLD	Gerber data for the solder side, data mirrored
TUTOR4.DRL	Excellon data

In addition, you will find production data for the "TutorCPU" sample circuit board in the "Data" folder:

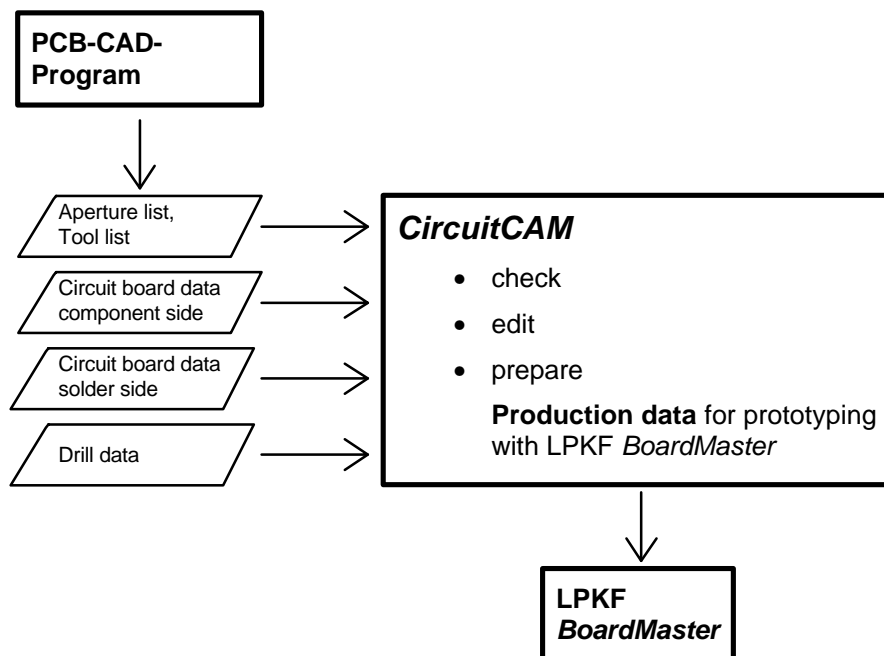|	
TUTORCPU.WHL	Aperture list for Gerber data
TUTORCPU.NCT	Tool list for Excellon data
TUTORCPU.CAM	CAM data, *CircuitCAM* version 3.0 format
TUTORCPU.CMP	Gerber data for the component side
TUTORCPU.SLD	Gerber data for the solder side
TUTORCPU.DRL	Excellon data

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

Exercise 1

In this section you will become familiar with the complete sequence of operations required to prepare production data for passing to LPKF *BoardMaster*.

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

- Import production data

- Prepare production data for prototyping

- Export production data

```
┌─────────────────┐
│ PCB-CAD-        │
│ Program         │
└─────────────────┘
         │
         ▼
   ╱Aperture list,╱ ──────►   ┌──────────────────────────────┐
  ╱ Tool list    ╱            │ CircuitCAM                   │
                              │                              │
  ╱Circuit board data╱ ────►  │     • check                  │
 ╱ component side   ╱         │                              │
                              │     • edit                   │
  ╱Circuit board data╱ ────►  │                              │
 ╱ solder side      ╱         │     • prepare                │
                              │                              │
   ╱Drill data╱ ──────────►   │       Production data for    │
                              │       prototyping            │
                              │       with LPKF BoardMaster  │
                              └──────────────────────────────┘
                                           │
                                           ▼
                                    ┌──────────────┐
                                    │ LPKF         │
                                    │ BoardMaster  │
                                    └──────────────┘
```

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 are 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.)

Tips & tricks:	New documents are always named "Untitled" followed by a number. You may save these documents as a file by selecting SAVE AS from the FILE menu.

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

How to create a new CircuitCAM document:

1. Start the *CircuitCAM* program.

   The document "Untitled1" will already be opened.

2. Select SAVE AS from the FILE menu.

3. Enter the filename *Tutor1* in the FILENAME text box of the SAVE AS dialogue box.

4. Click on "**SAVE**" to save the document as a file under the name "Tutor1.cam".

|	
Tips & tricks:	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.

Customizing the 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:
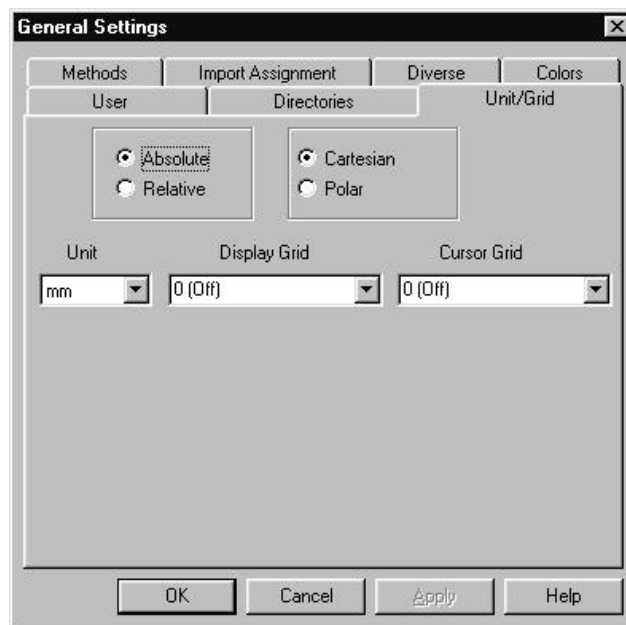
The additional functions in the GRID+UNIT toolbar are:

  Toggle the coordinates display in the status bar between ABSOLUTE COORDINATES and RELATIVE COORDINATES.

  Toggle the display format in the status bar between CARTESIAN COORDINATES and POLAR COORDINATES.

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

mm ▼   DISPLAY UNIT
All values will be displayed in the unit selected here.

1/20 ▼   DISPLAY GRID
Shows the dots in the grid selected here in the background.

1/10 ▼   CURSOR GRID
Allows placement in the grid selected here, also known as snap to grid.

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.

In the first exercise this 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

Aperture list

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

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

How to import the aperture list:

1. Select IMPORT... from the FILE menu or

   click on [icon] in the FRON TO END toolbar..

   The OPEN dialogue box is displayed:
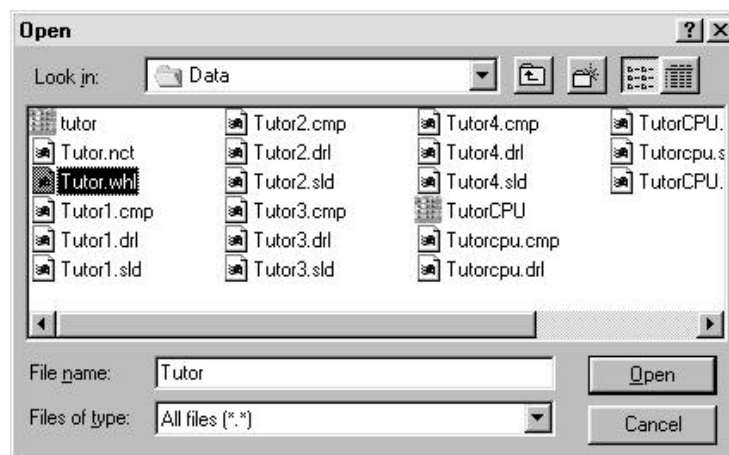
2. Select the file **TUTOR.WHL** and click on "**OPEN**" to import the aperture list.

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). The first time the two fields Apertures/Tool template and Aperture list are empty. Make your entries here.
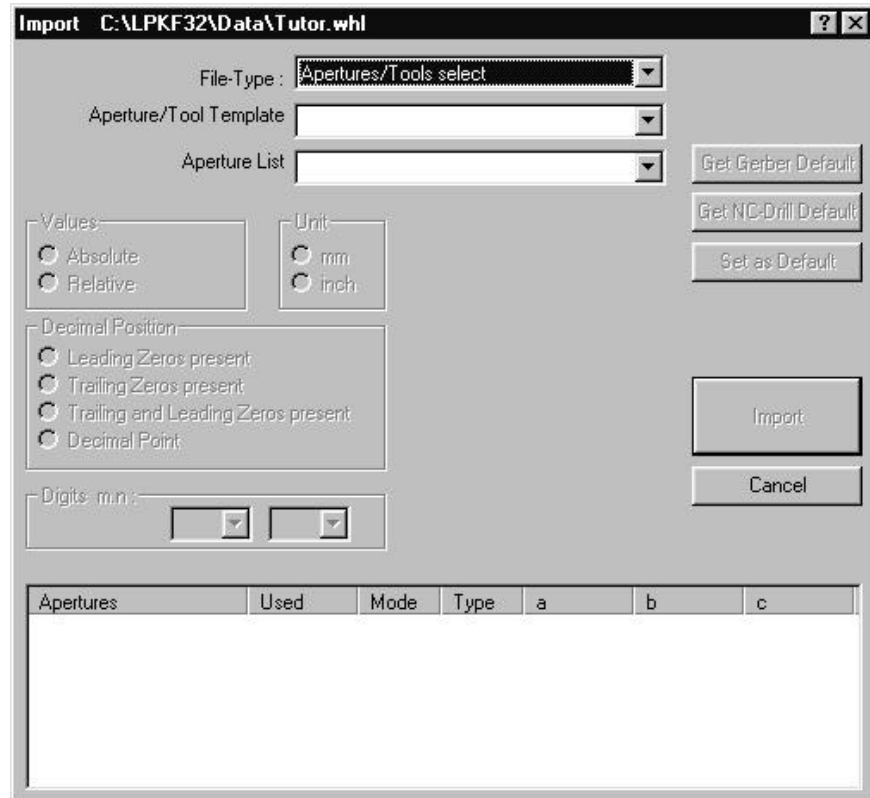
File of type:
Here you must specify the type of the file being imported so that *CircuitCAM* can interpret the data correctly.

APERTURE LIST / TOOL LIST:
Here you should enter a name for the aperture list which will be used within *CircuitCAM*. *CircuitCAM* already provides a number of default names for you to select from; these are also used by the various jobs within *CircuitCAM*.

The lower right list box (shown as APERTURE/TOOL TEMPLATE in the illustration) depends on the file type selected and has two variants:

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).

LAYER:
Appears if the file type GERBER, EXCELLON,... is selected. Here you should specify the layer to which the production data is to be assigned.

3. Select the **TUTORAPE.TXT** translation file from the APERTURE/TOOL TEMPLATE list box:

Aperture/Tool Template | TutorApe.TXT (Gerber)
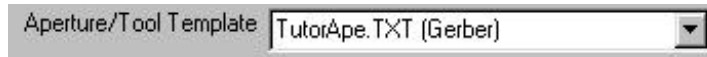
4. Select the **APETUTOR** list file from the APERTURE/LIST box and enter **APETUTOR1**:

Aperture List | ApeTutor1

5. All the settings necessary for importing the aperture list are now completed. If the translation file has been correctly selected, ie 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.

Apertures	Used	Mode	Type	a	b	c
D10	✔	⊥*	◯	0.2 mm		
D11	✔	⊥*	◯	1 mm		
D12	✔	⊥*	◯	0.3 mm		
D13	✔	⊥*	◯	1.1 mm		
D14	✔	⊥*	◯	1.5 mm		
D15	✔	⊥*	◯	1.3 mm		

6. 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.

7. Click on "**IMPORT**" to import the aperture list.

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

8. 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:

Configuration Name	Unit	Mode	Zeroes	m	n
ApeTutor	inch	abs	trailing	2	3
ApeTutor1	inch	abs	trailing	2	3

9. Double-click on the **APETUTOR1** entry to display the definitions for this aperture list:

ConfigItem Name	Mode	Type	a	b	
D10	⊥*	◯	0.2 mm		
D11	⊥*	◯	1 mm		
D12	⊥*	◯	0.3 mm		
D13	⊥*	◯	1.1 mm		
D14	⊥*	◯	1.5 mm		
D15	⊥*	◯	1.3 mm		
D16	⊥*	◯	1.6 mm		
D17	⊥*	◯	0.05 mm		
D18	**	◯	2.5 mm	0.6 mm	
D19	**	▢	2.5 mm	0.6 mm	
D20	⊥*	▢	6.35 mm		
D21	⊥*	▢	6.35 mm		

> **Tips & tricks:** 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 or in the Reference Manual.

10. Click on "**CLOSE**" to close the list window.

The aperture list has now been imported into *CircuitCAM*.

> **Tips & tricks:** 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.

Tool list

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

The steps for importing the tool list are similar.

How to import the tool list:

1. Select IMPORT... from the FILE menu... or

click on in the Front To End toolbar.

