



CircuitPro 1.5

UseCases

Version 1.1

English

LPKF Laser & Electronics AG

Osteriede 7

D-30827 Garbsen

Phone	+49 5131-7095-0
Fax	+49 5131-7095-90
E-mail	info@lpkf.com
Internet	www.lpkf.com

Editor	LPKF Laser & Electronics AG Osteriede 7 D-30827 Garbsen Germany Phone: +49 5131-7095-0 Fax: +49 5131-7095-90 Email: info@lpkf.com
Order no.	10011141
Filename	CircuitPro1.5_HB_V1.1_ENG.docx
Version	1.1
Date created	31.05.2012
Date printed	31.05.2012
Copyright	© 2012 LPKF AG This document and its contents in whole and in part are subject to copyright. The reproduction, translation or duplication of the contents as photocopy or any digital form requires written permission of LPKF AG.

Information about this document

General

This document contains all information for assembly, putting into operation, operation, and maintenance or trouble-shooting of the system delivered. All information is intended for persons with basic knowledge in installation and operation of software-controlled machines. General knowledge on operational safety as well as basic knowledge on using PCs running *Microsoft Windows*® are required.

Availability

A complete and legible copy of this document must be available at the operation station. Every person who is authorised to operate the system must have read this document. The facility owner is required to ensure that all safety measures listed in this document are observed.

Notations

Various text attributes, notations, and structures are used to enhance the readability of this document. The text attributes in this document mean the following:

Attribute	Function
Bold	Important information
<i>Italics</i>	Brand name
<i>Bold italics</i>	LPKF brand name
[...]	Button
\...\ <...>	Input/display field Checkbox
{...}	Radio button
>...>...>	Menu trail
/.../	reference to a number in the figure

Figures

Figures (photos or drawings) are displayed in a frame. Each figure has a numbered figure title, e. g. "Fig. 1: Overview". Numbers in the figure mark certain components or actions. Arrows in the figure show the direction of the action.

Tables

Data, facts, and important contexts are clearly arranged in tables. Each table has a numbered table title, e. g. "Table 1: Scope of delivery". Every table has a highlighted header row containing the column headings.

Information about this document

Operation descriptions

Activities or procedures that are realised in steps are presented in activity sequences. Each sequence consists of at least three components:

Component	Meaning
■ Title	Description of the objective of the procedure marked by a preceding "■".
1. Step	Consecutively numbered sequence of an procedure.
➔ interim result	Interim result after an operation step. The procedure continues.
◆ Result	Result of the activity marked by a preceding "◆".

Symbols and signal words

The following symbols are used to mark important texts:

Symbol	Meaning
	Safety note WARNING - Hazard for persons CAUTION - Damage to the system
	Hint A hint is an information on the optimum usage of a feature.
	Note A note contains additional information.
©	Copyright
®	Registered Trademark

Registered Trademarks and Brand names

The LPKF Logo and the LPKF product names are registered trademarks of LPKF Laser & Electronics AG.

Microsoft and *Windows* are brand names or registered trademarks of Microsoft Corporation in the USA and/or other countries.

All other trademarks belong to the respective owner.

Standards

The following standards and guidelines were observed when creating this document:

Standard	Meaning
DIN 5008 05-2005	Rules for writing and layouting
VDI 4500 Bl.1,2 11-2006	Technical documentation - Definitions and legal basics
ISO/IEC 26514 11-2008	Requirements for designers and developers of user documentation
IEEE 1063 05-1987	Standard for software user documentation
ISO/IEC 9294 05-1990	Guidelines for the management of software documentation
ISO/IEC 18019 01-2004	Software and system Engineering – Guide lines for design and preparation of user documentation for software

Table of contents

I First steps in CircuitPro

1	How to produce a PCB	13
1.1	Switching on the machine and starting CircuitPro	14
1.2	Selecting a template and creating a new document.....	15
1.3	Importing files.....	16
1.4	Inserting rubout areas	19
1.5	Multiplying the design if needed	21
1.6	Inserting fiducials	23
1.7	Creating toolpaths.....	25
1.8	Loading the tool magazine and assigning the tools to positions	26
1.9	Starting processing	28
2	Dispensing solder paste using the ProtoMat S63 or S103.....	39
2.1	Starting the machine and CircuitPro	40
2.2	Selecting a template and creating a new document.....	41
2.3	Importing data	42
2.4	Creating solder paste paths.....	45
2.5	Creating toolpaths.....	47
2.6	Dispense preparation.....	49
2.7	Starting dispensing	55

II How to do

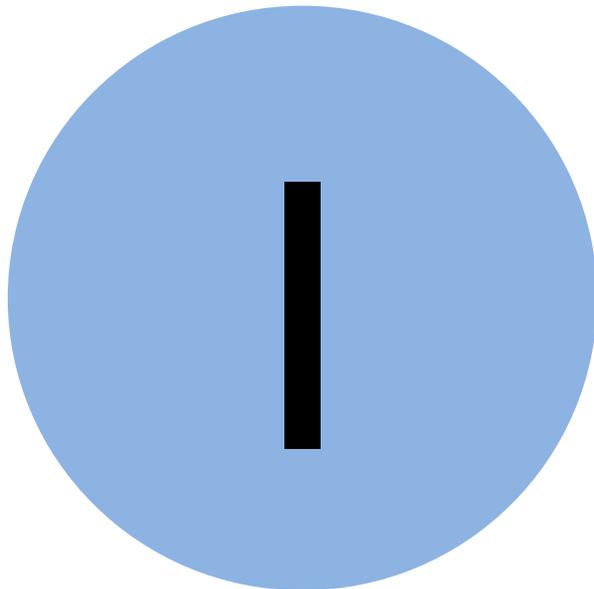
1	Creating a 3D part.....	65
1.1	Starting the machine and CircuitPro	66
1.2	Selecting a template and creating a new document.....	67
1.3	Creating the layout.....	69
1.4	Creating 2.5D objects	73
1.5	Creating toolpaths.....	76
1.6	Loading the tool magazine and assigning tools to holder positions	78
1.7	Starting processing	81
2	Creating a 3D part from a STEP file	89
2.1	Starting the machine and CircuitPro	90
2.2	Selecting a template and creating a new document.....	91
2.3	Importing data	93
2.4	Creating 2.5D toolpaths	97
2.5	Inserting fiducials	99
2.6	Creating toolpaths for fiducials	103
2.7	Loading the tool magazine and assigning tools to holder positions	106

3	Creating a multi-layer PCB with galvanic through-hole plating ..	111
3.1	Starting the machine and CircuitPro	112
3.2	Selecting a template and creating a new document.....	113
3.3	Importing data	115
3.4	Inserting rubout areas	117
3.5	Multiplying the design (if necessary).....	119
3.6	Creating toolpaths.....	121
3.7	Loading the tool magazine and assigning tools to holder positions	122
3.8	Starting the processing	124
4	Creating a front panel.....	137
4.1	Starting the machine and CircuitPro	138
4.2	Selecting a template and creating a new document.....	139
4.3	Creating the front panel design.....	140
4.4	Creating toolpaths.....	153
4.5	Loading the tool magazine and assigning tools to holder positions	156
4.6	Start processing	158
5	Creating a polyimide stencil	165
5.1	Starting the machine and CircuitPro	166
5.2	Selecting a template and creating a new document.....	167
5.3	Importing data	168
5.4	Processing data	170
5.5	Creating toolpaths.....	172
5.6	Loading the tool magazine and assigning tools to holder positions	176
5.7	Starting the processing	178
6	Creating a flexi-rigid PCB	187
6.1	Starting the machine and CircuitPro	188
6.2	Selecting a template and creating a new document.....	189
6.3	Importing data	193
6.4	Processing the individual physical layers	198
6.4.1	Drilling and milling the flexible material	199
6.4.2	Processing the FR4 material	209
6.4.3	Processing the prepreg material	219
6.5	Bonding the individual physical layers	233
6.6	Contour routing the bonded PCB.....	234

III Basics

1	CircuitPro: Basic CAM operations	251
1.1	Executing Process planning wizard	252
1.2	Importing the Gerber files	256
1.3	Importing the drill file	258
1.4	Inserting a rubout area (optional).....	261
1.5	Inserting fiducials (optional)	263
1.6	Creating toolpaths	265
2	CircuitPro: Basic machining operations	275
2.1	Switching to Machining view	276
2.2	Loading the tool magazine.....	277
2.3	Starting processing	280
3	Processing DXF files in CircuitPro.....	291
3.1	Importing the DXF file	292
3.2	Converting the DXF file.....	295
4	Processing Gerber and Excellon files	303
4.1	Selecting Gerber and Excellon files.....	304
4.2	Selecting the file format	307
4.3	Selecting the desired target layer	308
4.4	Setting/correcting size and format	309
4.5	Viewing/modifying aperture properties	312
4.6	Using the layer name defined in the Gerber file	314
	Appendix.....	317

First steps in CircuitPro 1.5



How to produce a PCB

1 How to produce a PCB

This tutorial shows you how to produce a double-sided circuit board without through plating.

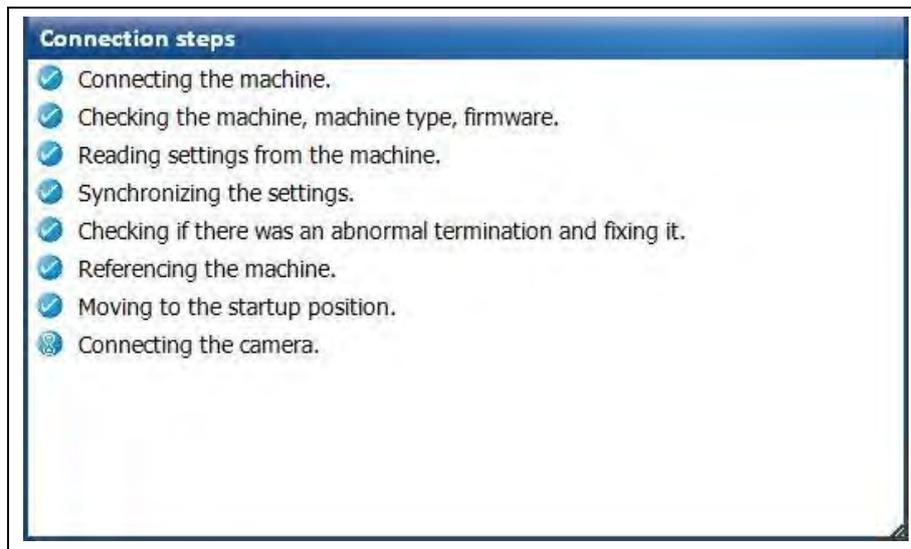
The following steps are necessary to complete the tutorial successfully:

- i. Switching on the machine and starting CircuitPro
- ii. Selecting a template and creating a new document
- iii. Importing data
- iv. Inserting rubout areas
- v. Multiplying the design if needed
- vi. Setting fiducials
Creating toolpaths
- vii. Loading the tool magazine and assigning the tools to positions
- viii. Starting production

1.1 Switching on the machine and starting CircuitPro

- Switching on the machine and starting CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed.

Fig. 1:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to the reference points and stops at the Pause position.

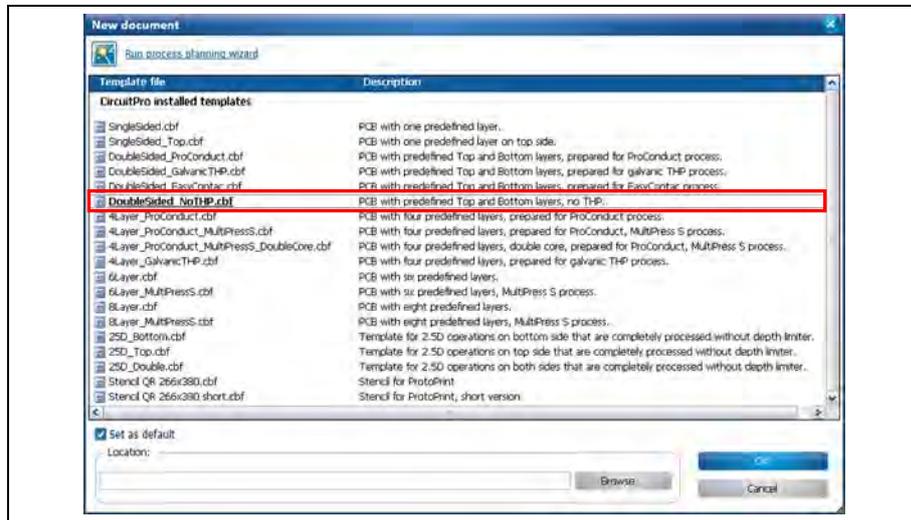
1.2 Selecting a template and creating a new document

■ Selecting a template and creating a new document

1. Click on File > New...

➔ Following dialog is displayed:

Fig. 2: New document



2. In order to produce a double-sided PCB without through-hole plating, select the template "DoubleSided_NoTHP".

3. Click on [OK].

4. Click on File > Save As...

5. Enter a file name.

6. Select the memory location.

7. Click on [Save].

➔ The file was saved.

◆ The template was selected and the new document was created.

1.3 Importing files



Tip

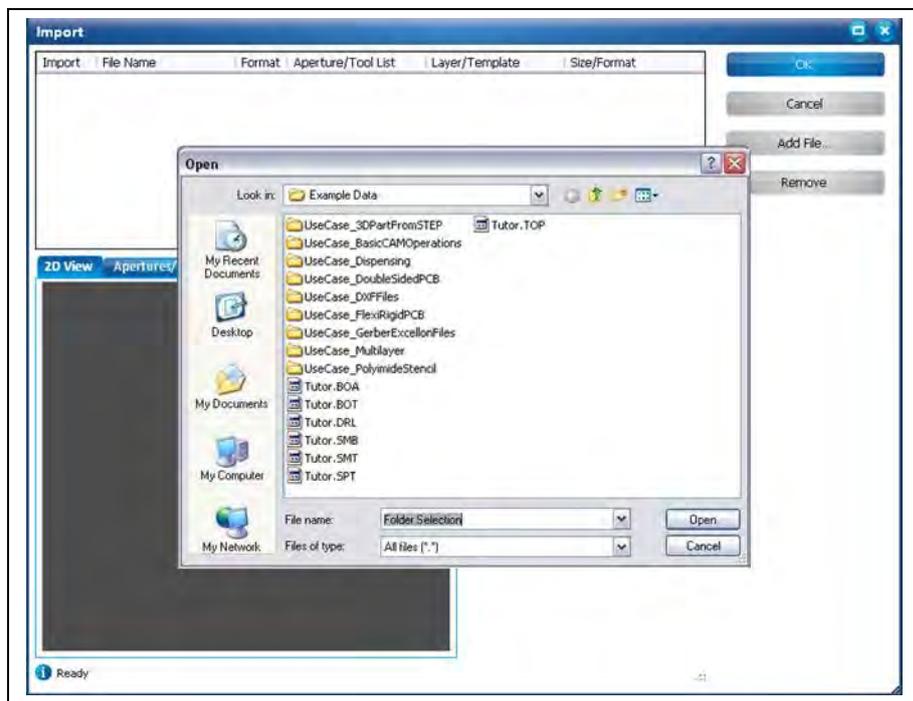
The LPKF tutor data are stored in “My Document\LPKF Laser & Electronics\ LPKF CircuitPro 1.5\Example Data”.