The OPEN dialogue box is displayed.

2. 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.

3. Select the **TUTORTOOLS.txt** translation file from the APERTURE/TOOL TRANSLATION list box:

4. Enter **TOOLTUTOR1** as the name for the tool list in the APERTURE/TOOL LIST list box.
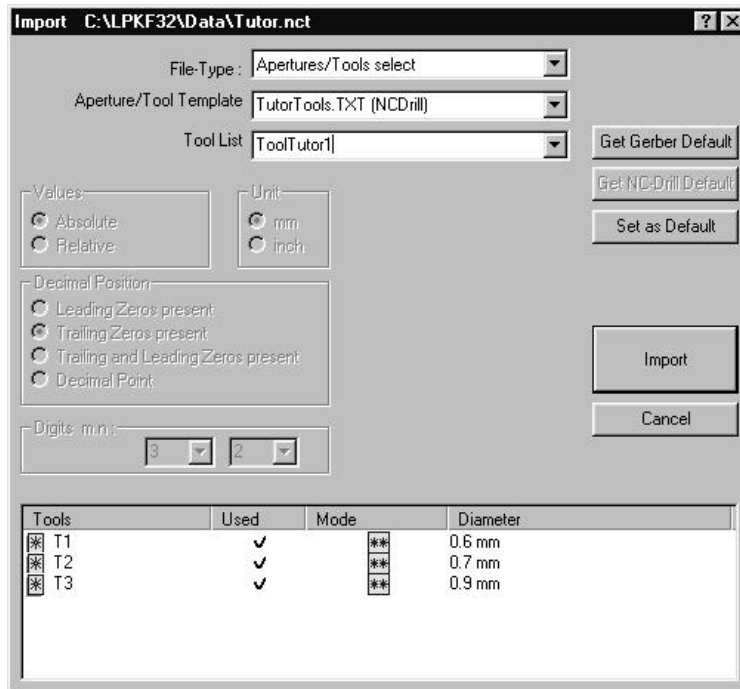
5. All the settings necessary for importing the tool list are now completed. If the translation file has been correctly selected, ie it matches the tool list to be imported 100%, the result will be displayed in the list box in the lower part of the Import dialogue box.

6. 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.

7. Click on "**IMPORT**" to import the tool list.

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

8. 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:
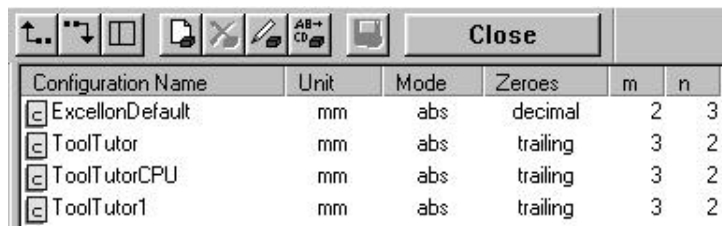
9. Double-click on the **TOOLTUTOR1** entry to display the definitions for this tool list:
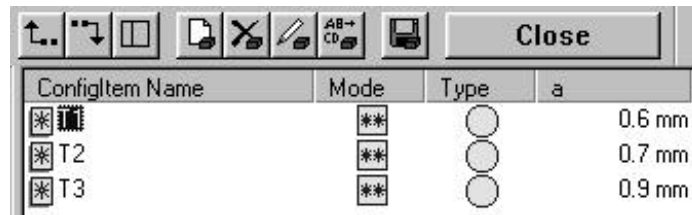
10. Click on "**CLOSE**" to close the function.

The tool list has now been imported into *CircuitCAM*.

> **Tips & tricks:** 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.

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:

1. Select IMPORT... from the FILE menu...

or click on [icon] in the Front To End toolbar.

The OPEN dialogue box is displayed.

> **Tips & tricks:** 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.

2. Hold down the **<Ctrl>** key and select the files **TUTOR1.SLD** and **TUTOR1.CMP**.

3. 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)

> **Tips & tricks:** Select the GERBER radio button if *CircuitCAM* does not automatically interpret the file to be imported as a Gerber file.

4. Assign the "ApeTutor1" aperture list previously defined to the Gerber data in the APERTURE/TOOL LIST list field by dropping down the list box with a click and selecting the **APETUTOR1** line:

5. Select the name **TOPLAYER** from the LAYER list box to assign the Gerber data for the component side to this layer.
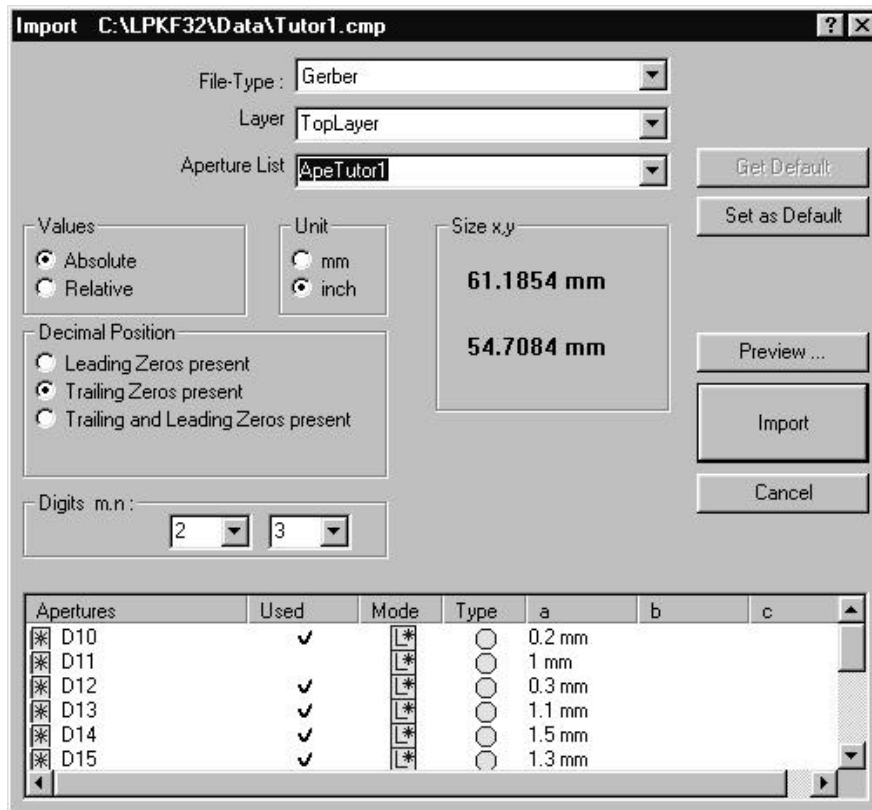
6. Clicking on the "**SETDEFAULT**" button causes the current settings for the layer to be saved. Clicking on the "**DEFAULT**" button causes the preselected default setting with the aperture list name GerberDefault to be used. Note that only one default setting can be used, the layer name must be reassigned for a second Gerber file.

7. 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):

> **Tips & tricks:** Select the GERBER radio button if *CircuitCAM* does not
> automatically interpret the file to be imported as a Gerber file.

8. Assign the "ApeTutor1" aperture list previously defined to the Gerber data in the
APERTURE/TOOL LIST list field by dropping down the list box with a click and
selecting the **APETUTOR1** line:

9. Select the name **BOTTOMLAYER** from the LAYER list box to assign the Gerber data
for the solder side to this layer.

10. 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.
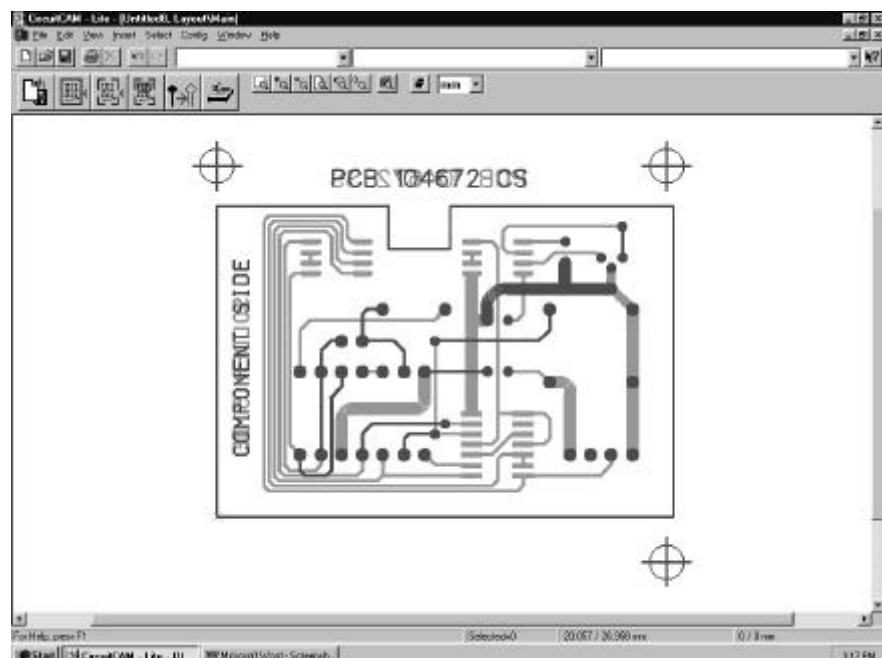
> **Tips & tricks:** We recommend that you always assign the names "BottomLayer"
> and "TopLayer" to the layers for the solder and component sides,
> as *CircuitCAM* uses these layer names for the preset isolation job.
>
> You can preview the data before it is imported by clicking on the
> "**PREVIEW**..." 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.

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:

1. Select IMPORT... from the FILE MENU, OR

   CLICK ON ▢ in the Front-To-End toolbar.

   The OPEN dialogue box is displayed:

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

2. Select the name Tutor1.DRL and click on "**OPEN**" to import the Excellon-file.

Assignments for the Excellon file Tutor1.drl:

> **Tips & tricks:** Select EXCELLON from the file type list box if *CircuitCAM* does not automatically interpret the file to be imported as an Excellon file.

3. Assign the "ToolTutor1 tool list previously imported to the Excellon data in the APERTURE/TOOL LIST list field by dropping down the list box with a click and selecting the **TOOLTUTOR1** line:

   Tool List | ToolTutor1 | ▼ |

4. Select the name **DRILLPLATED** from the LAYER list box to assign the Excellon data to this layer.
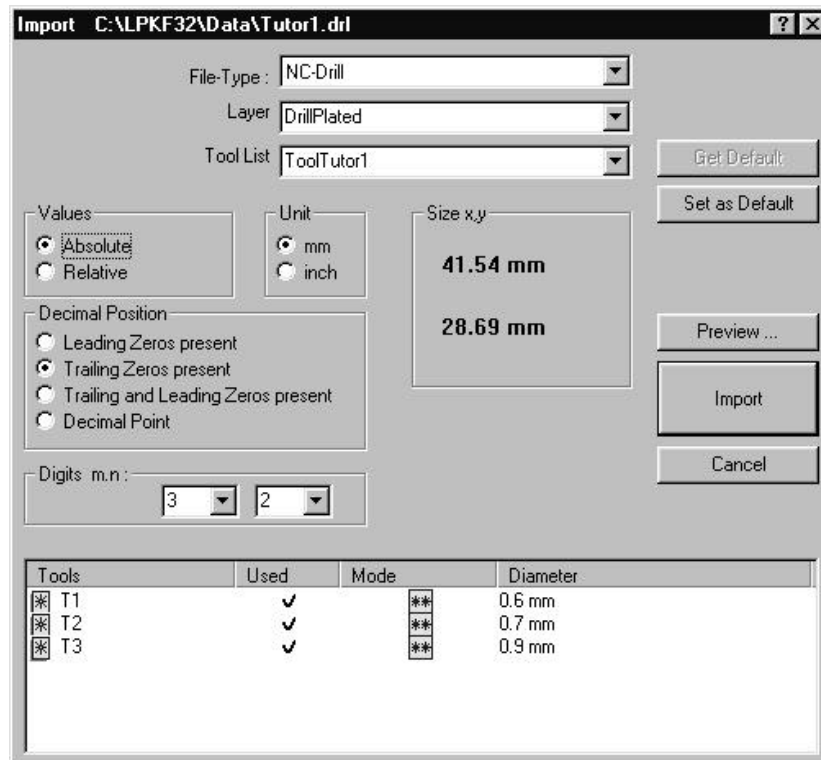
5. Click on "**IMPORT**" to import the file.