■ Importing data

1. Click on File > Import...

➔ Following dialog is displayed:

Fig. 3: Import



2. Select the following data:

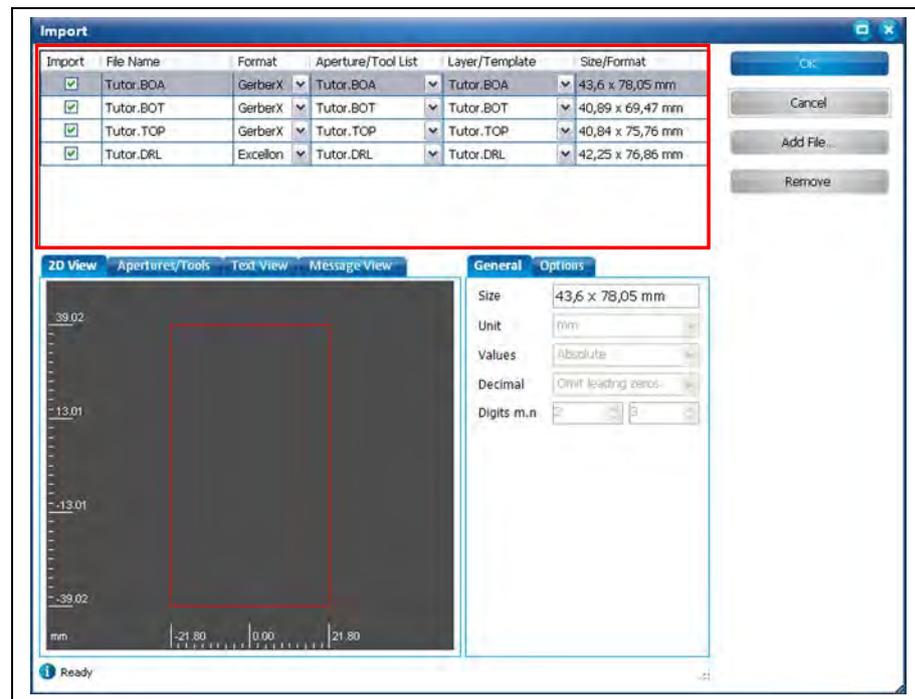
- Tutor.BOA
- Tutor.BOT
- Tutor.DRL
- Tutor.SMB
- Tutor.SMT
- Tutor.SPT
- Tutor.TOP

3. Click on [Open].

- Assign the imported file to their corresponding layers according to the following table:

File	Layer
.BOA	Board Outline
.BOT	Bottom Layer
.TOP	Top Layer
.DRL	Drilling Unplated

Fig. 4: Assigned layers



Note

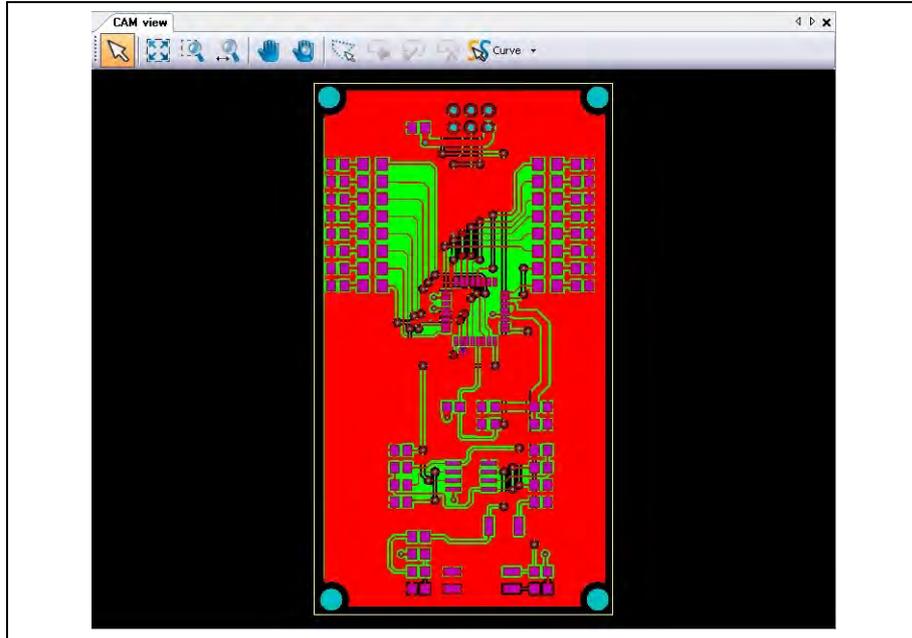
Instead of manually assigning the individual files to the layers, you can activate the options "Use layer name" and "Apply to all Gerber files". Activate the corresponding checkboxes in the tab "Options".

If a file contains layer names these are automatically assigned. Please note that this is only available for Gerber files. All other files require assigning the layers manually via the drop-down menu.

5. Click on [OK].

➔ The data is displayed in the CAM view:

Fig. 5: CAM view



◆ The data is imported.

1.4 Inserting rubout areas

Inserting rubout areas is used for creating a most precise isolation in certain areas by removing all redundant copper. This is a preferred option for fine-pitch arrays for example.

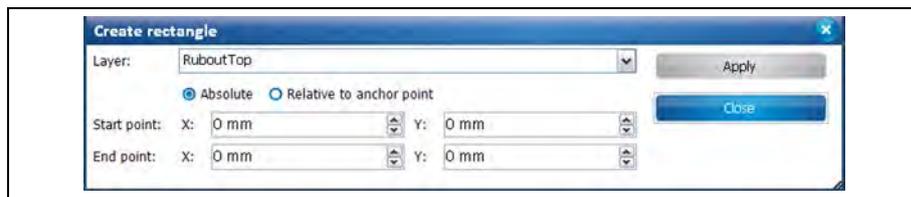
In the existing design, a rubout area around the terminals of the IC is needed.

- Inserting a rubout area.

1. Click on Insert > Rubout area > RuboutTop.

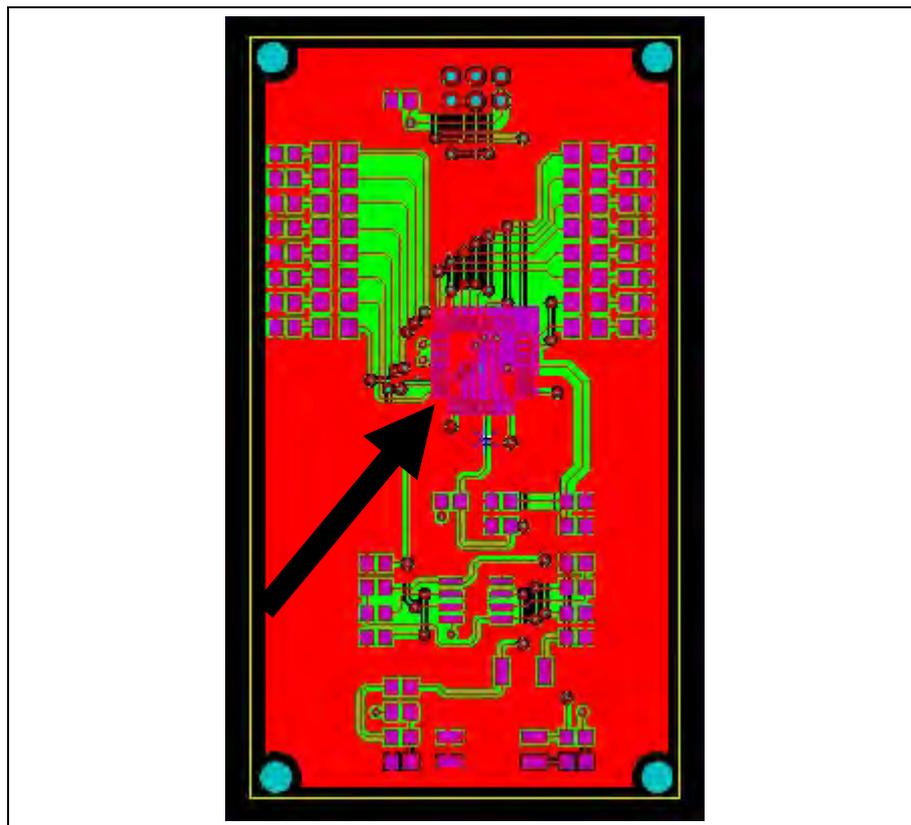
➔ Following dialog is displayed:

Fig. 6: Create rectangle



2. Draw a rectangle around the contact pads of the IC using your mouse (see arrow):

Fig. 7: Rectangle around the pads



3. Click on [Close].

➔ The dialog is closed.

◆ The rubout area was inserted.

**Note**

Depending on which layer the rubout area is to be created, it can be useful to hide the other layers for drawing the rubout area.

The display mode of the objects on the individual layers can be set in the Layers pane. You can choose between

- True width (filled objects are displayed filled, polylines are displayed with their true width)
- Outline (only outlines of the filled objects and polylines are displayed)
- Thin line (outlines of filled objects and thin line without defined width in case of polylines are displayed) and
- Unknown (used when importing CAM files with undefined objects)

1.5 Multiplying the design if needed

For producing whole panels, the design can easily be multiplied and placed on the base material depending on the size of the design and of the base material.

■ Multiplying the design

1. Select the whole design



Note

Please make sure, that the layers' option "selectable" is enabled if the layer contains data.

You will find the option in the "layers" pane.

2. Click on Modify > Step & Repeat.

➔ Following dialog is displayed:

Fig. 8: Step & Repeat



3. Enter "2", in the \ Repetition X\ field.
4. Enter "100", in the \ Distance X \ field.



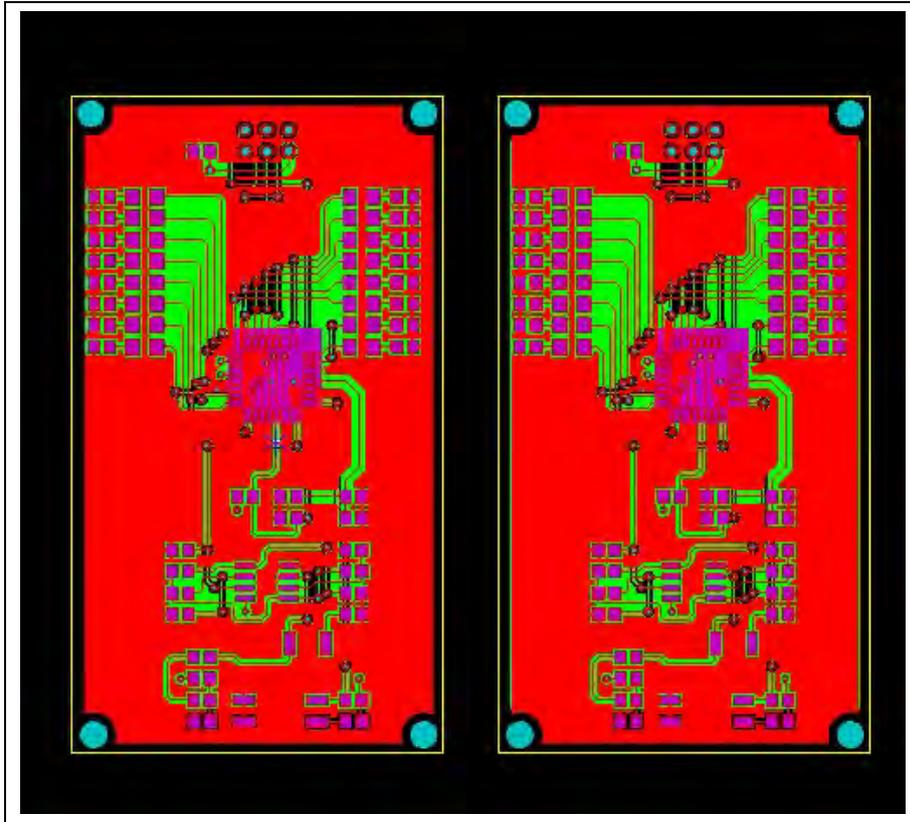
Note

The "Distance" values reflect the size of the design. The value for spacing between the designs has to be added.

5. Click on [Apply].

- ➔ The design is multiplied in X direction

Fig. 9: Layout in CAM view



6. Click on [Close].
- ➔ The dialog is closed.
- ◆ The design was multiplied.

1.6 Inserting fiducials

For aligning the top and bottom sides of the circuit board you need fiducials.

Fiducials are optical marks on the surface of the circuit board with a defined diameter of 1.5 mm.

The fiducials are drilled into the board and are recognised by the cameras of the ProtoMat systems.



Note

For working with fiducials you need the camera system for fiducial recognition.



Tip

Ideally you insert four fiducials for aligning the top and the bottom sides.

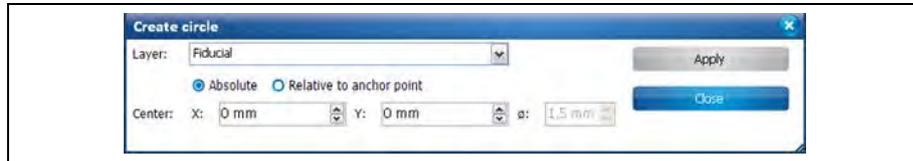
You are also able to work with two fiducials. In this case you have to insert them diagonally into the layout.

■ Inserting fiducials

1. Click on Insert > Fiducial > Fiducial.

➔ Following dialog is displayed:

Fig. 10: Create circle



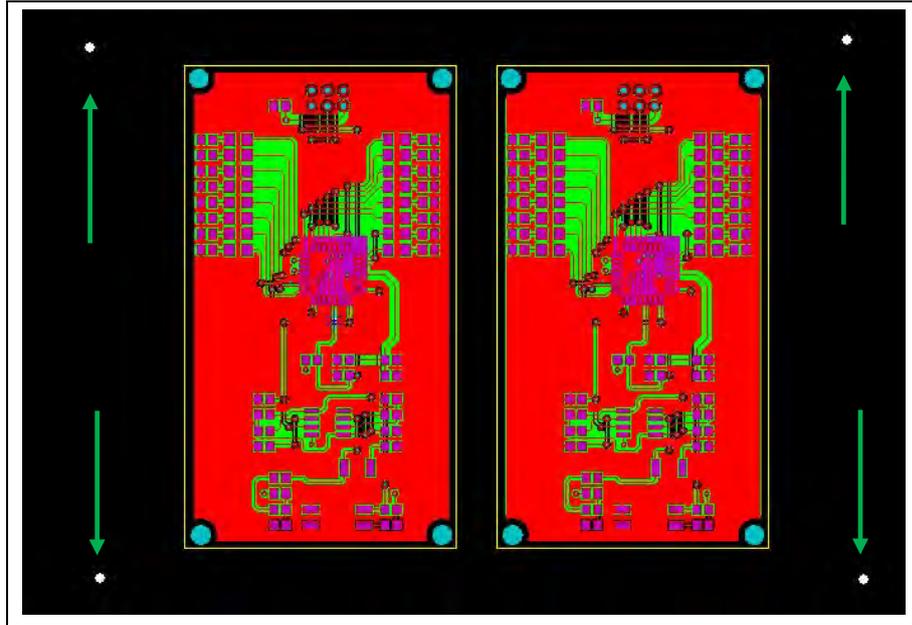
2. Left click in the CAM view where you want to place each fiducial hole.

Or

2. Create the fiducial using the dialog "Create circle" and entering the X and Y position of each fiducial.

➔ The design now looks like follows:

Fig. 11: Two
fiducials



3. Click on [Close].
 - ➔ The dialog is closed.
 - ◆ The fiducials were inserted.

1.7 Creating toolpaths

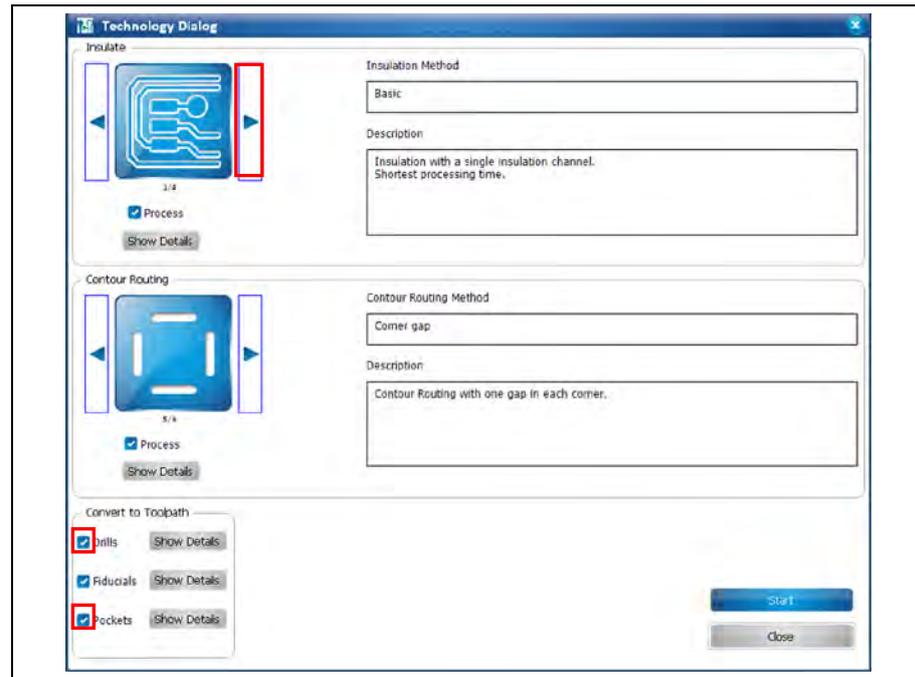
Toolpaths have to be created from the imported data, for producing the circuit board.

■ Creating toolpaths

1. Click on Toolpath > Technology Dialog...

➔ Following dialog is displayed:

Fig. 12:
Technology
Dialog



Note

In the technology dialog, several settings can be modified by clicking on the [Show details] buttons. For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

2. Click on the right-pointing arrow button in the "Isolation" section until the "Partial rabout" method is selected.
3. Disable the following functions by clicking on the corresponding checkmarks:
 - Drills
 - Pockets
4. Click on [Start].
 - ➔ The results for the generated toolpaths are displayed.
5. Click on [Close].
 - ➔ The dialog with the computation results is closed.
 - ◆ The toolpaths were created.

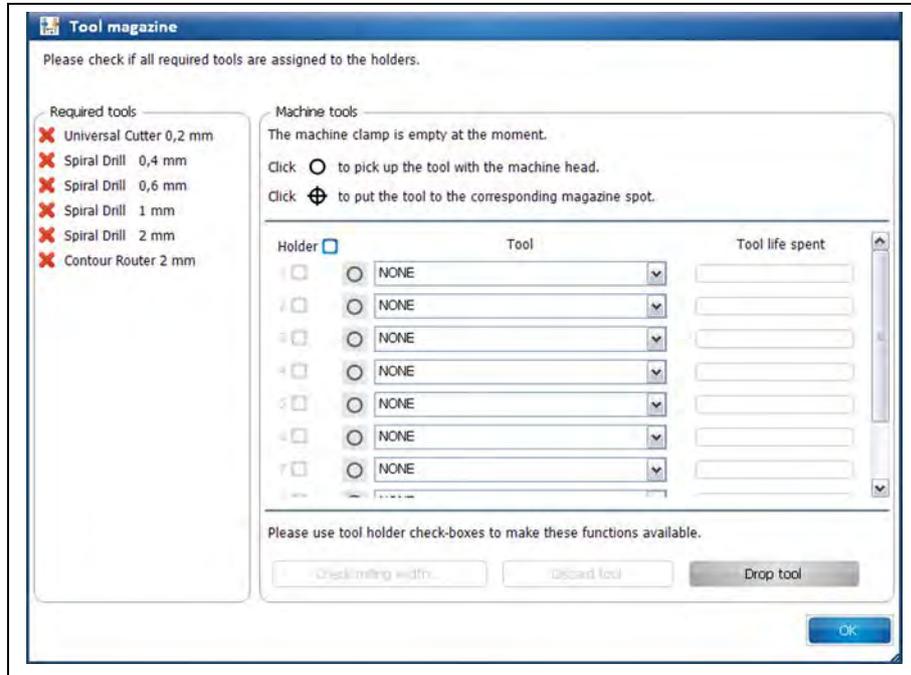
1.8 Loading the tool magazine and assigning the tools to positions

- Loading the tool magazine and assigning the tools to positions

1. Click on Edit > Tool magazine...

➔ Following dialog is displayed:

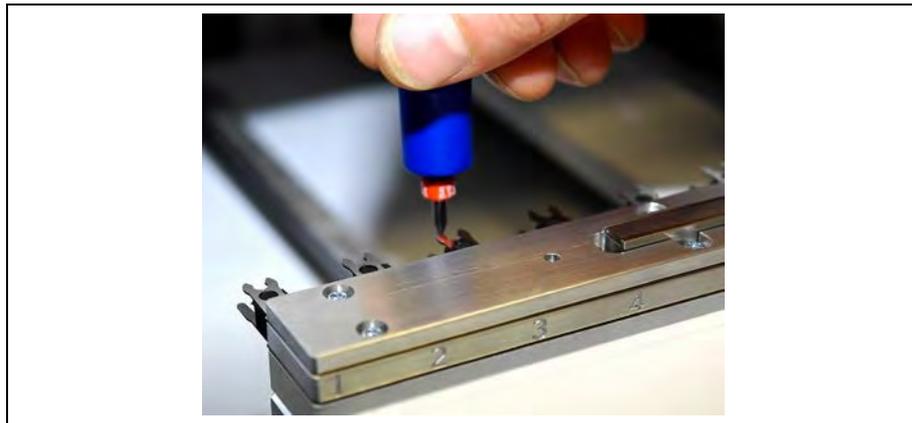
Fig. 13: Tool magazine



➔ The required tools for the job are displayed. Missing tools which are required for the job are marked with a red "X".

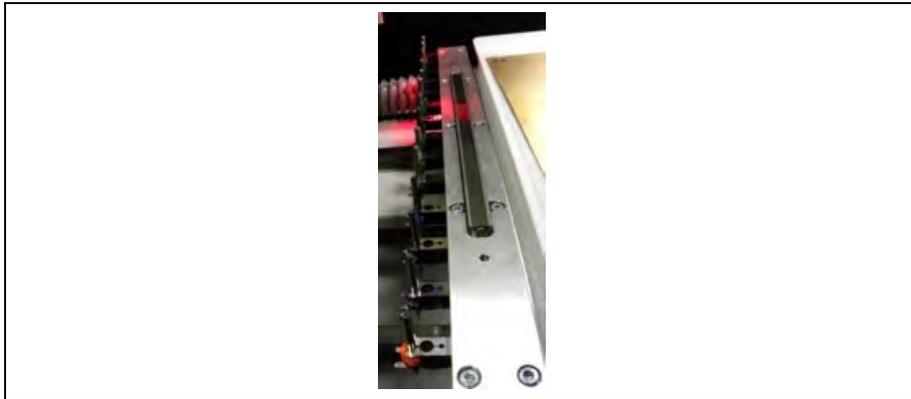
2. Insert the required tool into the tool magazine:

Fig. 14: Inserting the tool



3. In the dialog, assign the tool to the respective tool magazine position used.
4. Repeat the steps 2 and 3 until all required tools are assigned:

Fig. 15: Tools in the tool magazine



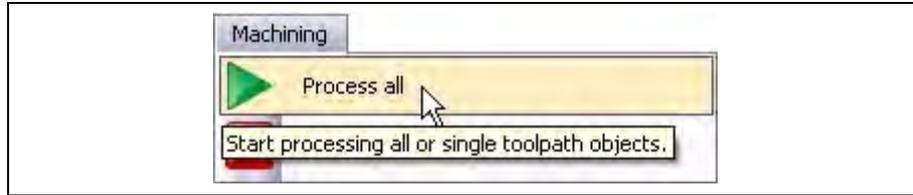
5. Click on [OK].
 - ➔ The dialog is closed.
 - ◆ The tools were loaded and assigned to their positions.

1.9 Starting processing

■ Starting processing

1. Click on Machining > Process all.

Fig. 16:
Machining >
Process all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the “Start processing” button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the “ladder” button. The selected phase and all following phases are processed in the correct order.

After the production started, the machine will process following phases in order. The phases are displayed via prompts.



Note

Depending on which ProtoMat you use the following phases could differ from the phases and messages displayed on your screen. Please follow the instructions on your screen.

For machines with manual tool exchange you are regularly asked to change the tool in the collet, for example.

Phase “MountMaterial”

1. Place the base material onto the machine’s table top.
2. Fasten the base material onto the table top using the adhesive tape.

Phase "MaterialSettings"

➔ Following dialog is displayed:

Fig. 17: Material settings

■ Entering the material settings

1. Enter the correct values for the material used.

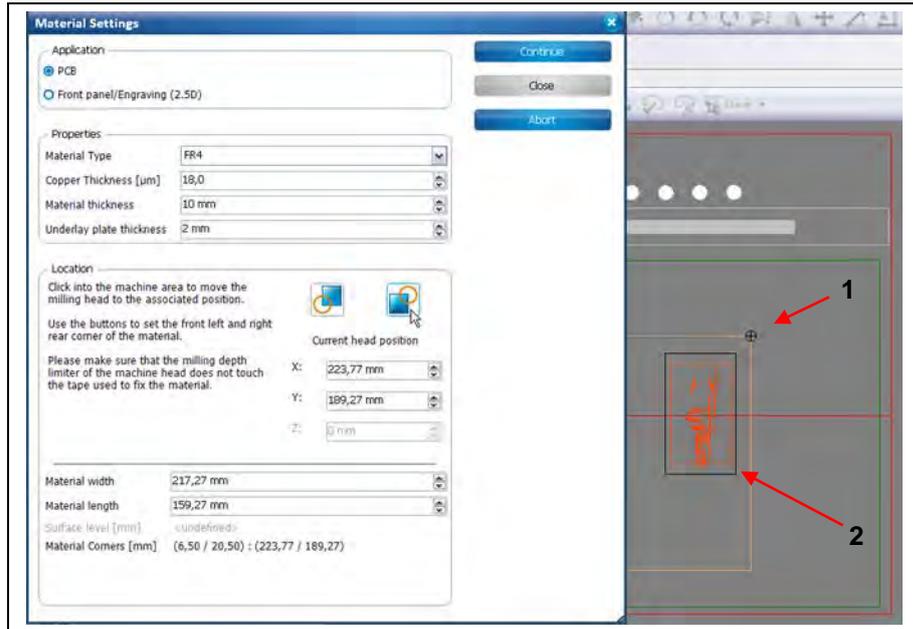


PCB is selected as default.

Note

2. Change the values of copper thickness and material thickness necessary.
3. Define the processing area:
 - a) Move the dialog „Material Settings“ off to the side.
 - b) Using your mouse in the machining view, click on the right corner of your material:

Fig. 18: Right corner



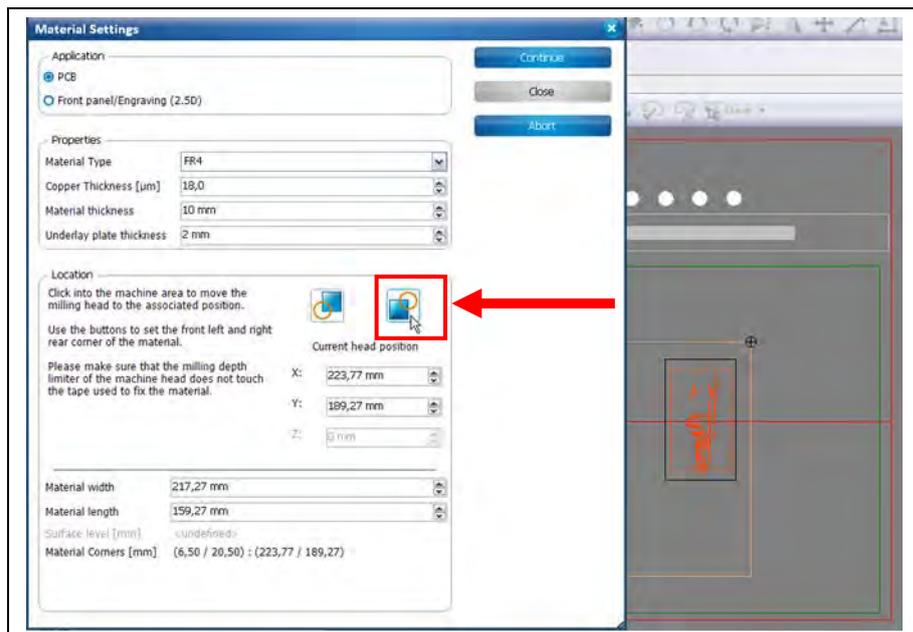
/1/ Click in the machining view

/2/ Material

➔ The machine head moves to this position.

c) Now click on the corresponding button in the dialog "Material Settings":

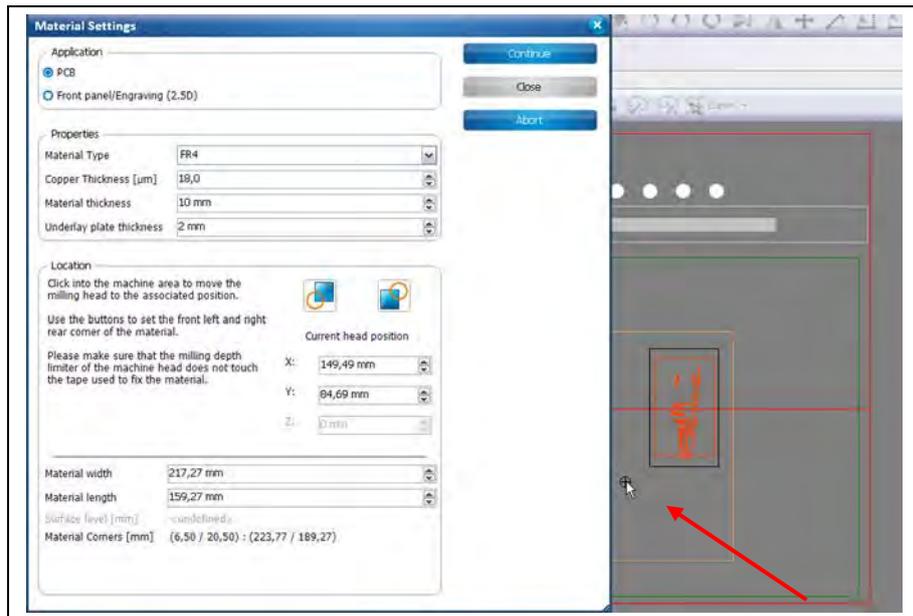
Fig. 19: Click on button



➔ The processing area is adapted.

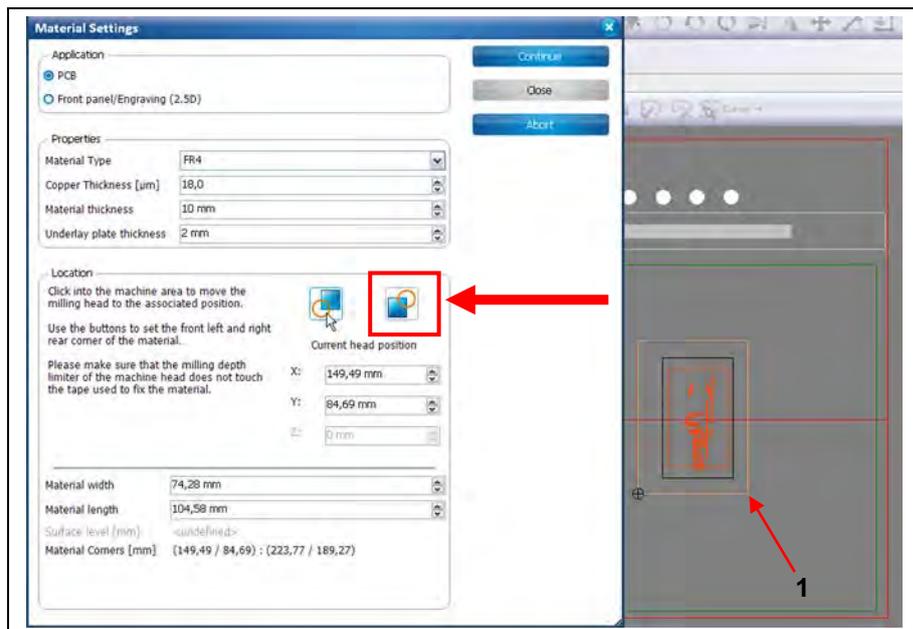
d) Using your mouse in the machining view, click on the left corner of your material:

Fig. 20: Left corner



- ➔ The machine head moves to this position.
- e) Now click on the corresponding button in the dialog "Material Settings":

Fig. 21: Defined processing area



/1/ Defined processing area

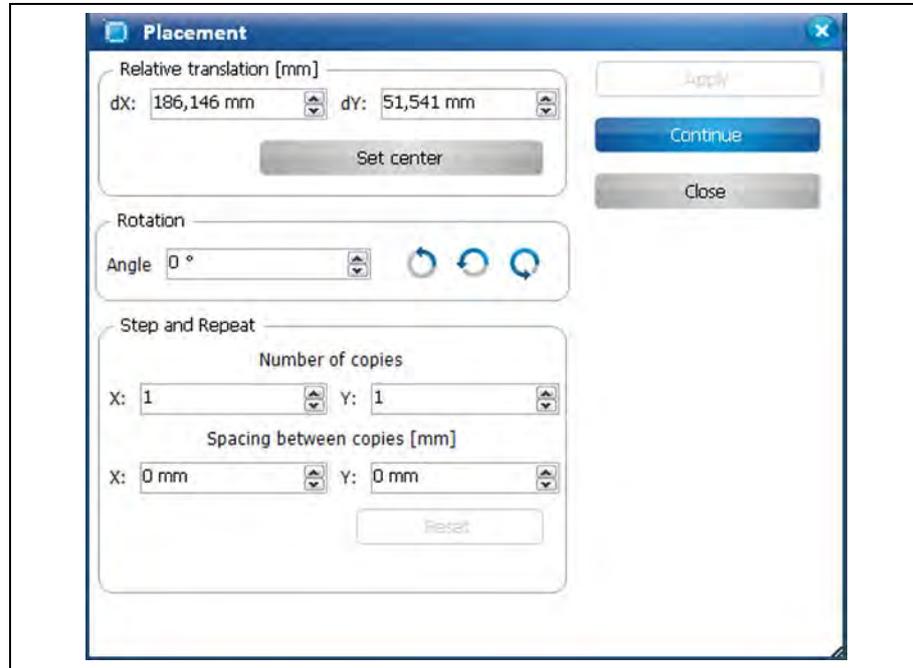
- ➔ The processing area was adapted to the material.
- 4. Click on [Continue].
- ◆ The material settings were entered.

Phase “Placement”

In this phase, the job can be positioned, rotated and multiplied within the processing area.

➔ Following dialog is displayed:

Fig. 22:
Placement



1. Drag the job to the desired position or use the dialog to position the job.
2. If desired, rotate the job by entering a rotation angle.
3. If desired, multiply the job by using the settings in the “Step and Repeat” section of the dialog.
4. Click on [Continue].

Phase “DrillFiducial”

➔ The machine picks up the tool “Spiral Drill 1.5 mm” and drills the fiducials.



If the spindle motor has not been used before, the motor is warmed up for 2 minutes.

Note

Phase “Marking Drills”

➔ The machine picks up the tool “Universal Cutter” and marks the positions for the drill holes.