The Excellon data have now been imported into *CircuitCAM* and will be displayed in the graphic window.
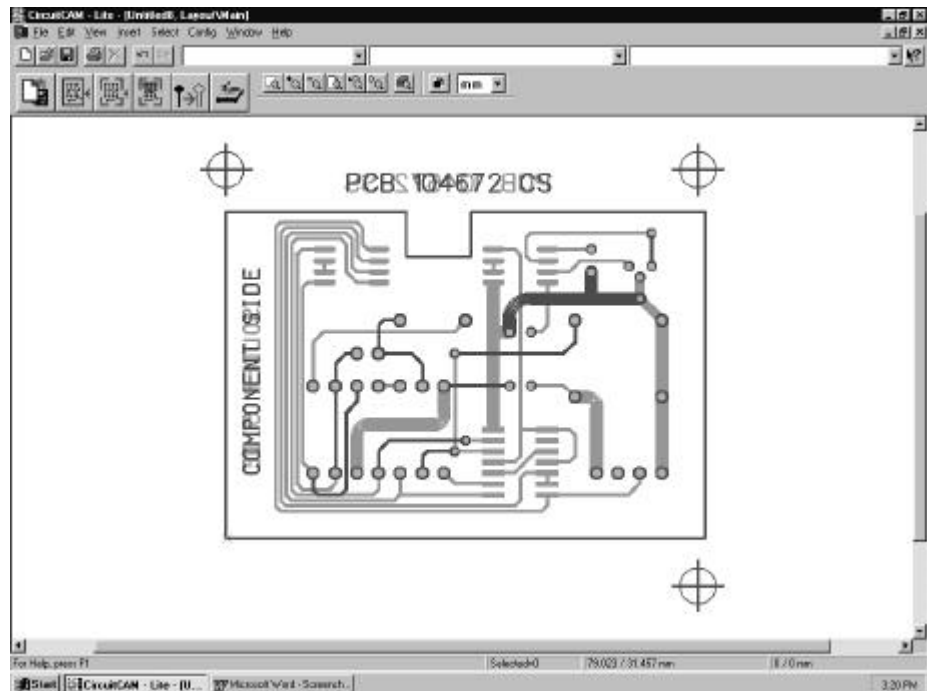
Tips & tricks: You may import data into the existing data in *CircuitCAM* at any time. All production data (Gerber and Excellon) can be imported in one step.

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.

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.

1. Select TUTOR.CAT from the FILE-NEW menu. Import assignment for the Tutor files has already been defined in this template file.

2. Open the General preferences dialogue box from CONFIGURATION-GENERAL... and click on the Import Assignments tab.
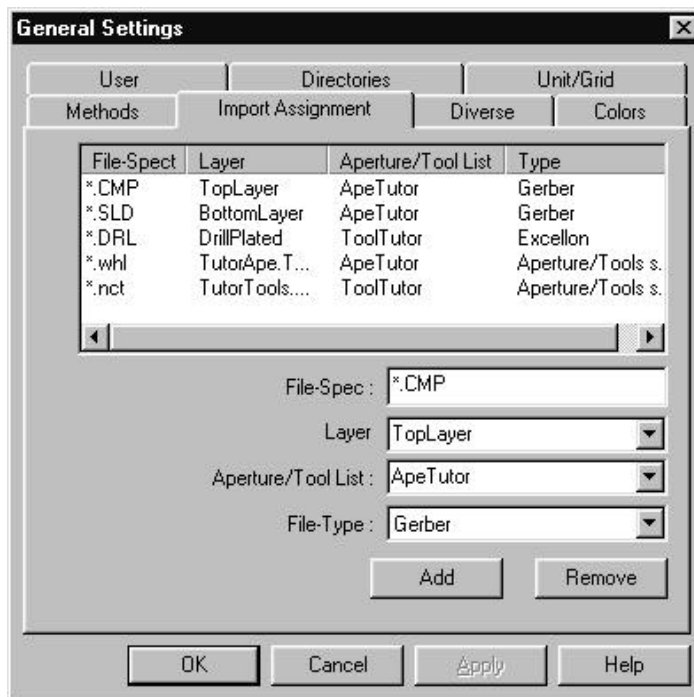
All the settings required for importing the Tutor files quickly have been predefined here. Set your Import assignments in the same way for the data you wish to import and save it to your own template file. (See Exercise 6, Own template files).

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.

Exercise 2

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.

> **Tips & tricks:** 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.

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 TO A 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
Click 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.
Keyboard: **<Home>**

Use the **<Cursor keys>** on the keyboard and the scrollbars to move the selection area.

Press **<End>** to centre the selection area on the mouse position.

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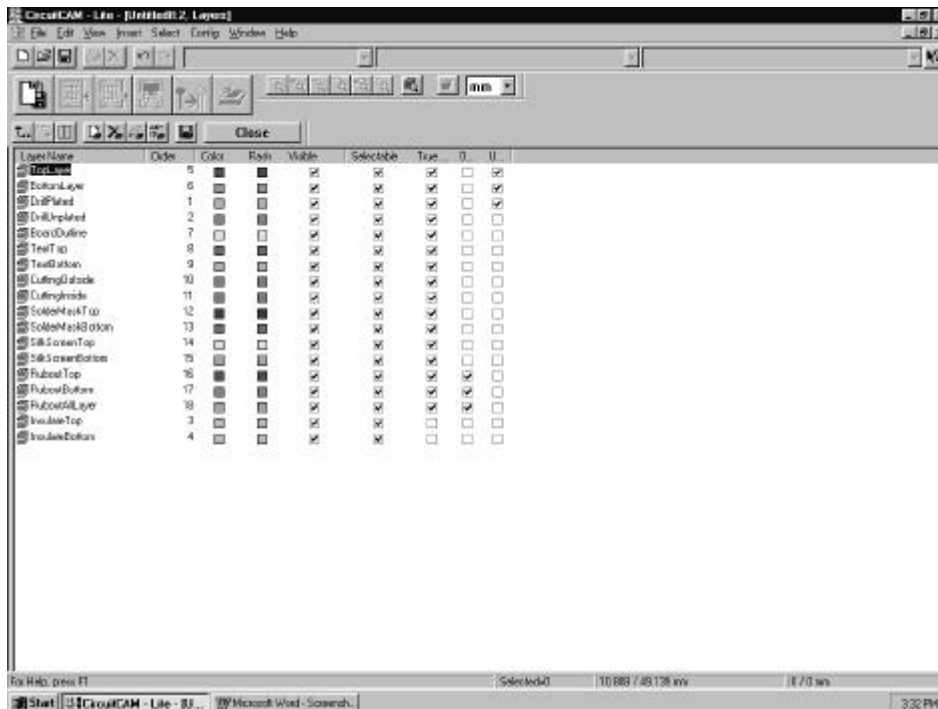
The toolbar at the top of the list window also gives you access to these other functions:

Create NEW LAYER

DELETE SELECTED LAYER

EDIT SELECTED LAYER

REDESIGN 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 .
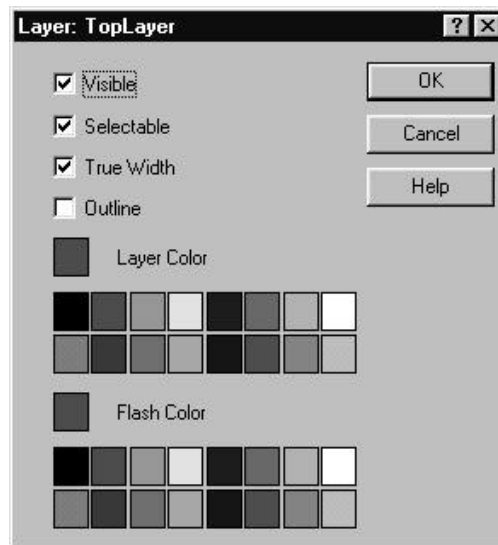
> **Tips & tricks:** You can also call the LAYER dialogue box by clicking on the line of the appropriate layer with the right mouse button.

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

	OUTLINE ILLUSTRATION ☐	OUTLINE ILLUSTRATION ☑
TRUE WIDTH ☑		
TRUE WIDTH ☐		

The graphical objects in the individual layers are displayed in the graphic window as superimposed levels. The SEQUENCE column in the LAYER list window indicates to you the level down from the top layer of the layer in the list.
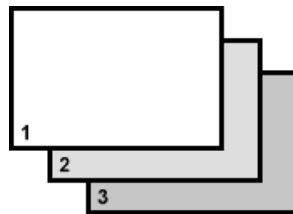
Remember that graphical objects in a lower level can be concealed by graphical objects in a higher level.

Click on the number of the layer which is to become the highest level in the SEQUENCE column of the LAYER list window. The number 1 will be assigned to this layer in the SEQUENCE column and the number of all other layers will be incremented by one, so that these layers are now one level deeper.

Tips & tricks:	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 or in the Reference Manual.

How to make a visual check on the imported data:

1. Click on 🔍 in the VIEW toolbar or press **<Home>** to display all graphical objects within the workspace.

2. Click on the lower, left corner of the circuit board frame to define an active Layer.

3. Click on the right mouse button and click now on "measuring" in the just opened menu.

4. Select a point in the graphic, from which the measurement can be accomplish.

5. Push the Mouse pointer to the endpoint of your measurement.

6. 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.

Tips & tricks:	You may choose between millimetres, microns, inches and mils (1/1000 inch) as the units in *CircuitCAM*; the default is millimetre. Change the unit using the UNIT list box on the GRID+UNIT toolbar.

7. Click <ESC> to finish the measurement.

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

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.

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 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 list box or by selecting a graphical object.

ALL
Click on this button to select all layers.

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

LINE
Lines may be selected if this button is enabled.

POLYGON
Polygons may be selected if this button is enabled.

CIRCLE
Circles may be selected if this button is enabled.

RECTANGLE
Rectangles may be selected if this button is enabled.

PAD/DRILLING
Pads (flashes) and drillings may be marked if this button is enabled.

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

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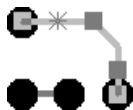
The handle closest to the cursor position when you clicked is displayed as a cross ✳ and serves as the anchor point. The selected object can therefore also have the following representations:

The anchor point coordinates are displayed in the STATUS BAR.

> **Tips & tricks:**   Use the **<+>** or **<->** keys to move the anchor point to the next handle on the selected object.

Clicking on a handle and dragging with the left mouse button (the shape of the cursor changes) allows you to move it to a different position, thus changing the shape and size of a graphical object:

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

> **Tips & tricks:**   Select PROPERTIES from the CONTEXT MENU (right mouse button) to view or modify the properties of a selected object.
>
> 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:

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

2. Click on in the SELECT toolbar to enable the NEW selection mode.

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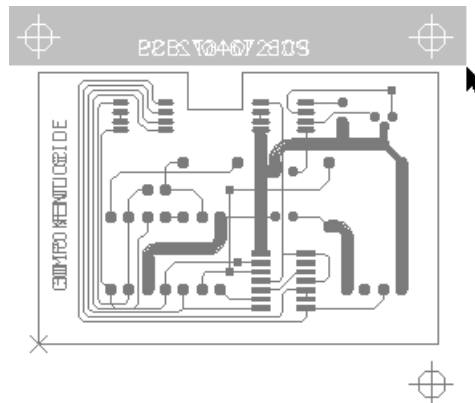
4. Click on in the SELECT toolbar to enable the PLUS selection mode.

5. Drag a box around the two lower registration marks using the left mouse button to select them as well:
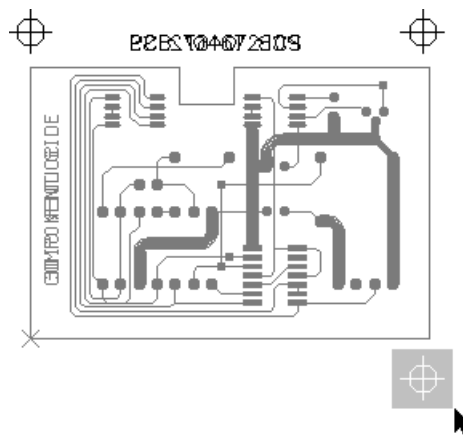
    All the objects outside the circuit board are now selected.

6. 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.

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:

Tips & tricks: 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.

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

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.

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

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

2. Click on "**CLOSE**" to close the LAYER list window.

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

> **Tips & tricks:** You must always click on a handle of an object, ie 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:
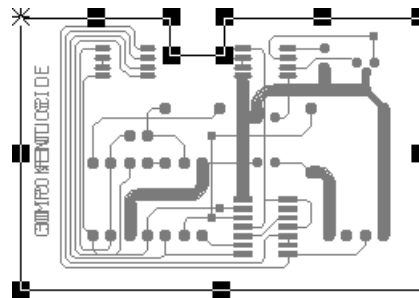
How to move selected objects to a different layer:

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

Press **<ESC>** to deselect all graphic objects.