Phase “Drilling Unplated”

- ➔ The machine picks up the required tool and drills the holes.



This phase may use more than one tool.

Note

Phase “Milling Bottom”

- ➔ The machine picks up the required tool and mills the isolation tracks.

Phase “Flip Material”

1. Flip the material.



If you are using a ProtoMat S43, S63 or S103 flip the material along the machine’s X-axis.

Note

If you are using a ProtoMat E33 flip the material along the machine’s Y-axis.

2. Confirm by clicking [OK].



The display in the machining view changes. The position of the design is adjusted to the circuit board. The side of the circuit board to be processed is now the “Top” side.

Note

Phase “Read Fiducials_Top”

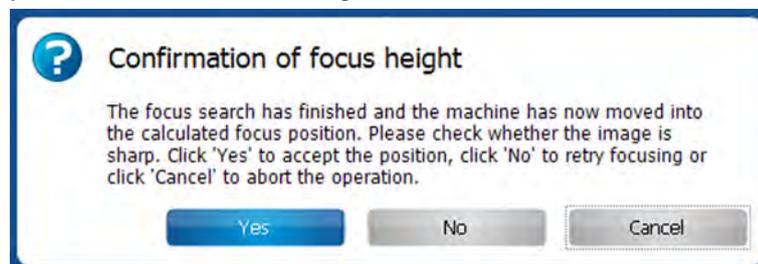
S43, S63 and S103



If the fiducial search is performed for the first time (after having started CircuitPro) the camera is performing an autofocus five times.

Note

Afterwards the following message is displayed which prompts you to confirm the focus height:



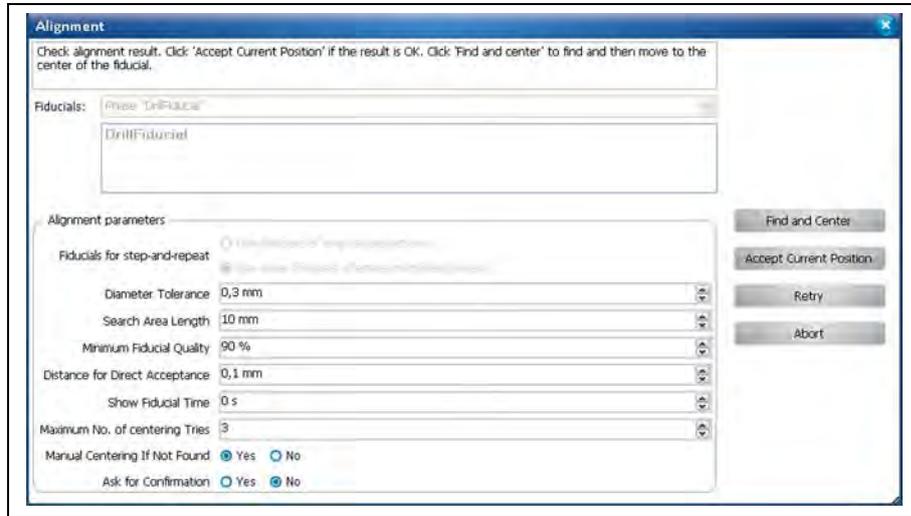
- ➔ The camera moves to the positions of the fiducials and determines the exact position.

If the material is placed at nearly the same position as before, the positions of the fiducials are recognised automatically.

The Top side is thus aligned to the Bottom side.

The following dialog is displayed if the fiducials have not been recognized automatically:

Fig. 23:
Alignment



1. Enlarge the search area by increasing the value of the field \Search Area Length\.
2. Start the search again.
3. Repeat above steps if necessary.



Note

Enlarging the search spiral increases the time required for searching the fiducials. Try to put the material at the same position as before when turning the material over (if this is not predetermined by reference pins).

E33, S43 without camera

If no camera is available for processing the “Read Fiducials_Top” phase, the Top side is aligned to the Bottom side using the reference pins. The “Read Fiducials_Top” phase is not processed in this case.

Phase “Milling Top”

- ➔ The machine picks up the required tools and mills the isolation tracks on the Top side.

Phase “Contour Routing”

- ➔ The machine picks up the required tools and drills and mills the outline of the circuit board.

Phase "Board Production Finished"

- ➔ A message informs you that the production is finished.
- ◆ The production of the circuit board is finished.



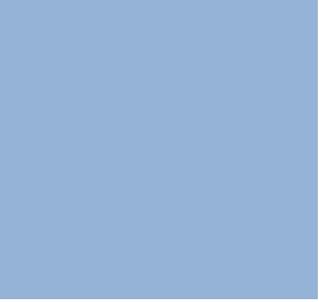
Note

If desired, continue with dispensing solder paste on your PCB. Therefore please refer to the tutorial "Dispensing solder paste using the ProtoMat S63 or S103".



Tip

After PCB production you can carry on with inserting labeling or applying solder resist. For these cases you can purchase the LPKF systems ProMask and ProLegend. Furthermore LPKF offer systems for applying solder paste, equip and solder PCBs.



**Dispensing solder
paste using a
ProtoMat S63 or S103**

2 Dispensing solder paste using the ProtoMat S63 or S103

This tutorial shows you how to dispense solder paste on your board with CircuitPro.

The following steps are necessary to complete the tutorial successfully:

- i. Starting the machine and CircuitPro
- ii. Importing data
- iii. Creating solder paste paths
- iv. Dispense preparation
- v. Starting dispensing



Note

Before you start dispensing, your PCB board's drilling and milling must be finished (for more information about milling and drilling, see the other tutorials in this document).



Note

This tutorial is based on the tutorial "How to produce a PCB".

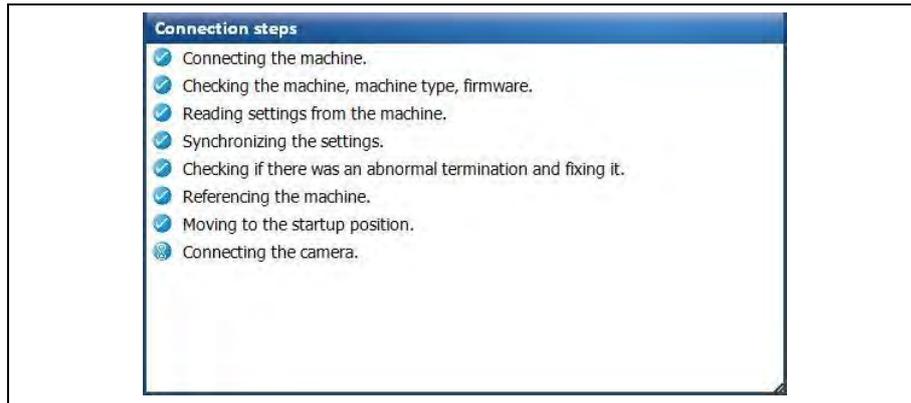
Requirements:

- Dispenser head
- Red plastic needle
- Solder paste (at room temperature)
- Calipers (capable of measuring in millimeters)
- Completed PCB
- Gerber Files (Solder Paste and Drill Layers)
- Compressed Air (@45 psi or 3 bar)

2.1 Starting the machine and CircuitPro

- Starting the machine and CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed:

Fig. 24:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to its reference points and subsequently moves to the Pause position.

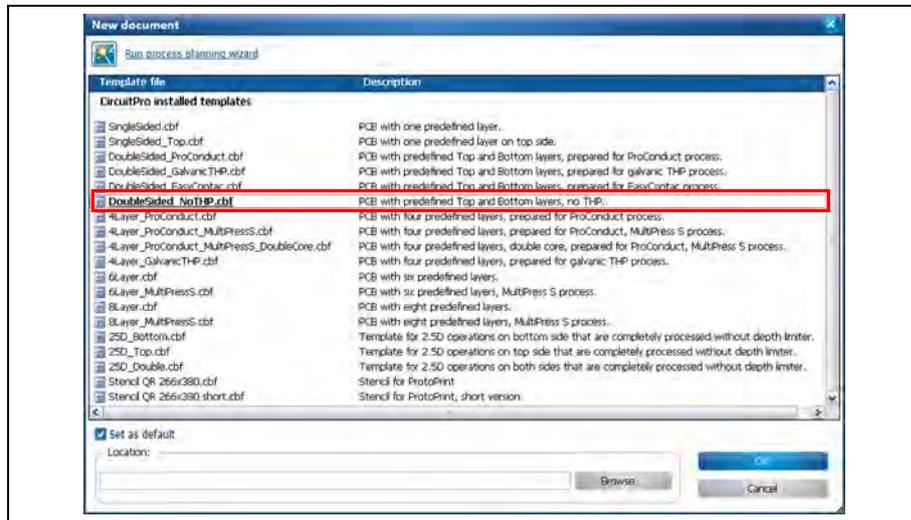
2.2 Selecting a template and creating a new document

■ Selecting a template and creating a new document

1. Click on File > New...

➔ Following dialog is displayed:

Fig. 25: New document



2. Select the template "DoubleSided_NoTHP".

3. Click on [OK].

4. Click on File > Save As...

5. Enter a file name.

6. Select the memory location.

7. Click on [Save].

➔ The file was saved.

◆ The template was selected and the new document was created.

2.3 Importing data



Tip

The LPKF tutor data are stored in folder “My documents\LPKF Laser & Electronics\LPKF CircuitPro 1.5\Example Data”.



Note

Before you start dispensing, your PCB board’s drilling and milling must be finished.



Note

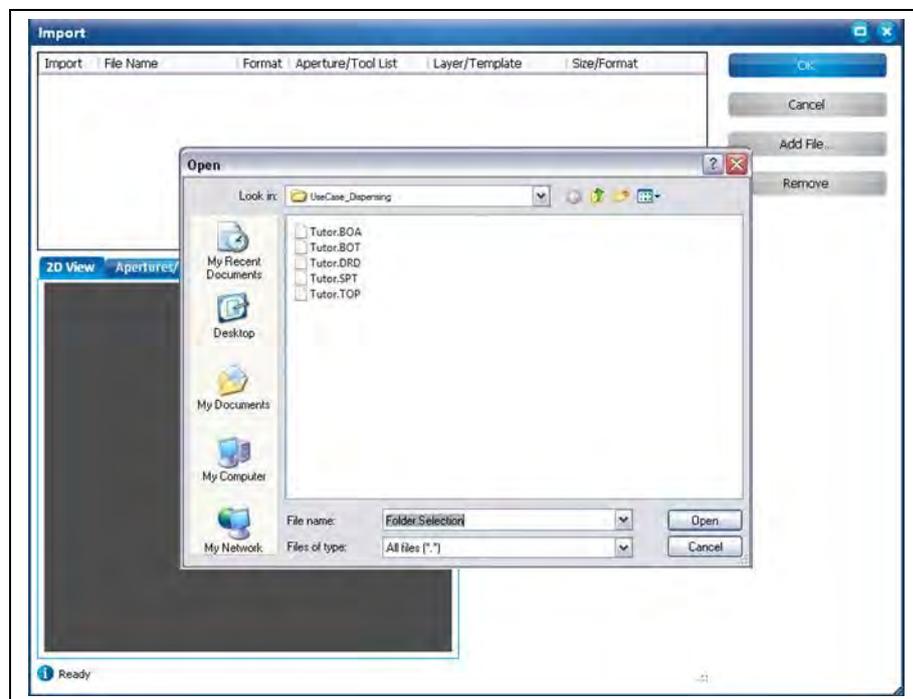
In this case, the PCB created in the tutorial “How to produce a double-sided PCB” is necessary to dispense solder paste on it. You can use another file as well if it includes solder paste data.

■ Importing data

1. Click on File > Import...

➔ The following dialog is displayed:

Fig. 26: Import



2. Select following files in the folder “UseCase_Dispending”:

- Tutor.SPT (contains the data for dispensing)
- Tutor.DRD (contains the drill data)
- Tutor.BOA



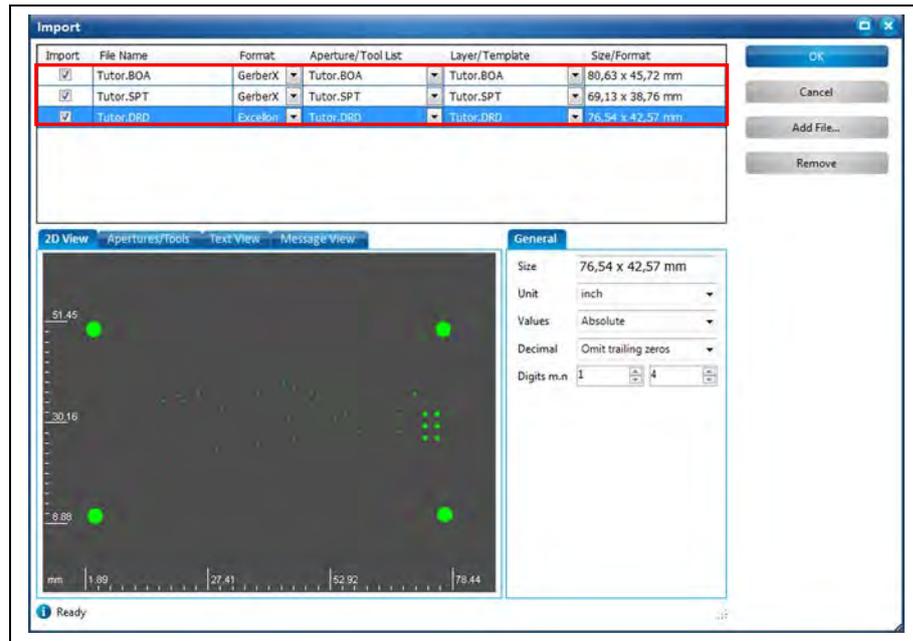
Note

To display the Excellon file "Tutor.DRD" correctly, the external dimensions of the PCB must be available. Therefore you have to import the file "Tutor.BOA".

3. Click on [Open].

➔ The files are shown in the table:

Fig. 27: Assign layer



4. Assign the imported files to the corresponding layers according to the following table:

File	Layer
.SPT	Solder Paste Top
.DRD	Drilling Unplated
.BOA	Board Outline



Note

Instead of manually assigning the individual files to the layers, you can activate the options "Use layer name" and "Apply to all Gerber files". Activate this corresponding checkboxes in the tab "Options".

If a file contains layer names these are automatically assigned. Please note that this is only available for Gerber files. All other files require assigning the layers manually via the drop-down menu.



Tip

For dispensing you only need the layers "SolderPasteTop" to dispense solder paste and "DrillingUnplated" to recognise the position.

The data on the other layers may help you to determine the side/alignment of the circuit board.

5. Click on [OK].

➔ The data are displayed in the CAM view.

◆ The data are imported.



Tip

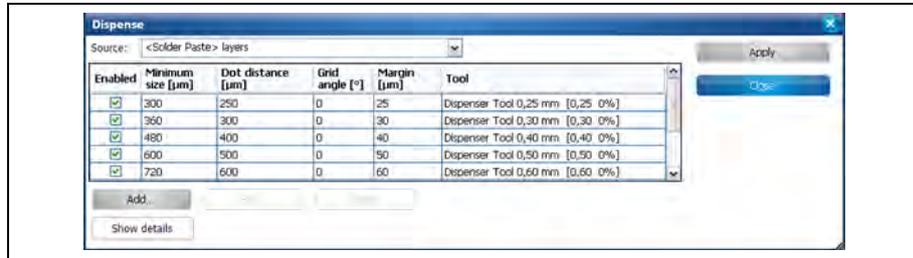
In the "Layers" pane you can hide the other layers for a clear display.

Therefore you remove the checkmark in the column "visible" of the according layer.

2.4 Creating solder paste paths

- Creating solder paste paths
- 1. Click on Toolpath > Dispense.
- ➔ Following dialog is displayed:

Fig. 28: Dispense



Note

The dialog “Dispense” allows you to make different settings concerning solder paste paths. For a detailed description, see the corresponding chapter in den CircuitPro compendium.