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

1. Set the SELECTABLE property for the **BOTTOMLAYER** layer in the LAYER list window.
2. Set the NOT SELECTABLE property for the **BOARDOUTLINE** layer in the LAYER list window.
3. Select the circuit board outline on the "BottomLayer" layer in the graphic window.
4. Press **<Delete>** to delete the selected circuit board outline.
5. Press **<Home>** to redraw the picture.

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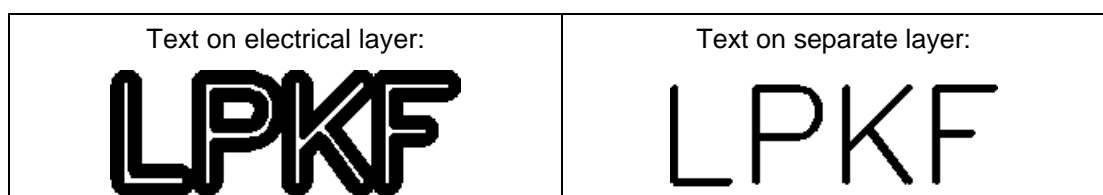
Since the procedure for moving text objects onto a separate layer is comparable with moving the circuit board outline to the "BoardOutline" layer, we will only describe the various operations in general terms here.

How to place text objects on a separate layer:

1. Select the text objects on the component side.

2. Select **TEXTTOP** from the LAYER list box on the DEFAULT toolbar to move the selected text objects to this layer:

3. Select the text objects on the solder side.

4. Select **TEXTBOTTOM** from the LAYER list box on the DEFAULT toolbar to move the selected text objects to this layer.

5. Press **<ESC>** to unselect all graphic objects

Tips & tricks: You should also move the fastening holes from an electrical layer to a mechanical layer, eg "DrillUnplated" just like the circuit board outline and the text objects.

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.

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.

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:

Component side ("InsulateTop")   Solder side ("InsulateBottom")
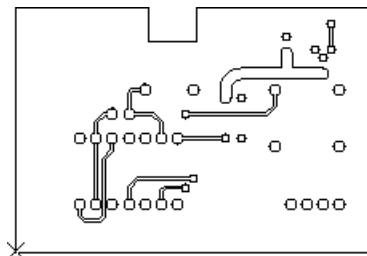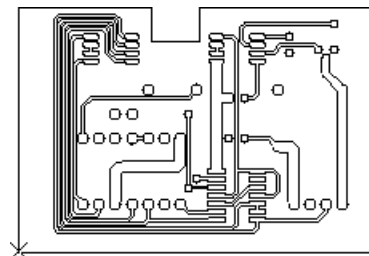
Tips & tricks:	Save your work to protect against data loss. Do this by selecting SAVE from the FILE menu. The document will be saved under the filename "Tutor2" already entered.

Export production data

Once the isolation channels have been calculated, the prepared production data can be ex-ported 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 submenu of the FILE menu

or click on [icon] in the FRONT TO END toolbar.

The file created will be saved to the working subfolder ("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.

Tips & tricks:	You can also export the production data in the most common CAD formats such as Gerber, Excellon and HP-GL (only PLUS version) in addition to the LpkfMillDrill format.
	Do this by selecting the appropriate line in the EXPORT submenu of the FILE menu.

Export ►	**LPKF CircuitBoardPlotter**
	LPKF Stencil Laser
	LPKF ►
	Gerber ►
	Excellon ►
	HP-GL ►
	Diverse ►
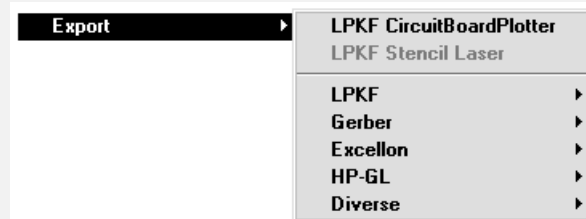
To conclude the second exercise, you can now close the "Tutor2" document by selecting CLOSE from the FILE menu.

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.

Exercise 3

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

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:

1. Select "**OPEN**" from the FILE menu.

2. Select the **TUTOR1** file and click on "**OPEN**".

> **Tips & tricks:** 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.

> **Tips & tricks:** 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:

1. Select the circuit board outline in the "TopLayer" layer.
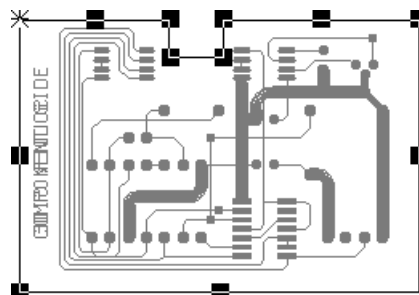
2. Click on [icon] in the FRONT TO END toolbar to open the CONTOUR ROUTING dialogue box:
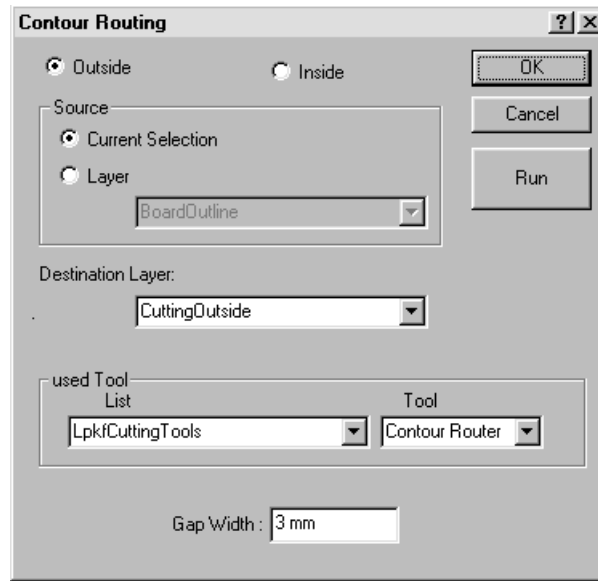
Tips & tricks: If the circuit board outline for the circuit board is already placed on a separate layer, eg on the "BoardOutline" layer, you may use this layer as the source for calculating the milling contour by clicking on the LAYER radio button and selecting **BOARDOUTLINE** from the LAYER list box.

---

3. Click on "**RUN**" to generate the milling contour.

Once calculation is complete, the milling contour will be displayed in the graphic window:
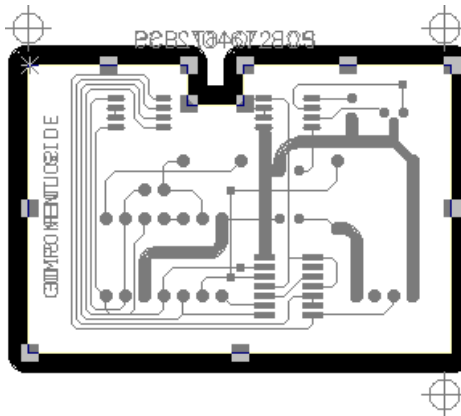
You can delete the circuit board outline from the "TopLayer" and "BottomLayer" since the milling contour has now been generated on the "CuttingOutside" layer:

4. Press **<Delete>** to delete the circuit board outline, which is still selected, from the "TopLayer" layer.

5. Select the circuit board outline in the "BottomLayer" layer.

6. 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.
	It may therefore be desirable to leave the circuit board connected to the board material by at least two webs to prevent personal injury and damage to the circuit board plotter.

How to insert the webs in the milling contour:

1. Select the contour generated on the "CuttingOutside" layer:
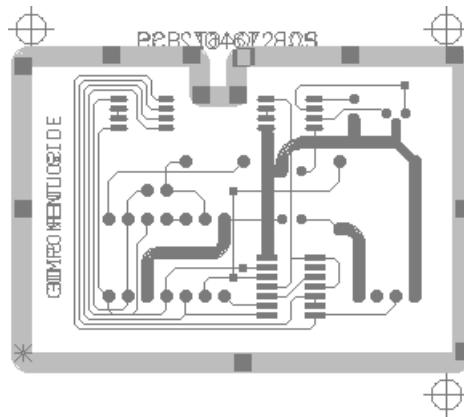
2. Use the **<+>** and **<->** keys to move the anchor point ✳ from handle to handle on the contour line:
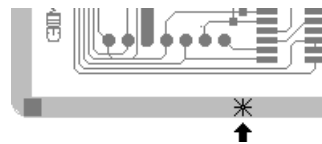
3. Press **<Ctrl>** + **<G>** or click on the icon in the FRONT TO END toolbar to insert a web at the anchor point position:

4. Repeat steps 2 and 3 to insert webs at subsequent positions:
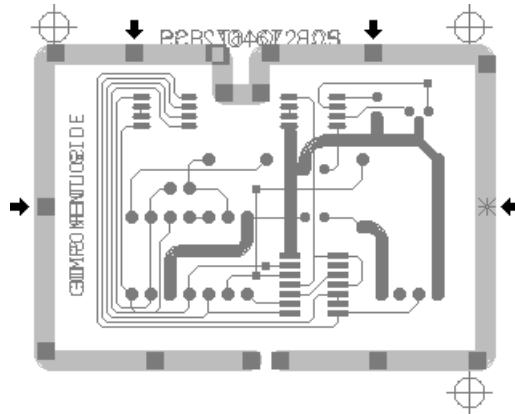
You have now generated a milling contour in which the circuit board is connected to the board material by five webs:
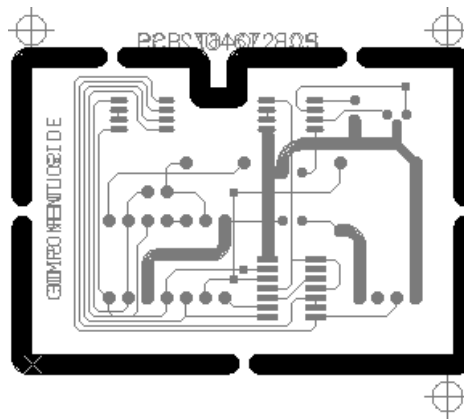
Tips & tricks: To save the current position of the exercise, select SAVE AS from the FILE menu.
Enter *Tutor3* as the filename and click on "**OK**".

Create rubout area

A rubout area is a zone in which not only the isolation channels around conductor tracks and pad 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. EMC) 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.
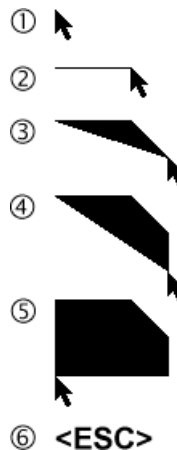
RECTANGLE
(default setting for Rubout area)
defined by two mouse clicks:
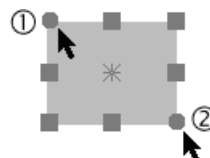
CIRCLE
defined from the centre point by two mouse clicks:
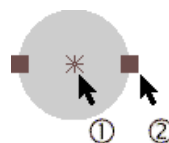
> **Tips & tricks:** Instead of clicking with the mouse, you can enter the cursor
> position numerically from the keyboard by selecting SET CURSOR...
> from the EDIT menu.
> The SET CURSOR dialogue box will be displayed and you can enter
> the new cursor position in the text boxes:
>
> | Set Cursor Point | ? X |
> | --- | --- |
> | x : 0 mm | OK |
> | y : 0 mm | Cancel |
>
> Click on "**OK**" 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.
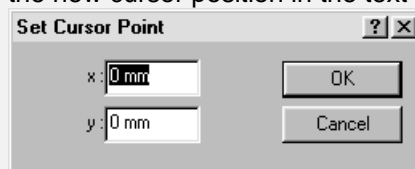
Three predefined layers are available for drawing Rubout areas, "RuboutBottom" for the bottom side of the circuit board, "RuboutTop" for the top face of the circuit board and Rubout all layers for both sides of the circuit board (standard).

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:

1. 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".

> **Tips & tricks:** 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.