2. Enable the desired dispenser tool you want to use by marking the corresponding checkbox.

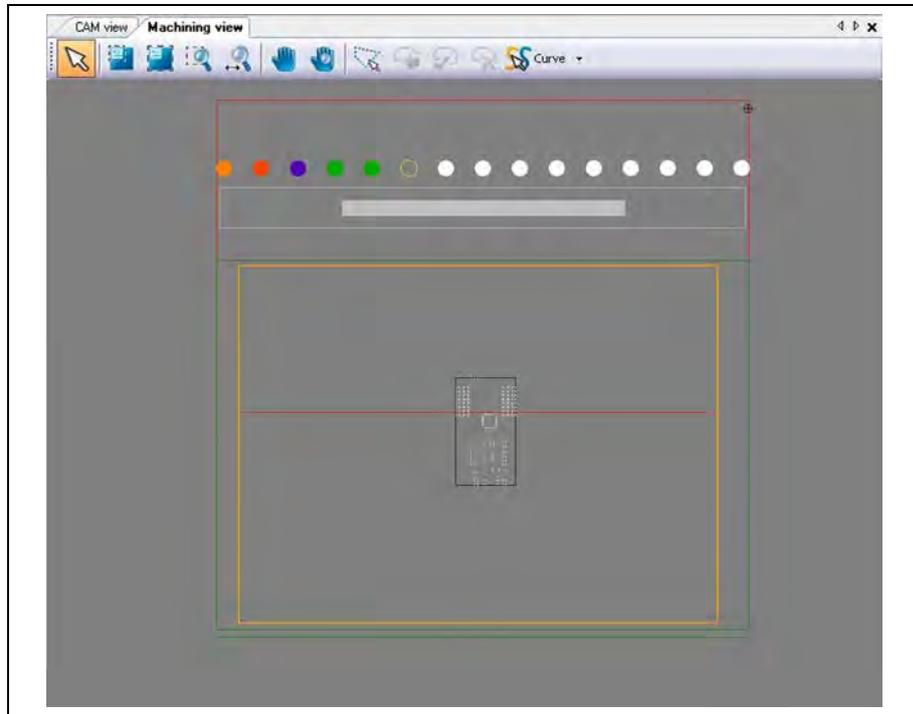
The following table describes the properties of the dispense tools:

Column	Description
Enabled	Enables/Disables the corresponding tool via checkmark.
Minimale size [µm]	Specifies the minimum pad size of the pad to be filled.
Dot distance [µm]	Specifies the dot in µm distance between the single solder paste dots.
Grid angle [°]	Specifies the rotation of the solder dot grid on the pad.
Margin [µm]	Specifies the minimum margin in [µm] between the solder paste dots and the pad margin.
Tool	Specifies the corresponding dispense tool.

3. Click on [Apply].

➔ The solder paste paths are shown in the machining view:

Fig. 29:
Machining view
with solder paste
paths



◆ The solder paste paths were created.

2.5 Creating toolpaths

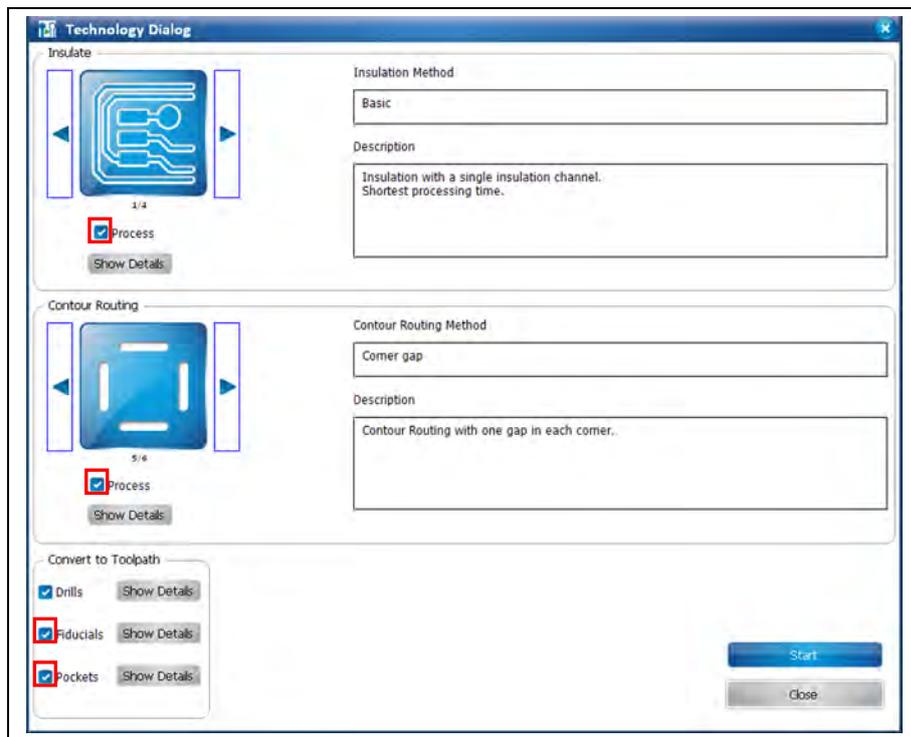


Note

Before you start dispensing, you have to create the toolpaths for the drill data. The drill holes are necessary to align the structured and drilled PCB.

- Creating toolpaths
- 1. Click on Toolpath > Technology Dialog...
- ➔ Following dialog is displayed:

Fig. 30:
Technology
Dialog



Note

In the technology dialog, several settings can be modified by clicking on the [Show details] buttons. For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

2. Disable the following functions by clicking on the corresponding checkmarks:
 - Insulate
 - Contour Routing
 - Fiducials
 - Pockets

3. Click on [Start].
 - ➔ The results for the generated toolpaths are displayed.
4. Click on [Close].
 - ➔ The dialog with the computation results is closed.
 - ◆ The toolpaths were created.

2.6 Dispense preparation

In this chapter you will learn how to set the dispenser offset by using the “Dispense preparation wizard”.

You will pass through following steps:

- i. Mounting the material
- ii. Setting the material height
- iii. Cleaning the dispenser needle
- iv. Calculating the dispenser offset

1. Click on Wizards > Dispense preparation wizard:

Fig. 31: Wizards > Dispense preparation wizard



- ➔ The following dialog is displayed:

Fig. 32: Prepare dispenser



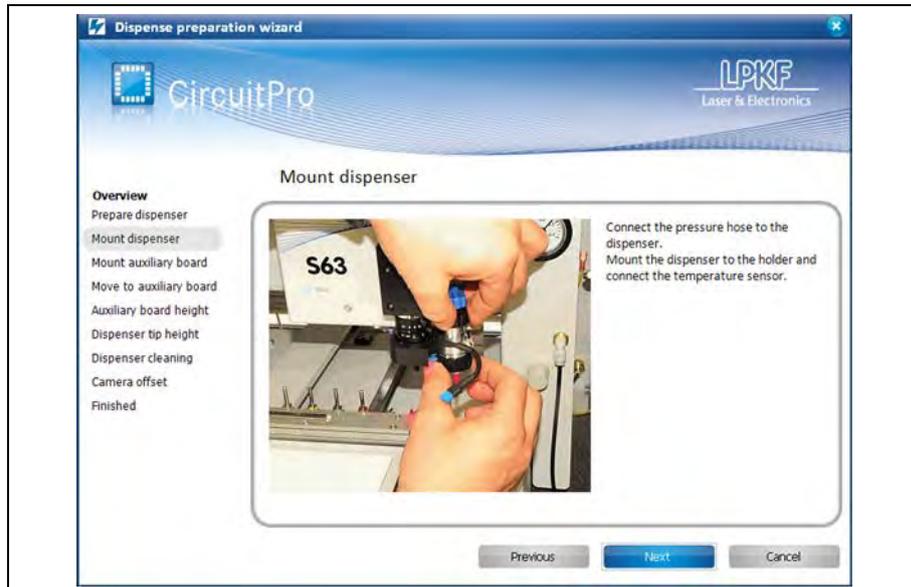
Note

If there is a tool in the clamp, it will be placed back into the tool holder.

2. Prepare the solder paste as described in the wizard.

3. Assemble the dispenser as described in the wizard:

Fig. 33:
Assembling the
dispenser



4. Click on [Next].

■ Mounting the auxiliary board

1. Use a scrap piece of copper material that is at least 50 x 50 millimeters.
2. Measure the overall thickness with a set of calipers and note this number.
3. Enter the thickness of the material in the thickness field:

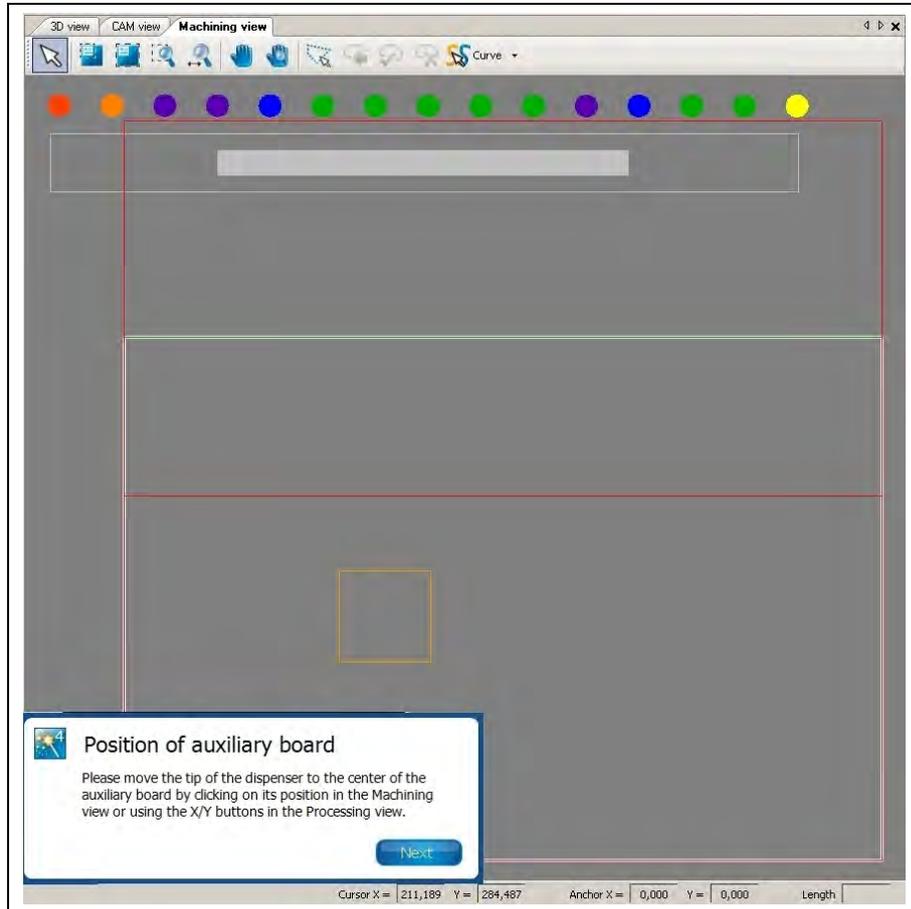
Fig. 34: Enter the
thickness



4. Click on [Next].
 5. Close the machine cover.
 6. Click on [Next], to start the positioning procedure.
- ◆ The auxiliary board was mounted.

- Setting the material height
 1. Move the wizard screen off to the side.
 2. Move the dispenser head to the center of your scrap piece by clicking on the working area with your mouse until the dispense needle is centered on the material:

Fig. 35: Position of auxiliary board

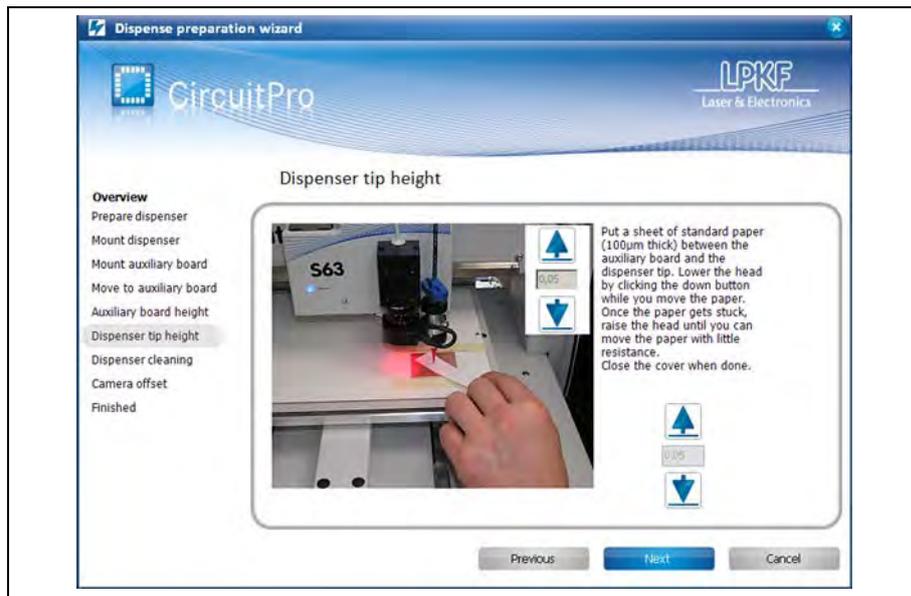


The machine will now search for the focus height of the material. It will repeat this 5 times, in a dice pattern.

Note

3. Follow the instructions displayed in the wizard:

Fig. 36: Head touching the paper



4. Take off the sheet of paper.
5. Click on [Next].
- ◆ The material height was set.

■ Cleaning the dispenser needle (or prepare needle)

1. Click on the dispense icon in the wizard:

Fig. 37: Dispense icon

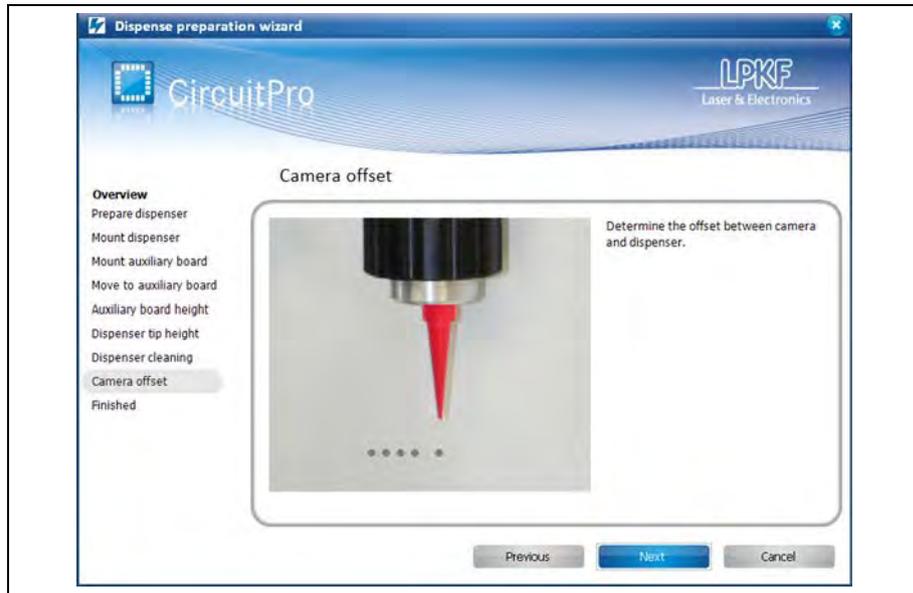


2. Repeat step 1 until a single stream of paste comes out of the needle.
3. Clean off any paste that has been dispensed.
4. Click on [Next].
- ◆ The dispenser needle was cleaned.

■ Calculating the dispenser offset

In this step, the machine is calculating the dispenser offset:

Fig. 38: Camera offset



1. Click on [Next].

➔ The machine will dispense 5 dots. Dot number 5 will be slightly offset from the first four. Then the camera will move over the dots and will zoom in.



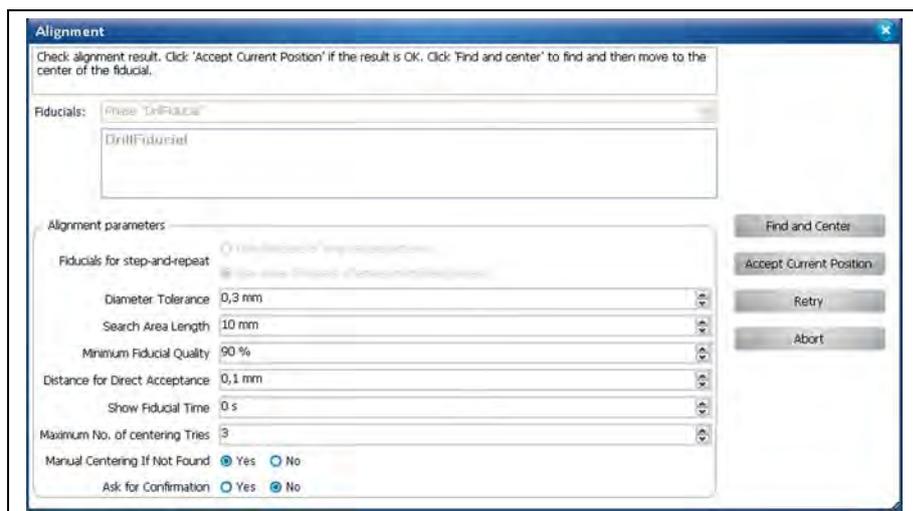
Make sure, that the camera zooms in the last of the five dispensed dots!

Note

Otherwise the offset is not calculated correctly.