2. Click above and to the left of the 14 pin SMD IC to set the starting point of the rectangle:
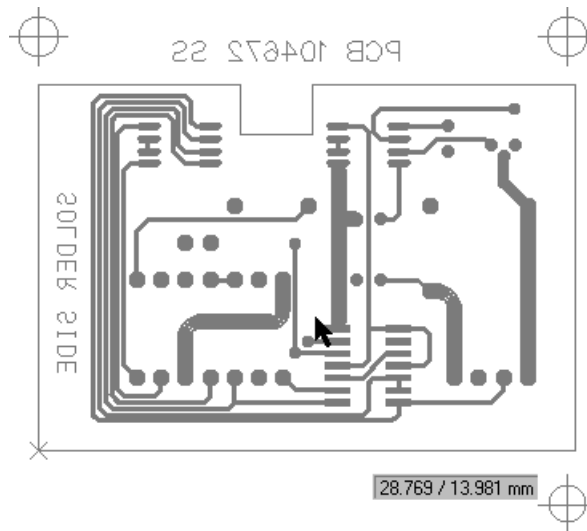
3. Move the cursor down and to the right to draw the rectangle around the SMD IC:
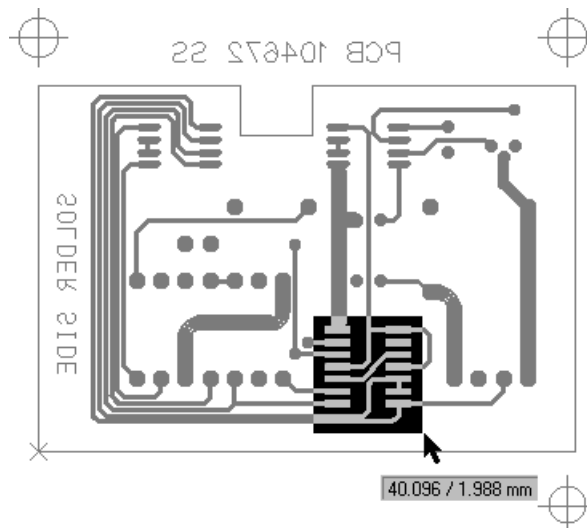
4. Click again to set the end point of the rectangle.

5. Click on ⬉ in the GRAPHICS toolbar 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 XX the corresponding layer from the List box "Layer" in the standard tool bar.

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

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

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.

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.

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.

At the end of the exercise 3 you can close the document "Tutor3" by clicking on "**CLOSE**" in the menu "**FILE**".

Exercise 4

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 "InsulateDefaultTop..." 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 ISOLATE... 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.

1. *CircuitCAM* - Lite

Settings on the Start tab
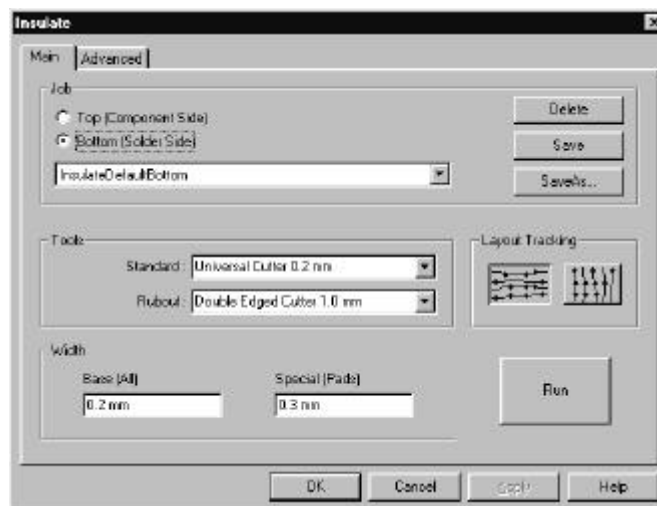
JOB:	Select the side of the circuit board for which the job is to be executed. The corresponding layer, "Top" or "Bottom", will then be used automatically.
TOOLS:	Here you should define the tools that will be used for milling the circuit board. The Lite version has two tools available for removing the copper areas most efficiently and quickly. Click on to drop down one of the list boxes from which you can select the tool required:
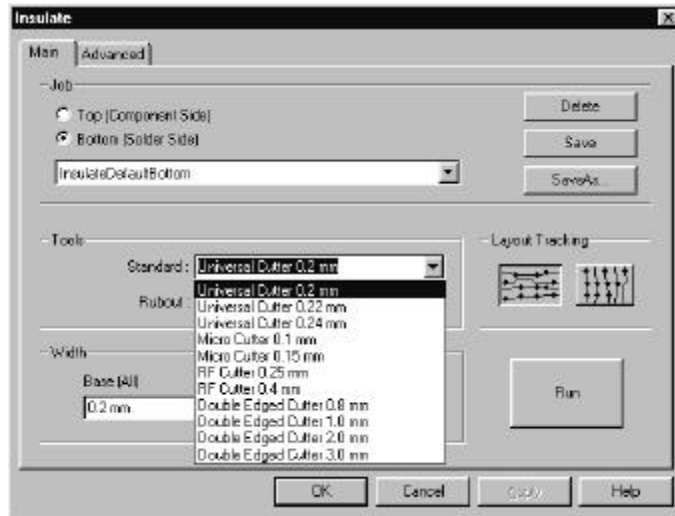
Isolation width:	Specify the width for the isolation around all electrical objects (conductor tracks and pads) in the DEFAULT text box. You can enter the width of the isolation around the pads that have been created by flash apertures in the SPECIAL text box.
PREFERRED DIRECTION:	Clicking on one of these 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

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.
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.
PRECISE 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.

Tips & tricks: 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 (By very slowly processors: 0,01mm)

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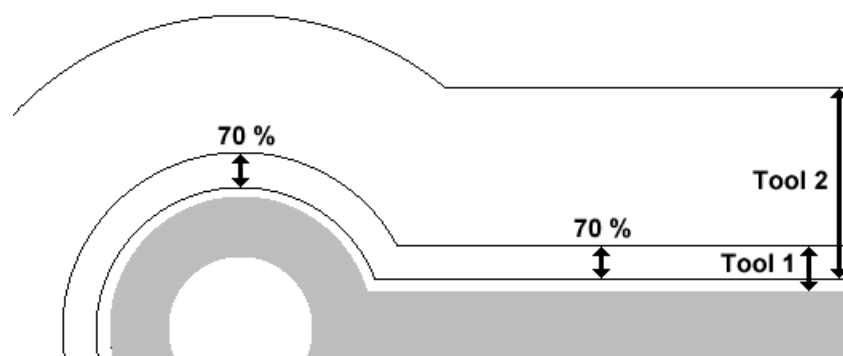
Tips & tricks: The next larger milling path should overlap the smaller milling path by between 50% and 90%.

2. *CircuitCAM* - PCB and PRO

All the features in *CircuitCAM* – Lite are also available in the PCB and PRO versions. These two versions have, however, additional features which will be described below.

Settings on the Start tab
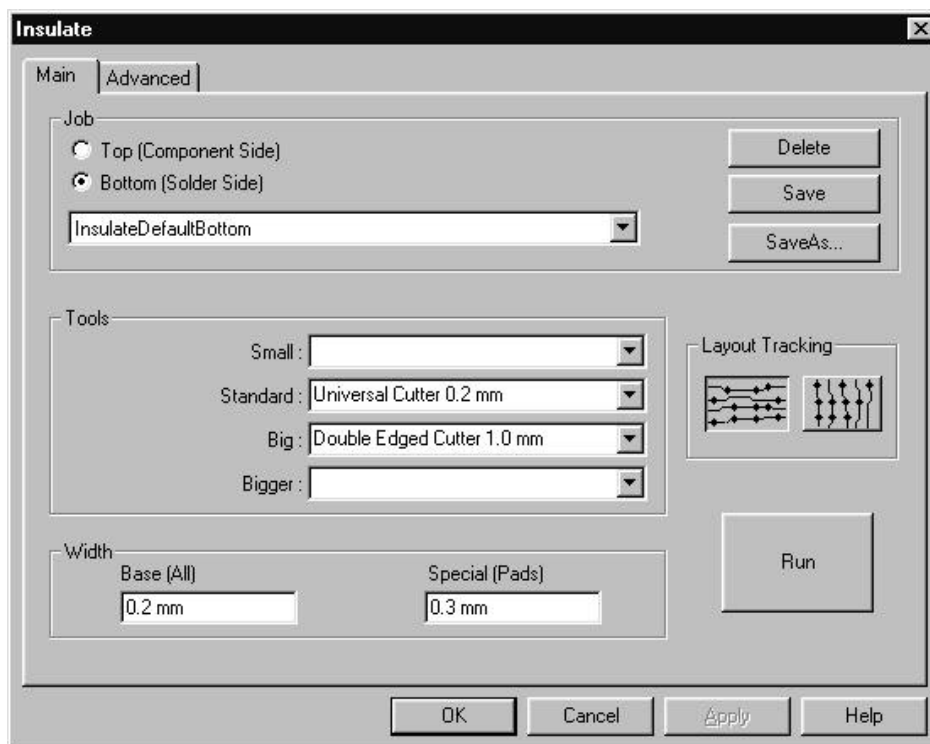
TOOLS: The PCB and PRO versions have two further tools in addition to the two tools in the Lite version:
The SMALL TOOL, WHICH IS USED IN AREAS OF THE CIRCUIT BOARD IN WHICH THE STANDARD TOOL CANNOT BE USED BECAUSE OF ITS DIAMETER.
THE BIG TOOL, WHICH IS USED AS A FURTHER RUBOUT TOOL TO REMOVE LARGE AREAS OF RESIDUAL COPPER.

Settings on the Advance tab
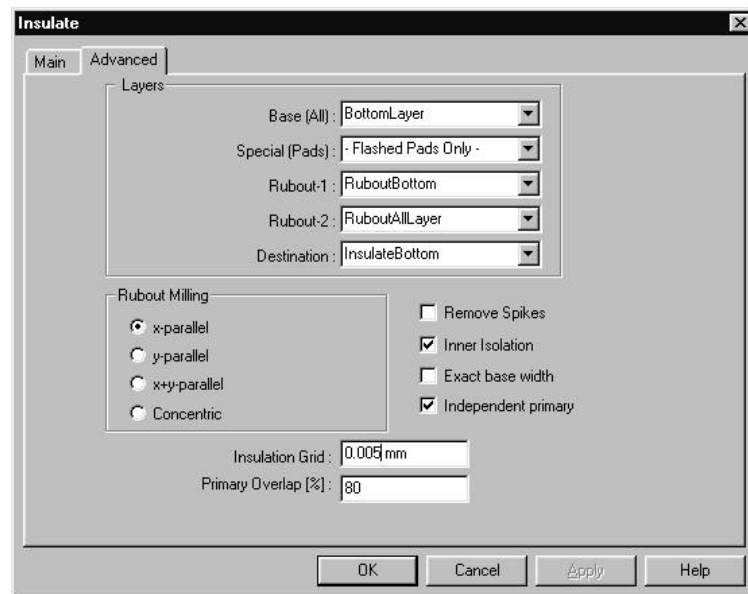
RUBOUT MILLING: Concentric or x and y parallel milling tracks can be gene-
rated here in addition to the x and y parallel rubout milling
tracks.

INNER ISOLATION TRACK: Unchecking this checkbox causes only the outer isolation
channel around a closed conductor track pattern to be ge-
nerated and not the internal channel.

INDEPENDENT PRIMARY CHANNEL:

Unchecking this checkbox prevents the generation of a
primary milling pass between two elements closer together
than the diameter of the tool (risk of short circuiting). A
milling pass will be generated in all cases if this checkbox
is activated (danger of a conductor track being made thin-
ner).

Tips & tricks: The "InsulateDefaultTop" and "InsulateDefaultBottom" jobs are
assigned by default to the buttons with the INSULATE ALL LAYERS
function in the FRONT-TO-END toolbar.
Customize these jobs to your requirements so that you have rapid
access to the isolation function desired via the toolbar.

How to customize the default job to your requirements:

1. Select the **INSULATEDEFAULTBOTTOM** job from the START tab in the ISOLATE
dialogue box.

Tips & tricks: It is a good idea to save the default job under a different name so
that it will still be available to you at a later date.
Click on to save the default job under a different name.
Enter the new name in the text box and click on .
Once you have saved the job under a new name, select the
INSULATEDEFAULTBOTTOM job again from the JOB tab.

2. Change the parameters on the START and ADVANCED tabs according to your requirements.

3. Click on in the START tab to save the "InsulateDefaultBottom" job with the new parameters.

4. Repeat these steps for the "InsulateDefaultTop" job.

5. The default jobs will be executed immediately with the settings you have modified once you click on in the FRONT-TO-END toolbar.

How to create a new job:

1. Select the job you wish to use as the basis for the job you are about to create from the START tab of the INSULATE dialogue box.

2. Change the parameters on the START and ADVANCED tabs according to your requirements.

3. Click on in the START tab to save the amended job under a new name.

4. Enter the new name in the text box and click on .

Tips & tricks:

You can delete a job by selecting it from the JOB tab and clicking on "**DELETE**" .

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.

Creating a *CircuitCAM* template file

CircuitCAM offers the option of creating a template file with user-specific basic settings comparable to the format templates in a word processor.

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.