➔ The following dialog is displayed:

Fig. 39:
Alignment



Read in the dot manually if centering failed

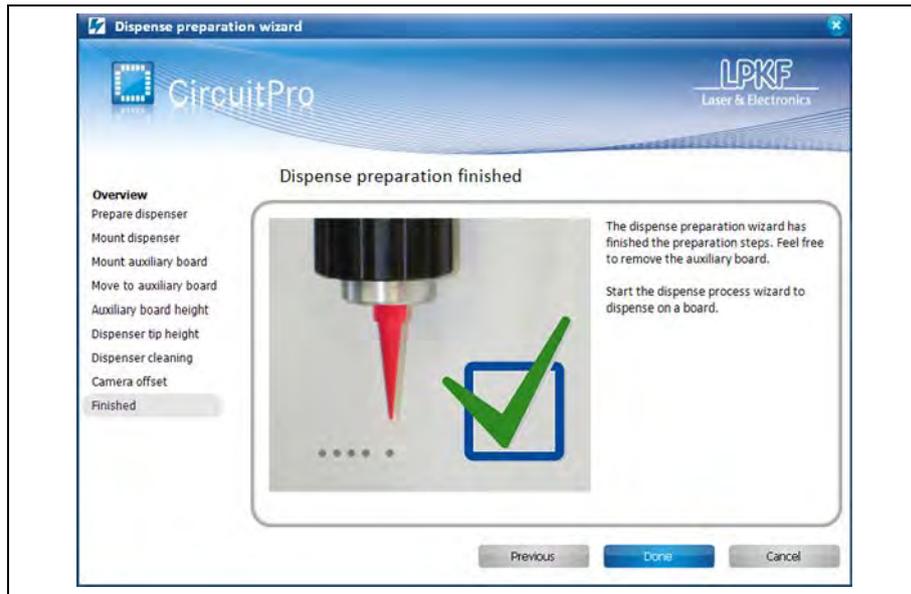
1. If the camera did not find the last dot, move the camera head manually to the position of the dot by using the X-/Y-arrow-icons

- Note
- in the pane „Processing“.
 2. If the dot is displayed centered in the pane “Camera“ click on [Find and Center] in the dialog “Alignment”.
→ The camera zooms in the dot.

2. Click on [Accept Current Position].

➔ The view changes as follows:

Fig. 40: Dispense
preparation
finished



3. Click on [Done].
- ◆ The dispenser offset was calculated.

The dispense preparation is finished now.

2.7 Starting dispensing

In this chapter you will learn how to dispense the solder paste by using the “Dispense process wizard”.

You have to perform following steps:

- i. Mounting the circuit board
- ii. Setting the fiducials (existing holes on the board)
- iii. Cleaning the needle
- iv. Dispensing the solder paste

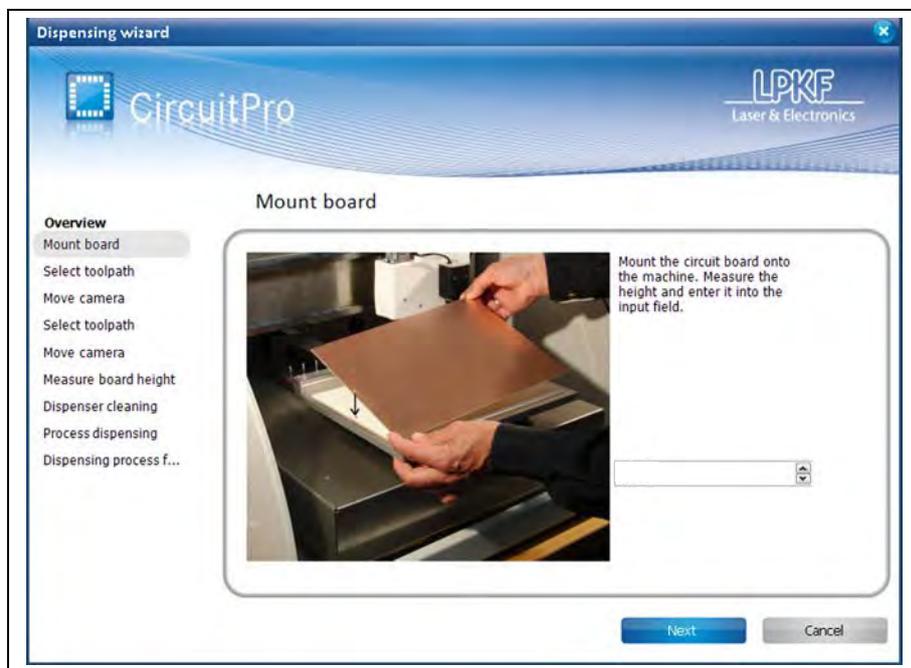
1. Click on Wizards > Dispense process wizard...

Fig. 41: Wizards > Dispense process wizard



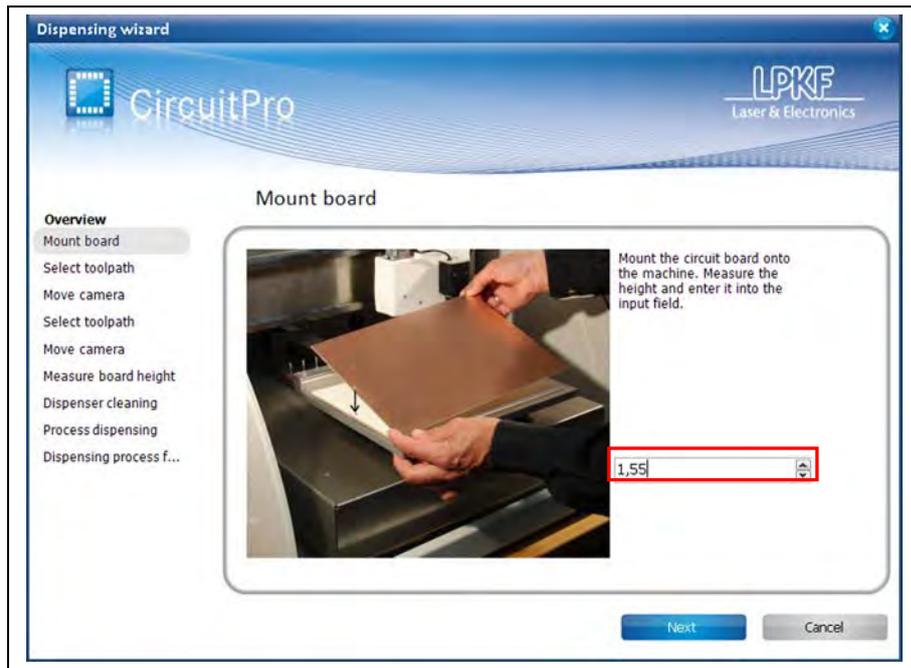
- ➔ The following dialog is displayed:

Fig. 42: Mount board



- Mounting the circuit board
 1. Measure the thickness of your current board with a set of calipers and note this number.
 2. Mount the material onto the machine with the top side of the board of the board facing up.
 3. Fasten the circuit board with adhesive tape.
 4. Enter the board thickness into the thickness field:

Fig. 43: Enter the thickness



5. Click on [Next].
- ◆ The circuit board was mounted.

■ Reading in the drill holes

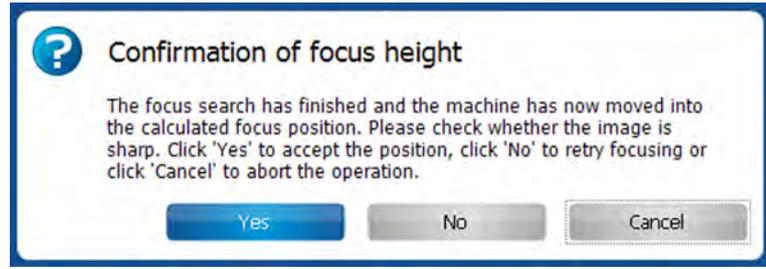
In this step two existing drill holes will be read in to align the circuit board.



Note

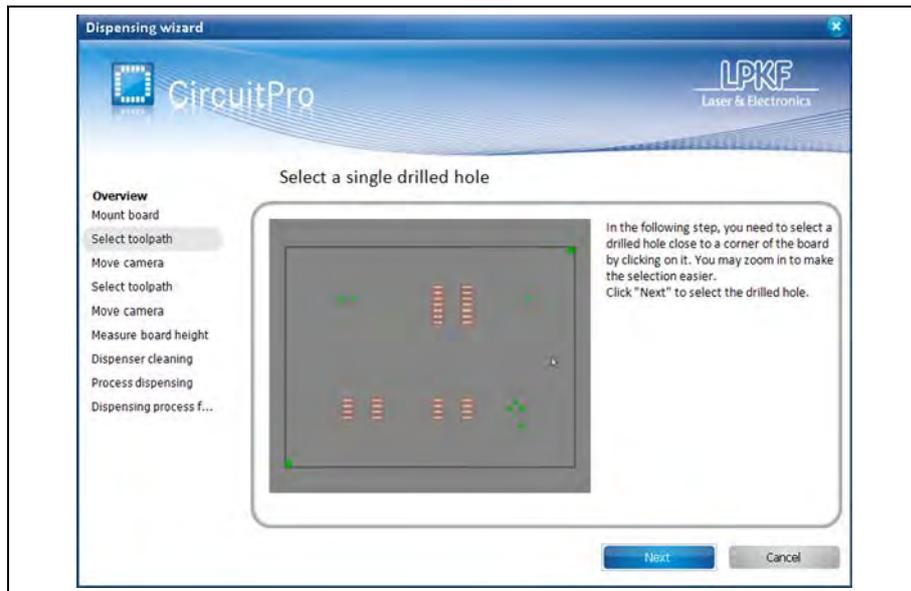
If the fiducial search is performed for the first time (after having started CircuitPro) the camera is performing an autofocus five times.

Afterwards the following message is displayed which prompts you to confirm the focus height:



1. Follow the instructions displayed in the wizard:

Fig. 44: Select drill hole



2. Click on [Next].

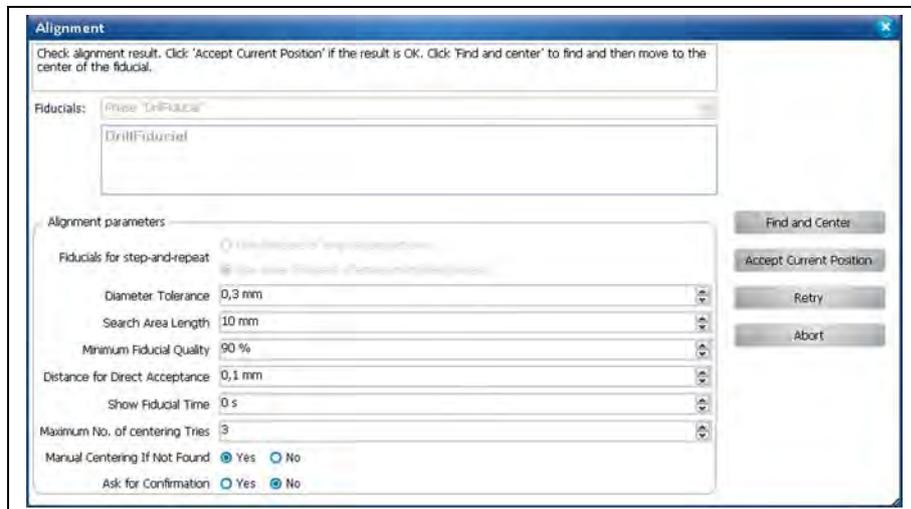
➔ Following message is displayed:

Fig. 45: Message "Select drill hole"



3. Left click on one of the drill holes near a corner of the board.
 - ➔ The camera moves to the drill hole and reads in the position.
 - ➔ The “Alignment” dialog is displayed:

Fig. 46:
Alignment



Note

Read in the drill hole manually if centering failed

1. If the camera did not find the drill hole, move the camera head manually to the position of the drill hole by using the X-/Y-arrow-icons in the pane „Processing“.
2. If the drill hole is displayed centered in the pane “Camera“ click on [Find and Center] in the dialog “Alignment”.
 - ➔ The camera zooms in the drill hole

4. Click on [Accept Current Position].
5. Repeat the steps 1-3 for the second drill hole in the opposite corner.
 - ◆ The drill holes for aligning the circuit board have been read in.

In the next step the machine will find the focus height on the material, similar to what it did during the “dispenser offset” step. Afterwards the wizard continues with step “Dispenser cleaning”.

■ Cleaning the needle

1. Click on the dispense icon in the wizard screen:

Fig. 47: Dispense icon



2. Repeat step 1 until a single stream of paste comes out of the needle.
3. Clean off any paste that has been dispensed.
4. Click on [Next].

◆ The dispenser needle was cleaned.

- Dispensing solder paste

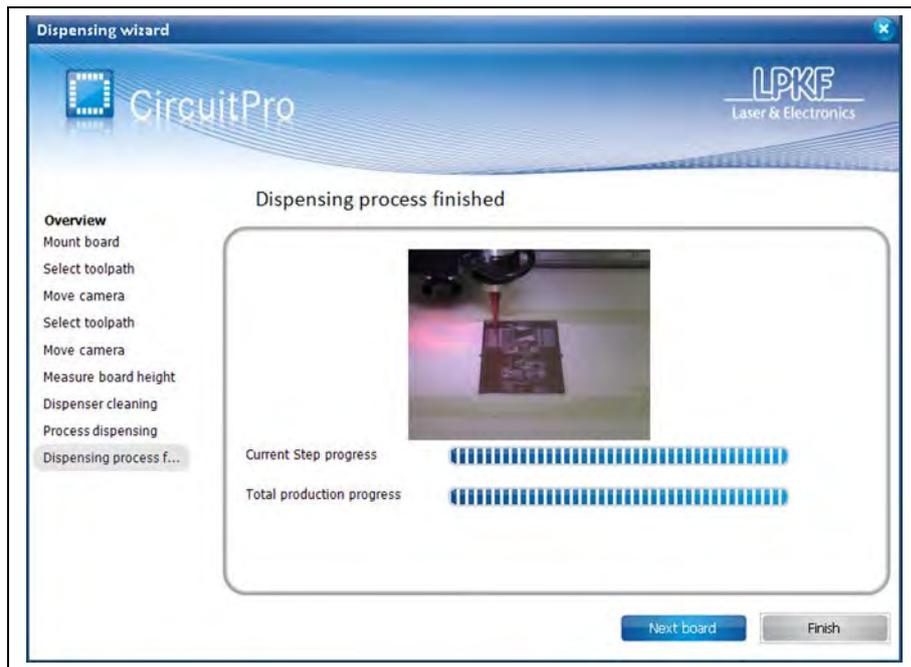


The machine will now dispense solder paste on your pads.

Note

1. When dispensing is completed, you can decide if you want to apply solder paste to second board. In this case, click on [Next board]:

Fig. 48: Process dispensing



2. After all boards are complete, click on [Finish].
 3. Remove the paste dispenser from the machine.
- ◆ The solder paste was dispensed.



Close the solder paste cartridge by using the cap. Then store the cartridge in the refrigerator.

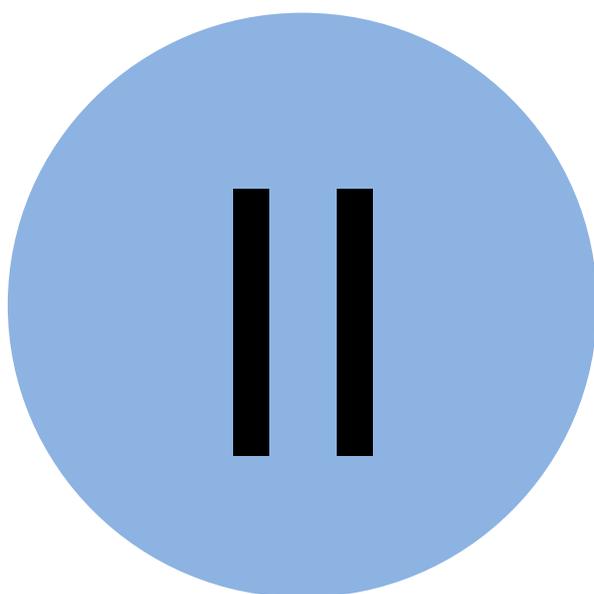
Note



For further process steps we recommend the systems LPKF ProtoPlace for placing components and LPKF ProtoFlow for lead-free soldering.

Tip

How to do



Creating a 3D part

1 Creating a 3D part

In this section you will learn how to create a 3D part of 5 mm-thick aluminum in CircuitPro. Pockets are placed on the top side. Furthermore there will be defined drill holes.



Note

To create a 3D part you need a ProtoMat S63 or S103!



Note

This exercise is based on a base material with the dimensions 100 mm x 100 mm x 5 mm.

You are able to use other base materials, too. Make sure that the dimensions of the used material has at least the dimensions 100 mm x 100 mm and a thickness of 5 mm +/- 1 mm.