CircuitCAM template files are also saved in the CAM format, but have the file extension "cat" and are saved to the "New_Templates" subfolder.

The available template files are listed in the NEW submenu of the FILE menu:
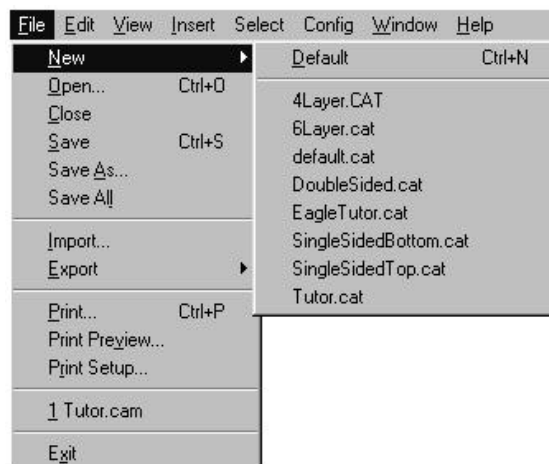
Various template files are saved to the templates folder when you install *CircuitCAM*:

Template	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.

Tips & tricks:	The most recently used documents are always listed in the FILE menu so that you can open them quickly.
	Click on TUTOR1.CAM 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:

1. Click on "all" in the SELECT menu to select all graphical objects.

2. Press **<Delete>** to delete the selected objects.

3. Select SAVE AS from the FILE menu.

4. Change to the templates folder "New_Templates" in the SAVE AS dialogue box.

   This folder is located in the "LPKF Prototyping" folder.

5. Select the **NEW TEMPLATE (*.CAT)** line from the FILE OF TYPE list box.

6. Enter *Tutor1* in the FILENAME text box.

7. Click on "**SAVE**" to save the "Tutor1.cat" template file.

8. 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.

Tips & tricks:	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.

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:

1. Select TUTOR.CAT from the NEW submenu of the FILE menu.

2. Import the following production data from the "Data" subfolder into the new document:

    - "Tutor2.cmp"

    - "Tutor2.sld"

    - "Tutor2.drl"

The following image appears in the graphic window once the production data are imported:
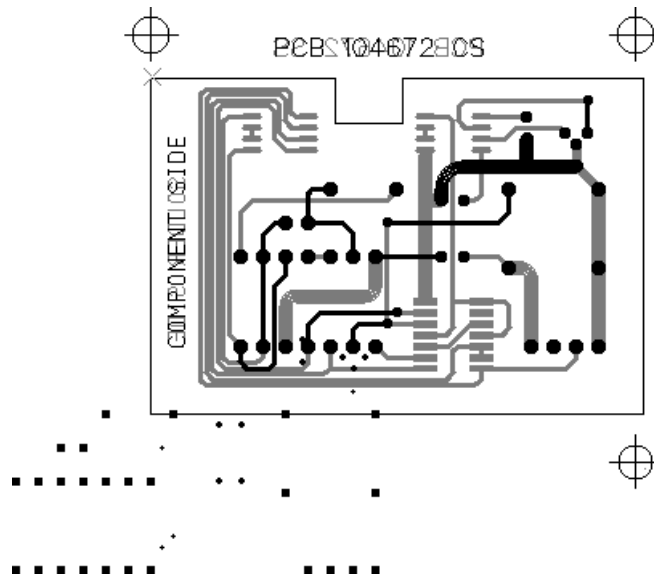
This exercise includes Excellon data which does not coincide with the Gerber data as a result of the specification of a different origin during output in the circuit board CAD program.

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

 MATCH UP LAYER

In this function, the active layer will be moved towards the origin using the coordinates of the anchor point for a selected object.

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:
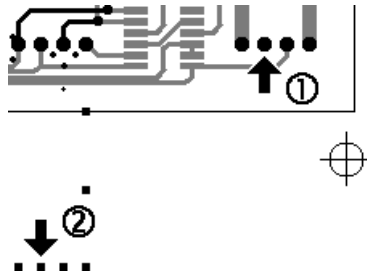
> **Tips & tricks:** A significant object can be a drill hole and it`s a solder pad, **as long as at is designed by a flash**. Both graphical objects have a center coordinate. So that selecting can be done very accurate. Don`t use graphical objects which are drawn and not flashed. Those graphical objects do have several markers very close and this makes it difficult to select a precise position. To make the selection easier you also can disable the line-select button.

1. Click on in the GRID+UNIT toolbar.

2. Select the lower drilling ② in the "DrillPlated" layer as your reference source.

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

4. Now you have to mark the upper pad ① of the layer "Top Layer" as target-point.

   The layer "Drill Plated" will be moved and the layers are registered.

> **Tips & tricks:** 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 register layers that are offset from one another. Since the further steps in preparation for Prototyping are identical with those in exercise 1, we will not list them here again.

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.

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.

Using import assignments

If you use production data with specified file extensions, e.g. "drl", "sld", "cmp", you can use import assignments in *CircuitCAM* to automate and thus simplify the import of production data. Here you determine which file extension represents what type of production data and assign a layer and 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.

How to specify import assignments:

1. Open the CONFIGURATION menu and select the GENERAL... line to open the GENERAL PREFERENCES dialogue box.

2. Click on the IMPORT ASSIGNMENTS tab.
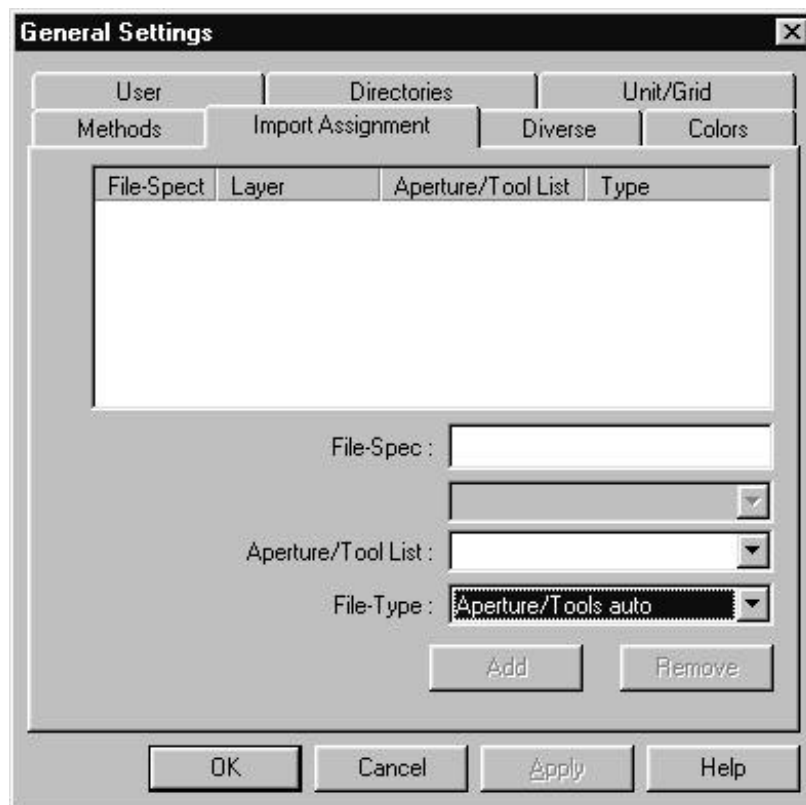
3. Enter **.drl* in the FILE SPECIFICATION text box to specify the import assignment for the Excellon production data (*.drl).

Tips & tricks:	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.

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

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

6. Select **DRILLPLATED** from the LAYER list box to assign the Excellon production data to this layer.

7. Click on "**ADD**" to accept the import assignment.

   The available import assignments are displayed in the top list box:
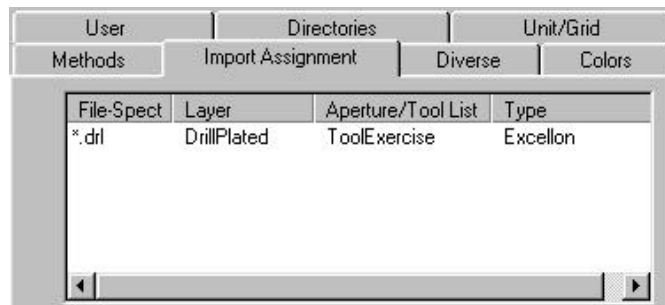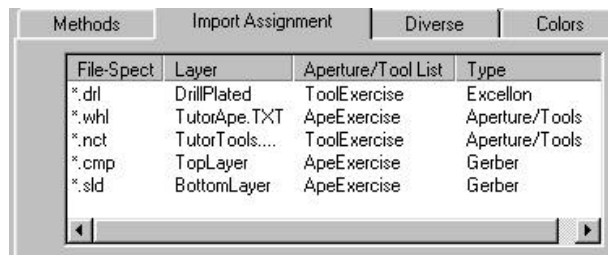
8. Repeat steps 3 to 7 to specify import assignments for the following production data:

	Aperture list	**Tool list**
FILE SPECIFICATION	*.whl	*.nct
APERTURE LIST / TOOL LIST	ApeExercise	ToolExercise
FILE OF 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 SPECIFICATION	*.drl	*.cmp	*.sld
APERTURE LIST / TOOL LIST	ToolExercise	ApeExercise	ApeExercise
FILE OF TYPE	Excellon	Gerber	Gerber
LAYER	DrillPlated	TopLayer	BottomLayer

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

Methods	Import Assignment		Diverse	Colors
File-Spect	Layer	Aperture/Tool List	Type	
*.drl	DrillPlated	ToolExercise	Excellon	
*.whl	TutorApe.TXT	ApeExercise	Aperture/Tools	
*.nct	TutorTools....	ToolExercise	Aperture/Tools	
*.cmp	TopLayer	ApeExercise	Gerber	
*.sld	BottomLayer	ApeExercise	Gerber	

9. Click on "**OK**" to close the GENERAL PREFERENCES dialogue box.

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 is indicated LAYER or the Listbox PATTERN in the Dialogbox IMPORT ASSIGNMENTS.

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:

1. Select TUTOR.CAT from the NEW submenu of the FILE menu.

2. Import the production data from the "Data" subfolder into the new document:

   • "Tutor3.cmp"

   • "Tutor3.sld"

   • "Tutor3.drl"

Tips & tricks: 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, so that you need only confirm the import for each file by clicking on "**OK**".

The following image appears in the graphic window once the production data are imported:
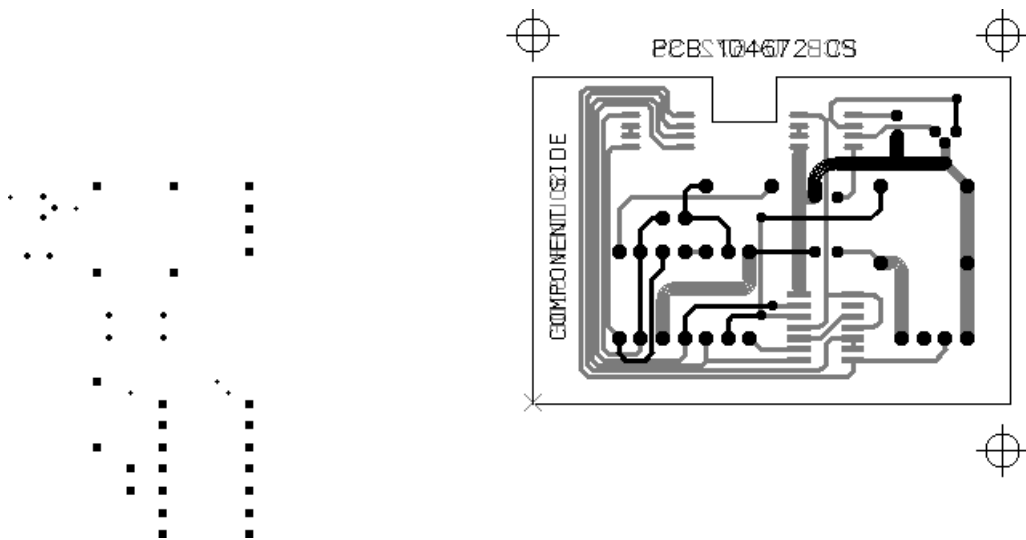
You will need the functions for mirroring or rotating graphical objects in addition to the SET ORIGIN and MOVE LAYER TOWARDS THE ORIGIN functions you have already used.

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

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

VALUE …	MIRROR X
90°	MIRROR X+90°
180°	MIRROR Y
270°	MIRROR Y+90°

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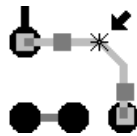
You may also mirror/rotate around the anchor point of an object which itself will not be mirrored/rotated:

1. Select the anchor point by clicking on the object.

The anchor point is indicated by a cross:
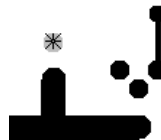
2. Select the objects you wish to mirror/rotate by dragging with the left mouse button **rather than** by clicking:

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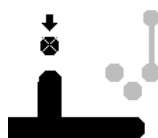
You can also position this anchor point completely independently of any object:

1. 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:
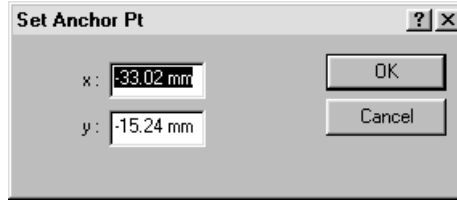
2. Enter the new coordinates for the anchor point in the text boxes and click on "**OK**".

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:

1. Click on the drill hole at the bottom right to select it.
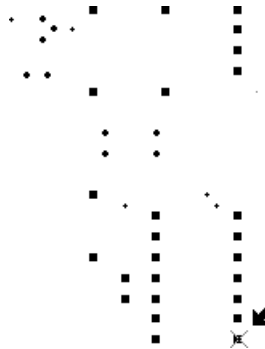
The position of this drill hole is now the reference point around which the layer will be rotated.

The active layer **DRILLPLATED** is shown in the LAYER list box of the DEFAULT toolbar:
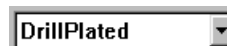
2. Click on LAYER in the SELECT toolbar to select all the objects in the active layer "DrillPlated".

3. 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:
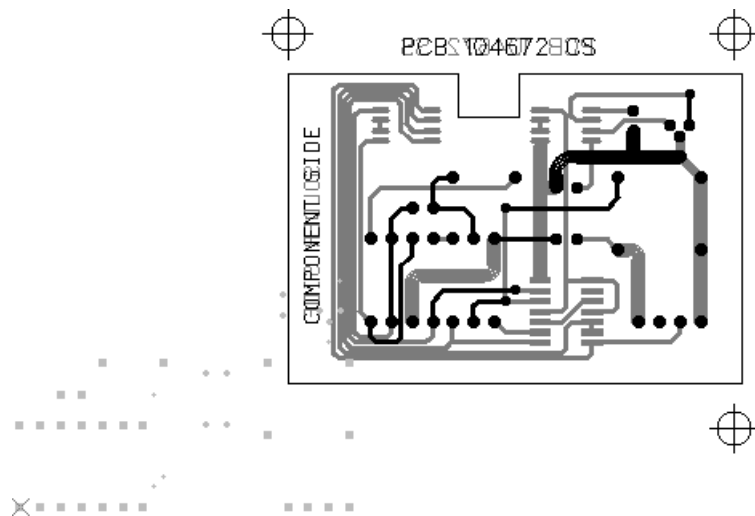
You can now register the layers against one another using the MATCH UP LAYER function, as shown in exercise 5.

Tips & tricks: 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.
You may used the tutorial files "Tutor4.cmp", "Tutor4.drl" and "Tutor4.sld" if you wish to practise mirroring layers.

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

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.

Appendix

Predefined layers

Layer name	Use	Template				
		default	Single sided	Double sided	4Layer	6Layer
TOPLAYER	Component side layout	✓		✓	✓	✓
BOTTOMLAYER	Solder side layout	✓	✓	✓	✓	✓
LAYER2	Layout Multilayer2				✓	✓
LAYER3	Layout Multilayer3				✓	✓
LAYER4	Layout Multilayer4					✓
LAYER5	Layout Multilayer5					✓
DRILLPLATED	Drillings (through-plated)	✓	✓	✓	✓	✓
DRILLUNPLATED	Drillings (not through-plated)	✓		✓	✓	✓
BOARDOUTLINE	Circuit board outline	✓	✓	✓	✓	✓
TEXTTOP	Text on component side	✓		✓	✓	✓
TEXTBOTTOM	Text on solder side	✓	✓	✓	✓	✓
CUTTINGOUTSIDE	Contour milling outside	✓	✓	✓	✓	✓
CUTTINGINSIDE	Contour milling inside	✓	✓	✓	✓	✓
SOLDERMASKTOP	Solder mask component side	✓		✓	✓	✓
SOLDERMASKBOTTOM	Solder mask solder side	✓	✓	✓	✓	✓
SILKSCREENTOP	Component print component side	✓	✓	✓	✓	✓
SILKSCREENBOTTOM	Component print solder side	✓		✓	✓	✓
RUBOUTTOP	Rubout area component side	✓		✓	✓	✓
RUBOUTBOTTOM	Rubout area solder side	✓	✓	✓	✓	✓
RUBOUTALLLAYER	Rubout area component side and solder side	✓		✓	✓	✓
INSULATETOP	Isolation channels component side	✓		✓	✓	✓
INSULATEBOTTOM	Isolation channels solder side	✓	✓	✓	✓	✓

Note: We recommend that you retain the names for these layers as *CircuitCAM* includes preconfigured jobs for these layers. (See next page.)

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 preferences (with the exception of the "User" and "Folders" 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 folder 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
EagleTutor.cat	Double-sided PCB With import assignment for Eagle data

The available files are listed in the NEW submenu when you select it from the FILE menu:
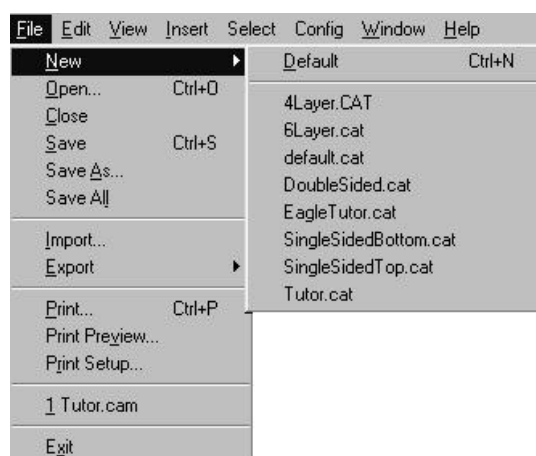
If, for example, you use the same aperture and tool list again and again, you can create your own template file which already contains both these lists. You will then not need to import these aperture and tool lists again for new circuit boards.

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.

Tips & tricks: You should save this template file under the name "default.cat" if you use the same template very frequently.

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.

Creating a 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**.

1. Start *CircuitCAM*.

   The template file called "default.cat" is the file that is started automatically when *CircuitCAM* is opened.
2. Now save this template file under a new name.
3. Click on SAVE AS in the FILE menu.
4. Now select the dialogue box opened as File of type NEW TEMPLATE (*.cat).
5. Then select the folder 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.

Saving an aperture list in the template file

Select Import from the File menu.

The OPEN FILE menu field will be displayed:
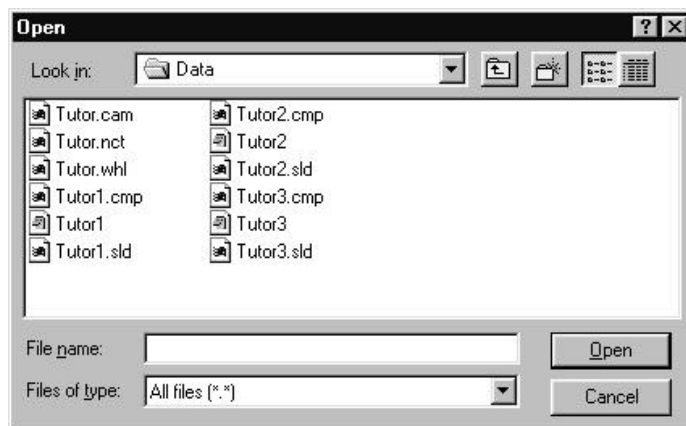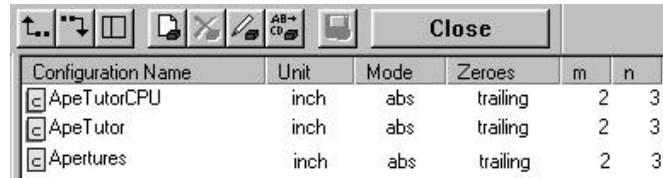
To import the aperture list, open the file **TUTOR.WHL** and enter a descriptive name, eg APERTURES to obtain the apertures/tool list. The file **TUTOR.WHL** is only an example in this case. You should carry out the same procedure with your own aperture list in future.

To import the aperture list **Tutor.whl**, make all the required settings in the Import dialogue box, as described in Exercise 1.

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.
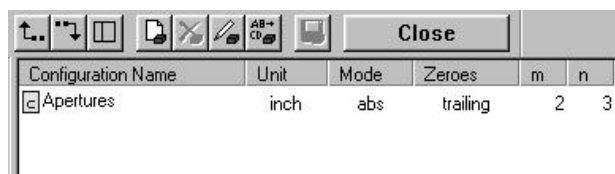
Let us assume that you only want to use your APERTURES list in future. This would mean that all other lists can be deleted.
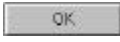
2. Left click to select APETUTORCPU and click on . This action will delete this aperture list.

3. Left click APETUTOR and click on . This action also deletes this aperture list.

4. 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.
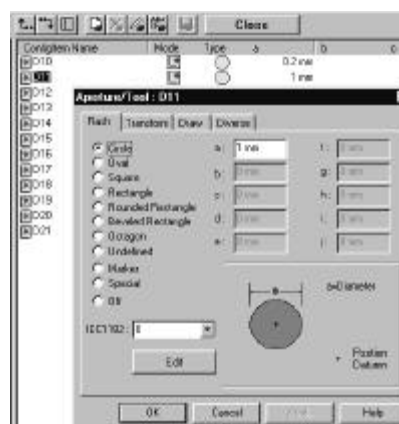
5. Clicking quits this dialogue box.

6. Click on

7. Now highlight the PERTURES list and click on (on the left, next to Close) to save this list to the own.cat template file.

Saving a modified isolation job

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

1. To define an isolation job, open the ISOLATE .... submenu from the EDIT menu.
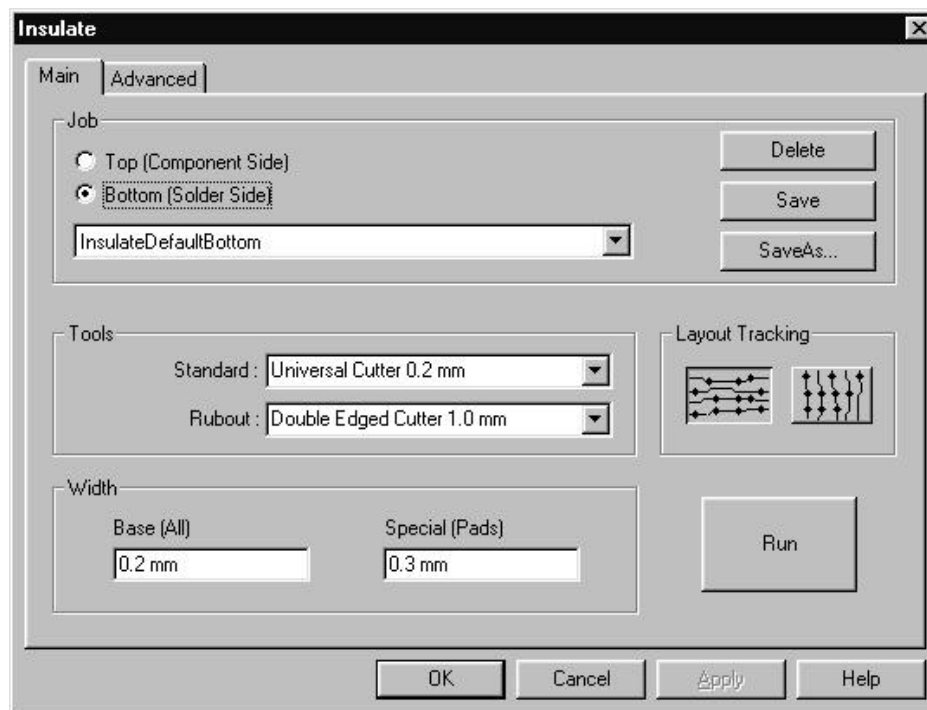
By way of an example, we will redefine the isolation width for Special and the preferred direction of the conductor tracks for a job on the solder side (bottom).