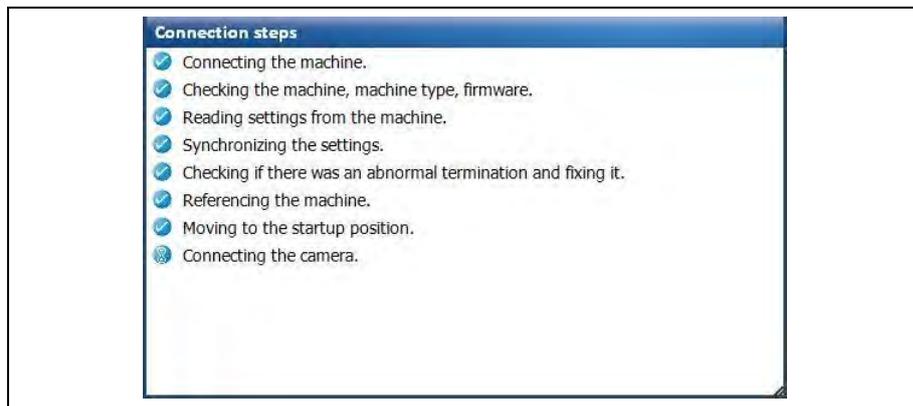
Therefore you have to perform following steps:

- i. Starting the machine and CircuitPro
- ii. Selecting the template and creating the new document
- iii. Creating the layout
- iv. Creating 2.5D objects
- v. Creating toolpaths
- vi. Loading the tool magazine and assigning tools to holder positions
- vii. Start processing

1.1 Starting the machine and CircuitPro

- Starting the machine and CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed:

Fig. 49:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to its reference points and subsequently moves to the Pause position.

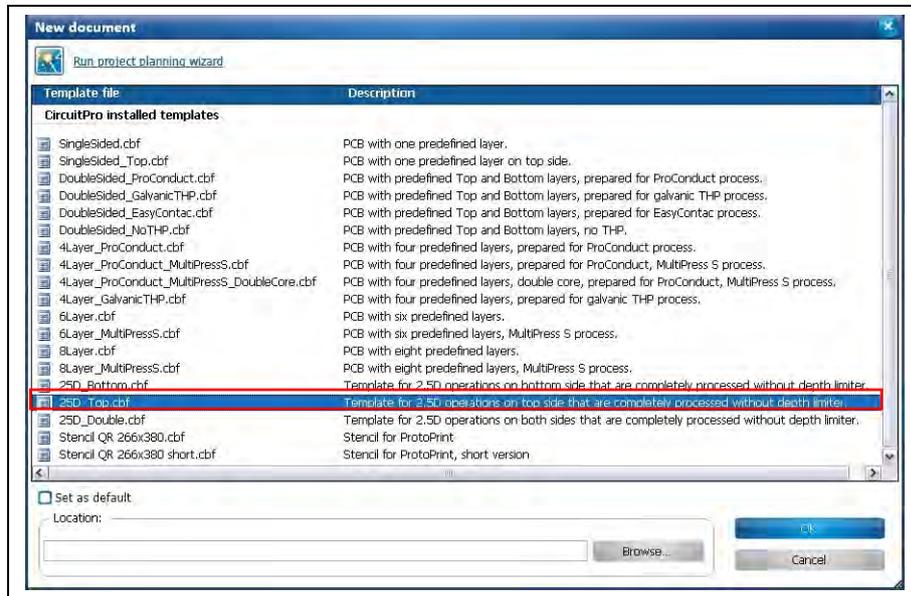
1.2 Selecting a template and creating a new document

- Selecting a template and creating a new document

1. Click on File > New...

➔ The following dialog is displayed:

Fig. 50: New document

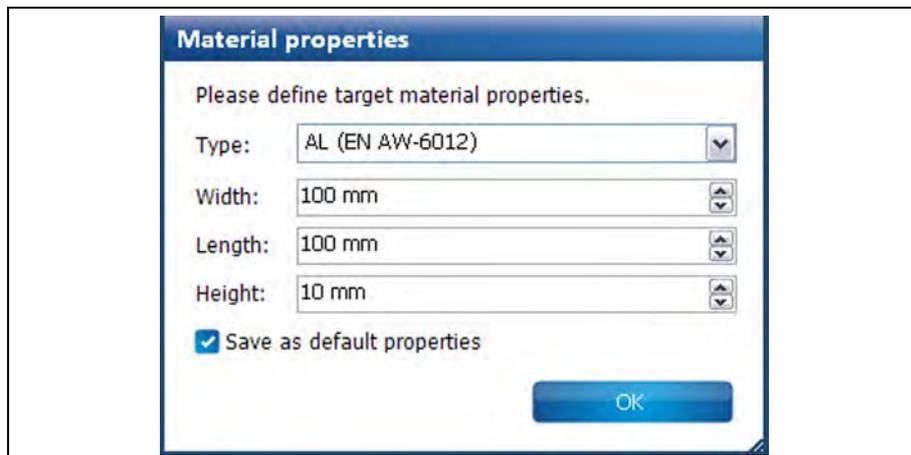


2. Select the template "25D_Top.cbf".

3. Click on [OK].

➔ The following dialog is displayed:

Fig. 51: Material properties



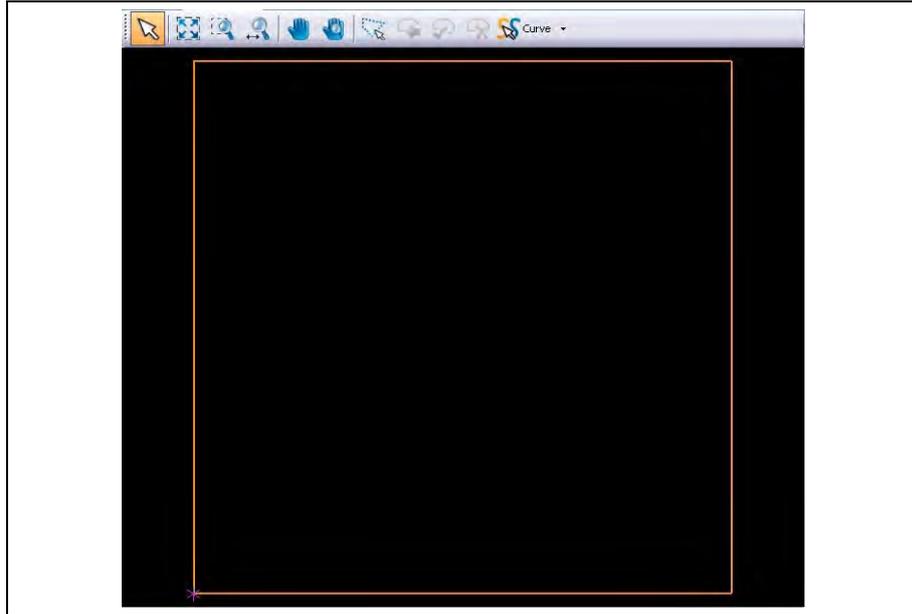
4. In the \Type\ field, select the material type "AL (EN AW-6012)" or "AL (EN AW-5083)". In this case these are two different aluminum alloys.

5. Enter the dimensions of your material in the fields \Width\, \Length\ and \Height\.

6. Click on [OK].

➔ The CAM view is changing:

Fig. 52: CAM view



7. Click on File > Save As...
8. Enter a file name for the new file.
9. Click on [Save].

◆ The new document is created.



Note

You are able to change the material properties such as the dimensions of the base material. Therefore you have to right-click in the CAM view and click in the context menu on "Material properties...".

1.3 Creating the layout

In this section you will perform following steps:

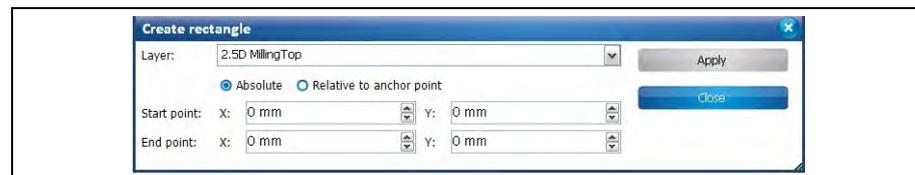
- i. Creating rectangles for the pockets
- ii. Creating circles for drill holes
- iii. Multiplying the circle

■ Creating rectangles for the pocket

1. Click on Insert > Rectangle...

➔ The following dialog is displayed:

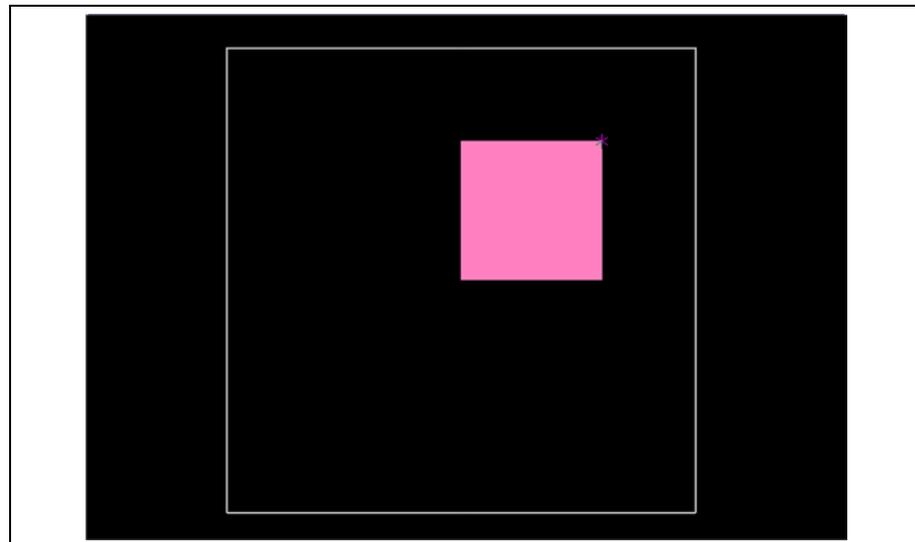
Fig. 53: Create rectangle



2. Select the layer to create the rectangle on, e.g. "2.5D MillingTop".
3. Select the option "Absolute" (start and end points are entered from the zero point).
4. Enter "50" in the fields \Start point X\ and \Start point Y\.
5. Enter "80" in the fields \End point X\ and \End point Y\.
6. Click on [Apply].

➔ The drawn rectangle is shown in the CAM view

Fig. 54: Rectangle in CAM view

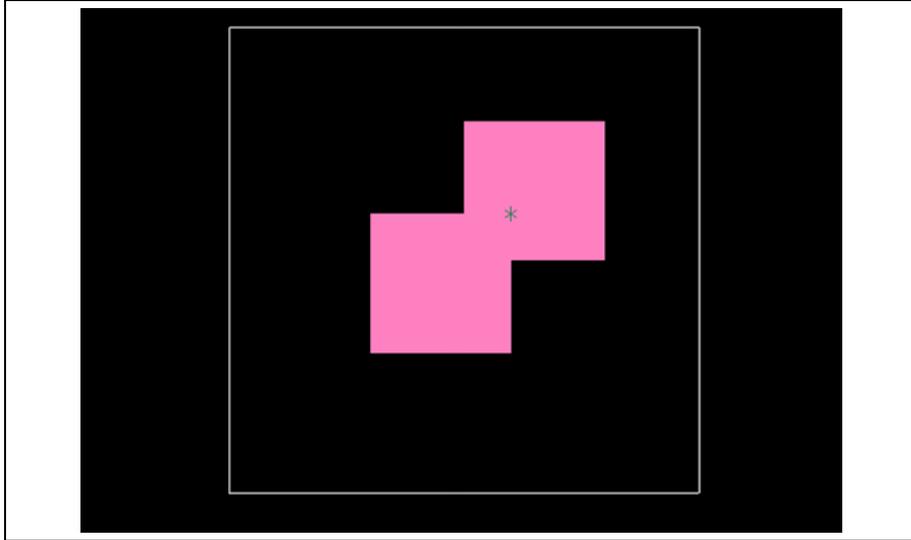


7. For creating another rectangle, enter the following values in the corresponding fields:
 - Start point X: 30
 - Start point Y: 30
 - Endpoint X: 60
 - Endpoint Y: 60

8. Click on [Apply].

➔ The second rectangle is shown in the CAM view:

Fig. 55: Second rectangle



9. Click on [Close].

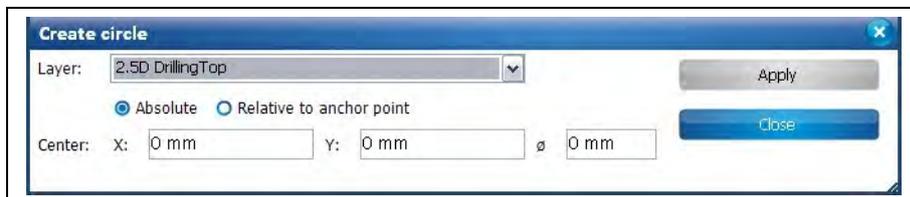
◆ The rectangles for the pockets are created.

■ Creating circles for drill holes

1. Click on Insert > Circle.

➔ The following dialog is shown

Fig. 56: Create circle



2. Select the layer „2.5D DrillingTop“ to create your circle.

3. Select the option “Absolute“ (the circle’s center is entered from the zero point)

4. Enter “15” in the fields \Center X\ and \Center Y\.

5. Enter “2” in the field \∅\.



Note

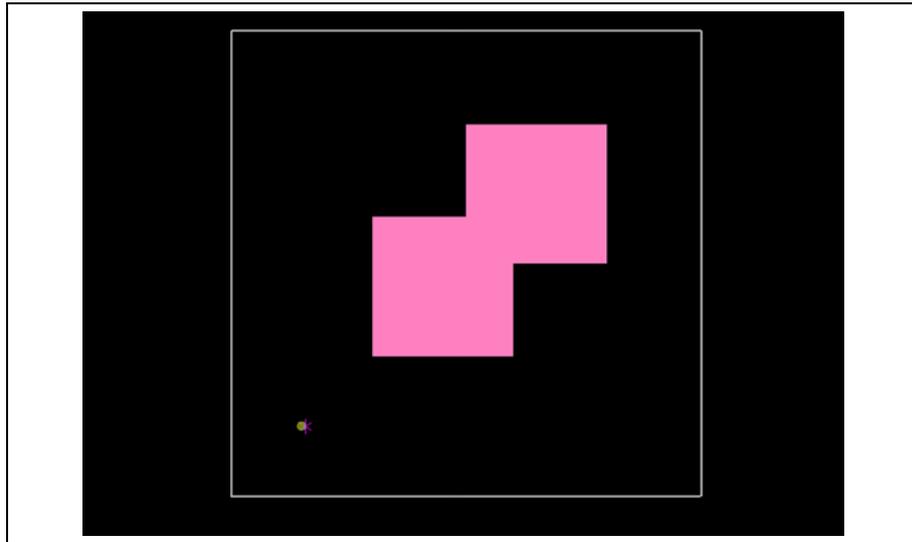
If you do not have a Spiral Drill 2 mm, you can enter the corresponding diameter of our drill tool used.

Therefore enter the diameter of your drill tool in the field \∅\.

6. Click on [Apply].

➔ The circle is shown in the CAM view:

Fig. 57: Circle in CAM view



7. Click on [Close].
- ➔ The dialog is closed.
- ◆ The circle was created.

■ Multiplying the circle

1. Highlight the created circle in the CAM view.
- ➔ The circle is highlighted in grey.
2. Press your right mouse button.
- ➔ The context menu is shown.
3. Click on "Step and Repeat...".

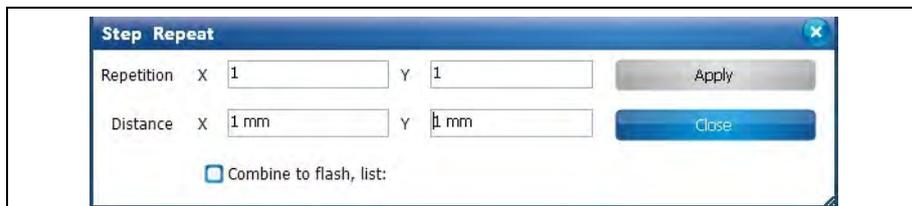


You can also click on Modify > Step and Repeat...

Tip

- ➔ The following dialog is shown:

Fig. 58: Step Repeat



4. Enter "5" in the field \Repetition X\.
5. Enter "5" in the field \Distance X\.



Note

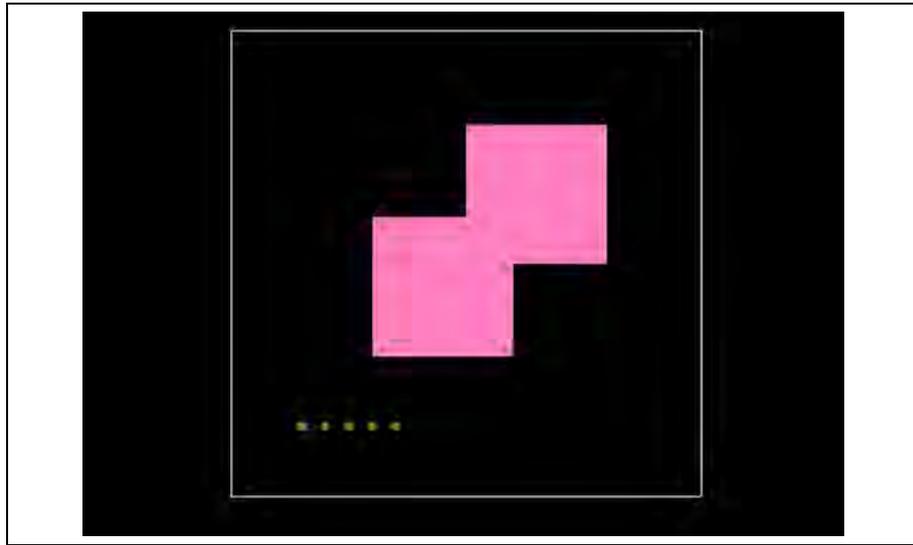
The dimension of the selected object is automatically displayed in the fields \Distance X\ and \Distance Y\.