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

3. 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.
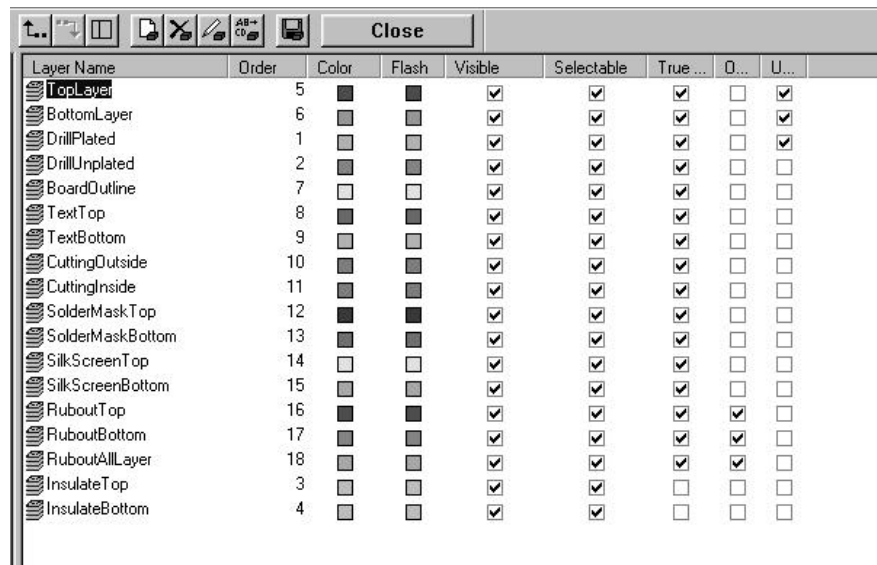
In both cases, this modification will always be available to you when you start a new project with this CAT file: FILE-NEW-OWN.cat .

4. Clicking "**OK**" quits this dialogue box.

Saving a new layer

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

1. Click on 🔍 to open the layer list.

In this list you may now modify the various parameters by clicking in the checkboxes, or even add a new layer.

2. 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.

3. Now highlight the EVELS Entry, and click on the 💾 button to save all the modifications in the layer list to your CAT file.

Saving the parameters in General preferences

The settings for the parameters under GENERAL PREFERENCES 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 PREFERENCES dialogue box by clicking on GENERAL... in the CONFIGURATION menu. Take a look at all the tabs and your settings.

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

The entries on the User and Folders 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* with 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.

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		
Default	0.2 mm	0.2 mm
Special	0.3 mm	0.3 mm
Tool		
Default	Universal Cutter 0.2 mm	Universal Cutter 0.2 mm
Large	Double Edge Cutter 1.0 mm	Double Edge Cutter 1.0 mm
Layer		
Basis	BottomLayer	TopLayer
Special	- Flashed Pads Only -	- Flashed Pads Only -
Rubout1	RuboutBottom	RuboutTop
Rubout2	RuboutAll	RuboutAll
Result	InsulateBottom	InsulateTop
Isolation grid	0.005 mm	0.005 mm
Rubout milling	x parallel	y parallel
Primary overlap	80	80 %

Output job: LPKFCircuitBoardPlotter

CircuitCAM has one prepared output job already:

"LPKF Circuit Board Plotter". Use this job to output the production data you have created in an LMD format thus making it available to LPKF *BoardMaster*.

This job is assigned to the ![button] button in the Front-To-End toolbar. Clicking on the button saves the production data to an LMD file. If *BoardMaster* is open, this LMD file is automatically imported into *BoardMaster* and placed on the screen in the middle of the work area.

Below we should like to show you which parameters are used to define an output task.

1. Select EXPORT SETTINGS... from the JOB CONFIGURATIONS submenu of the CONFIG menu to display the list window with all the available output jobs.

2. Double-click the entry for **LPKFCIRCUITBOARDPLOTTER** to display the various layers with the tools list and production phase assigned to each:

3. 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.

Tips & tricks: You will find a description of the various production phases in the LPKF *BoardMaster* Manual.
Click on 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 to modifications to a job.

A second job is provided for double-sided pcbs that are not to be through-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.

Customize translation file

A translation file is used to convert the aperture list (ASCII file) from any CAD system so that it can be imported into *CircuitCAM* without further modification. This significantly speeds up the work of users who continually need to work with different aperture lists, as there is no need for a manual definition of the various apertures. Furthermore, you can use the translation file to convert tool lists in Excellon or Sieb&Meyer format.

CircuitCAM already has a number of prepared translation files for the most common CAD systems. However, it may be the case under certain circumstances that you will need to customize a translation file individually to meet your needs.

Note:	The translation files may be found in the *CircuitCAM* "Ape_Templates" subfolder or in the folder you have specified in your *CircuitCAM* GENERAL PREFERENCES for the aperture and tool template files.

Structure of the translation file

Translation files are ASCII format files which you can edit in any text editor. In Windows, for instance, you have both the NOTEPAD and WORDPAD text editors.

A translation file is made up of three sections:

- Fixed parameters
- Aperture definitions
- Variable parameters

Tips & tricks:	You will need to define the fixed and variable parameters in the translation file if the format and aperture definition in the aperture list do not correspond to the default settings.

Section 1: fixed parameters

The following table shows a listing of the available fixed parameters with possible parameter values and the corresponding default setting.

Parameter name	Parameter value	Default setting for parameter value
APE_SCALE	<real number>	1
APE_UNITS	**INCH** **MIL** **MM**	MIL
COMMENT	<Character>	<Character pair> [<Character>	<Character pair>]	
CONFIGURATION	**ConfigExcellon** **ConfigGerber** **ConfigHpgl** **ConfigSiebMeyer**	ConfigGerber
COOR_MN	<integer>**.**<integer>	2.3
COOR_MODE	**ABSOLUTE** **INCREMENTAL**	ABSOLUTE
COOR_UNITS	**INCH** **MM**	INCH
COOR_ZEROS	**DECIMAL** **LEADING** **TRAILING**	TRAILING
KEY_STRING	<character string>	
REQUIRED_STRING	<character string>	
Legend: | Alternative (or) [...] 0 or multiple repetition		

APE_SCALE The unit can be scaled by this parameter if the unit for the aperture size matches none of the possible parameters under **APE_UNITS**.
Example: The aperture sizes are specified in 1/100 mm. Set **APE_UNITS** to **MM** and **APE_SCALE** to **0.01**.

APE_UNITS defines the unit for the aperture size within the aperture list. (**MIL, INCH** or **MM**)

COMMENT defines one or more characters which precede a comment line. Multiple characters/character pairs are separated by a space.
Example: **COMMENT @ # //** means that any line beginning with '@', '#' or '//' is a comment line.

CONFIGURATION defines the format for which this translation file has been defined. (Gerber, Excellon, HP-GL or Sieb&Meyer)

COOR_MN   specifies the m.n code of the Gerber format. The appropriate values are specified here if the Gerber data are always created with the same number of integer and decimal places.

COOR_MODE   specifies the coordinate mode of the Gerber format. **COOR_MODE ABSOLUTE** is selected for Gerber data with absolute values, **COOR_MODE INCREMENTAL** for relative values.

COOR_UNITS   specifies the unit for the Gerber data, imperial (**INCH**) or metric (**MM**).

COOR_ZEROS   specifies the preference for the zeroes in the Gerber format. Set this parameter to **LEADING** if the Gerber data are always created with leading zeroes, or to **TRAILING** if trailing zeroes are always used. Select either **TRAILING** or **LEADING** if both leading and trailing zeroes are exported.

KEY_STRING   (only for automatic detection)
                     Specifies the keyword that the aperture list must contain to ensure that this translation file is used to convert the aperture list. The scan process is ended immediately the keyword is found.

REQUIRED_STRING
                     (only for automatic detection)
                     Specifies the keyword that the aperture list must contain to narrow the search in automatic detection of the translation file. The scan process continues after finding the keyword to find the most suitable translation file. All translation files lacking this keyword will be ignored immediately in the scanning process. This can be used to restrict the range of translation files to be considered.

Section 2: Aperture definition

Every aperture in the aperture list must be capable of being assigned precisely by one of the entries in the aperture definition of the translation file. The apertures are distinguished by D code, mode, form and size. Each line in the aperture list will be checked for a match with the entries in the translation file. The aperture information will be converted if a match is found, otherwise the aperture concerned will be ignored.

The following syntax will be used for the aperture definitions:

1,<DRAW_FORMAT><FLASH_FORMAT>**:"**<APE_FORMAT>**"**

<DRAW_FORMAT> parameter	Meaning
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.

<FLASH_FORMAT> parameter	Meaning	Size 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 (diameter)
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)
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*.	

<APE_FORMAT>
parameter

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 the aperture list:

```
1  D12   Draw    Circle          8        8       Aperture No.12

2  D20   Flash   Rectangle      20       60       Aperture No.20
```

Associated definition in the translation file:

```
1,CN:"%s D%% Draw Circle %a"

1,NR:"%s D%% Flash Rectangle %a %b"
```

Example 2

Where the separator between the various parameters is not a space, the relevant character
is shown in square brackets [ ]:

Information in the aperture list:

```
12   Flash    Oval       X0.6Y1.20

20   Flash    Rectangle  X1.0Y0.7
```

Associated 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 are capable of writing more precise specifications of the *Gerber* format to the aperture list in addition to the aperture information. Variable parameters may be used for the evaluation of this information so that it is not necessary to modify the translation file for each change to this supplementary information.

The aperture list will be searched during the conversion process for the strings defined for the variable parameters and assigned to the appropriate parameter where a match is found.

Variable parameter	Meaning
Coordinates unit	
100 : "<text>"	The string entered for <text> defines **INCH** as the coordinates unit.
101 : "<text>"	The string entered for <text> defines **MM** as the coordinates unit.
Coordinate mode	
110 : "<text>"	The string entered for <text> defines **ABSOLUTE** as the coordinates mode.
111 : "<text>"	The string entered for <text> defines **RELATIVE** as the coordinates mode.
Integer and decimal places	
120 : "<text>%d<text>"	The string entered for <text> defines the **M** code (number of integer places). **&d** stands for the digit that defines the M code in the aperture list.
121 : "<text>%d<text>"	The string entered for <text> defines the **N** code (number of decimal places). **&d** stands for the digit that defines the N code in the aperture list.
122 : "<text>%1d.%1d<text>"	The string entered for <text> defines the **M.N** code (number of integer and decimal places). The first variable **&1d** stands for the digit that defines the M code in the aperture list. The second variable **&1d** stands for the digit that defines the N code in the aperture list.
Coordinate zeroes	
130 : "<text>"	The string entered for <text> defines the coordinate zeroes as **LEADING**. (trailing suppressed.)
131 : "<text>"	The string entered for <text> defines the coordinate zeroes as **TRAILING**. (leading suppressed.)
132 : "<text>"	The string entered for <text> defines the coordinate zeroes as **DECIMAL**.

Example

Information in the aperture list:

```
The Gerber data unit is metric

m.n format = 3.2

absolute values

leading zeros suppressed
```

Associated definition in the translation file:

```
101 : "The Gerber data unit is metric"

110 : "absolute values"

122 : "m.n format = 1%d.1%d"

131 : "leading zeros suppressed"
```

Tips & tricks in customizing the translation file

If you find that the graphical display of the circuit board does not meet the desired result, you will generally find that incorrect conversion of the aperture lists and tool lists by the translation file is responsible.

If, for instance, the conductor tracks or pads are displayed incorrectly in form or size, it will mean that the aperture and tool definitions in the translation file do not correspond to the format of the aperture and tool list to be imported. This must now be rectified by customizing the translation file to the aperture and tool list as described above.

It is frequently the case, that just one parameter value in the translation file has been incorrectly defined, meaning that the problem can be rapidly fixed. The table below lists which parameters may be the cause of what incorrect graphical representations:

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\>"**

Sample translation files for various CAD systems

Some sample aperture lists from various CAD systems and the translation files for them prepared by LPKF are listed below.

> **Note:** These translation files were prepared using the aperture lists available to LPKF. It is possible that there are differences in individual structures or aperture definitions.
> LPKF can accept no liability for the completeness or correctness of the translation files supplied. Modifications should be made by the user.

Alegro

Translation file ("alegro.txt"):

```
APE_UNITS MIL
CONFIGURATION ConfigGerber

1,CN : "LINE     %a  D%%"
1,CC : "CIRCLE   %a  D%%"
1,EQ : "SQUARE   %a  %S  D%%"
1,NR : "RECTANGLE  %a  %b  %S  D%%"
1,NS : "FLASH %s %a D%%"

100 : "INPUT-UNITS     INCHES"
```

Associated 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        96        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 ("eagle.txt"):

```
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"
```

Associated 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 ("ecam2.txt"):

```
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"
```

Associated 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 ("pads.txt"):

```
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"
```

Associated 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 ("protel.txt"):

```
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"
```

Associated 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
```

Index