The desired distance between the objects must be added to this value.

6. Click on [Apply].

➔ The drawn circle is multiplied in X direction.

Fig. 59:
Multiplying the
circle in X
direction



7. Click on [Close].

➔ The dialog is closed.

◆ The drawn circle was multiplied.



Note

Assign layer

The layer which contain the drawn objects can be changed manually, if necessary.

1. Highlight the objects.
2. Press your right mouse button.
3. Click in the context menu on "Assign objects to layer".
→ The layer's list is shown.
4. Click on the desired layer on which you want to place the objects.
→ The objects were moved to the desired layer.

1.4 Creating 2.5D objects

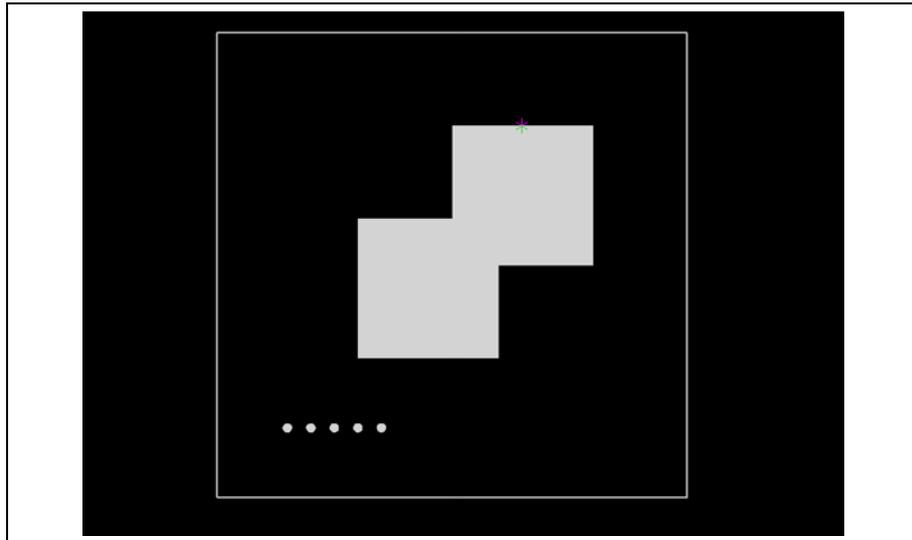
The opened file only contains 2D objects yet.

The drawn 2D objects on the layers “Milling Top” and “Drilling Top” must be converted to 2.5D objects with a defined depth.

■ Creating 2.5D objects

1. Select all objects in the CAM view:

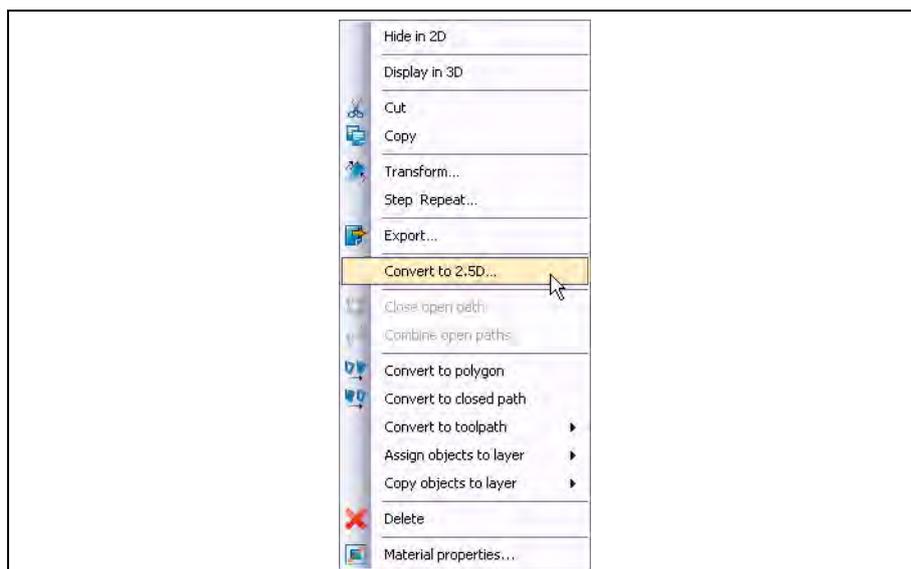
Fig. 60: Select all objects



2. Press your right mouse button.

➔ The context menu is displayed.

Fig. 61: Context menu



3. Click on “Convert to 2.5D...”.

➔ The following dialog is displayed:

Fig. 62: Convert into 2.5D



Note

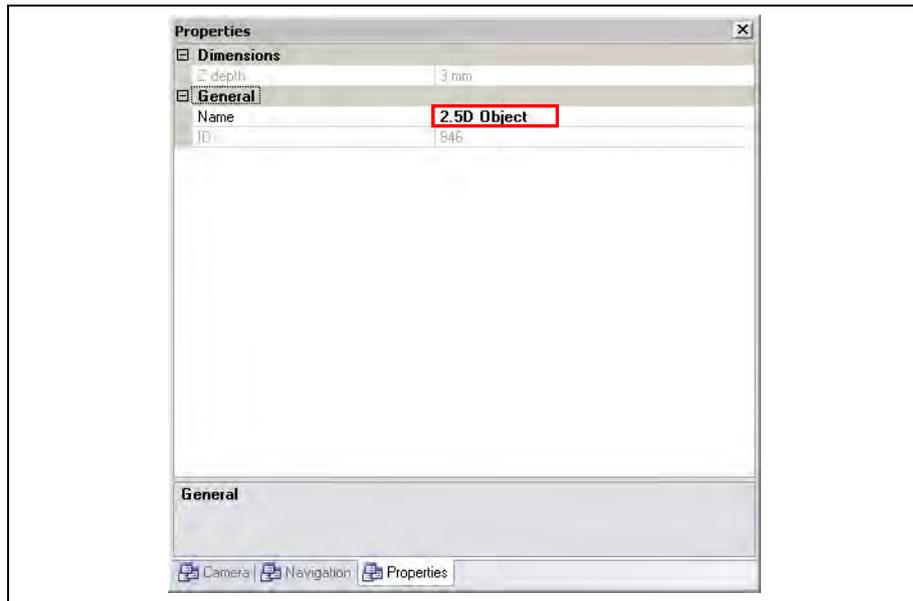
The value in field \Destination layer\ is preselected with “Same as source layer”. The objects will be converted into 2.5D objects and remain on their origin layer.

4. Enter “3” in field \Z value\.

5. Click on [Convert].

➔ The selected objects were converted to 2.5D objects. You can regard the object’s properties by highlighting them and clicking in the pane „Properties“:

Fig. 63: Properties of the 2.5D object



6. Click on [Close].

➔ The dialog is closed.

◆ The objects were converted in 2.5D objects.



Note

Changing the object's Z value

You are able to change the object's Z value afterwards:

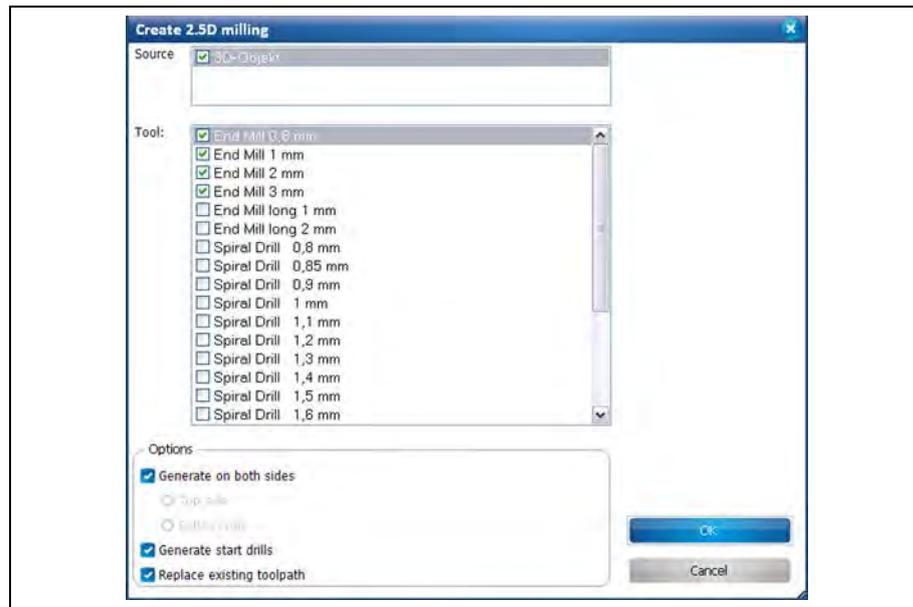
1. Highlight the desired object.
 2. Press your right mouse button.
→ The context menu is shown.
 3. Click on "Change Z value...".
→ The dialog „Change Z value“ is shown.
 4. Enter a new value in the field \New Z value\.
 5. Click on [Apply].
→ The Z value was changed.
 6. Click on [Close].
→ The dialog is closed.
-

1.5 Creating toolpaths

Before the machine can process the material, the 2.5D toolpaths have to be created.

- Creating 2.5D toolpaths
- 7. Click on Toolpath > 2.5D milling...
- ➔ The following dialog is displayed:

Fig. 64: Create 2.5D milling



- 8. Activate the 3D object by checkmark in field \Source\.
- 9. Activate the needed tools for the job by checkmark.



Note

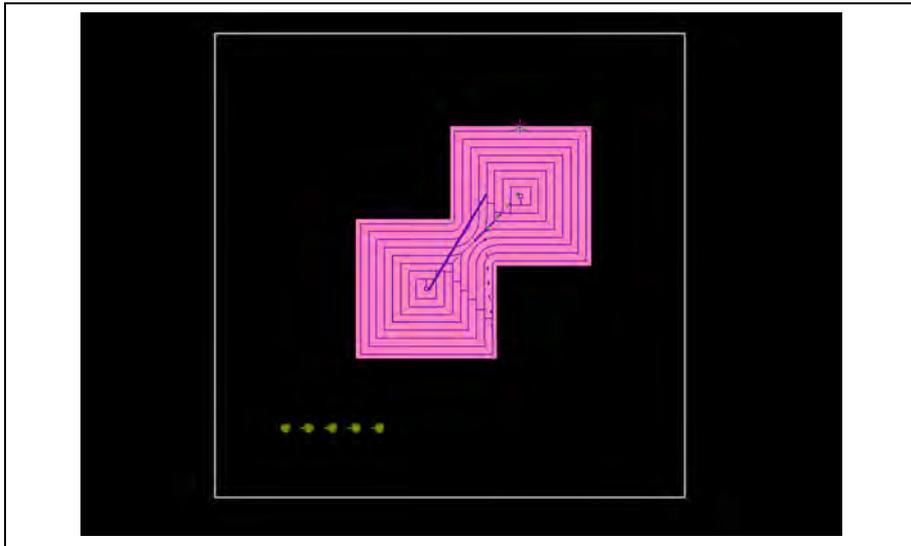
Make sure, that you activate the drill tools with the appropriate diameter corresponding to the size of your drill holes. For example the Spiral Drill 1 mm if you want to create drill holes with a diameter of 1 mm.

For the milling of the pockets you need End Mill tools. Activate the available tools in the list.

You also need drill tools for generating start drills if you have not disabled this option. If the pocket should be milled with an End Mill 2 mm, you need to create the start drills with a Spiral Drill 2 mm.

- 10. Click on [OK] to create the toolpaths.
- ➔ The CAM view changes as follows:

Fig. 65: CAM view



11. Click on the tab "3D view".

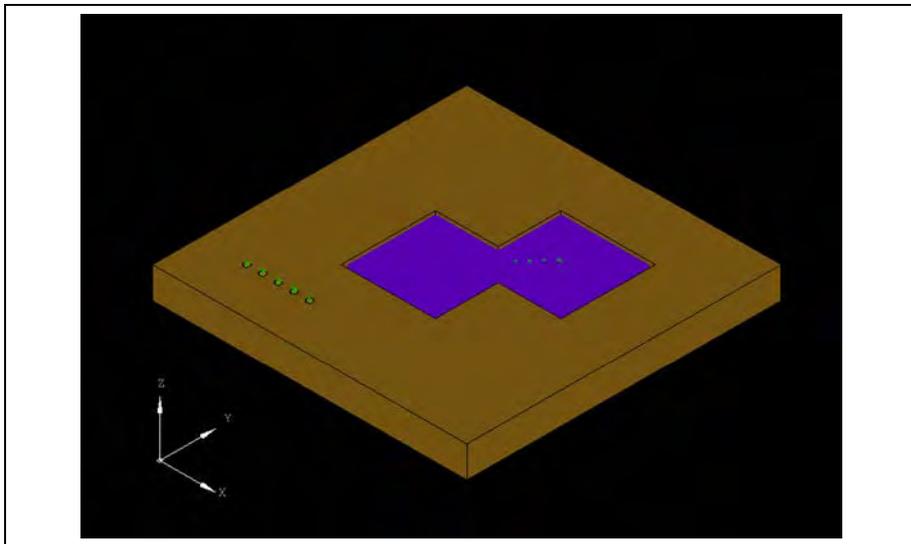


If the tab „3D view“ is not displayed, you can activate it by clicking on the menu item View > 3D Wiew.

Note

➔ The 3D tab displays the 3D part:

Fig. 66: 3D view



Tip

You can rotate the object in the 3D view by activating this icon in the toolbar.



◆ The 2.5D toolpaths were created.

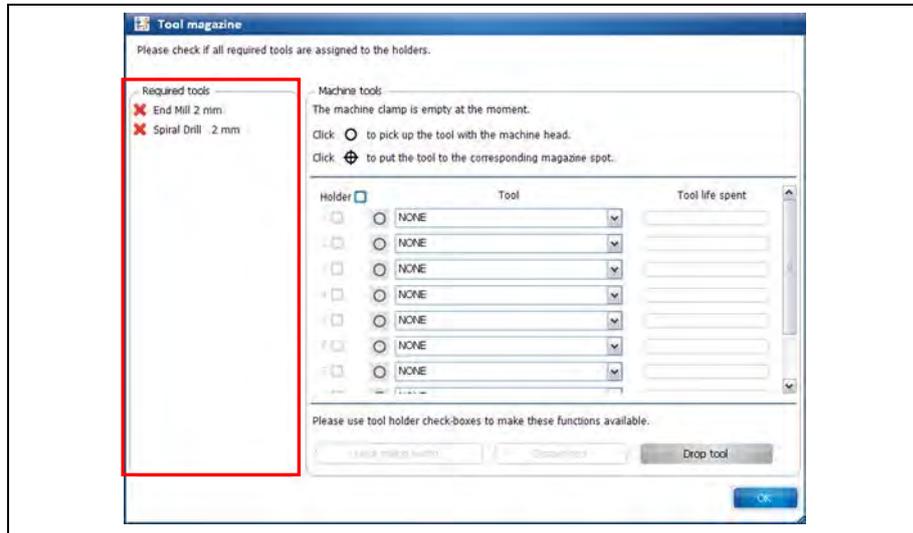
1.6 Loading the tool magazine and assigning tools to holder positions

- Loading the tool magazine and assigning the tools to holder positions

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

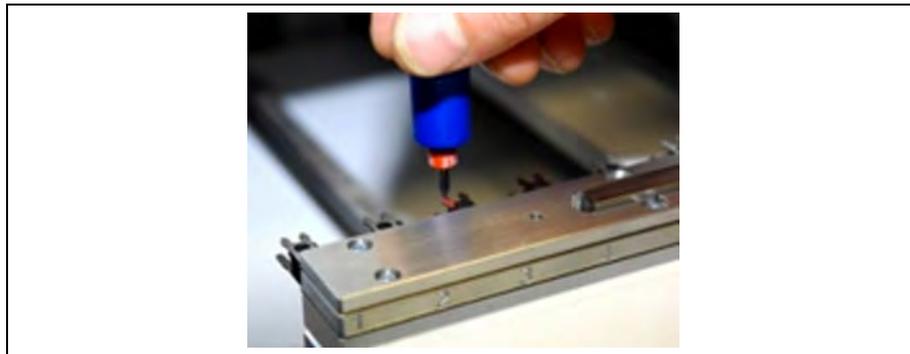
Fig. 67: Tool magazine



➔ The tools required for the job are displayed in section “Required tools”. Tools that are missing in the tool magazine are marked by a red “X”. Required tools which are already inserted in the tool magazine are marked by a green checkmark.

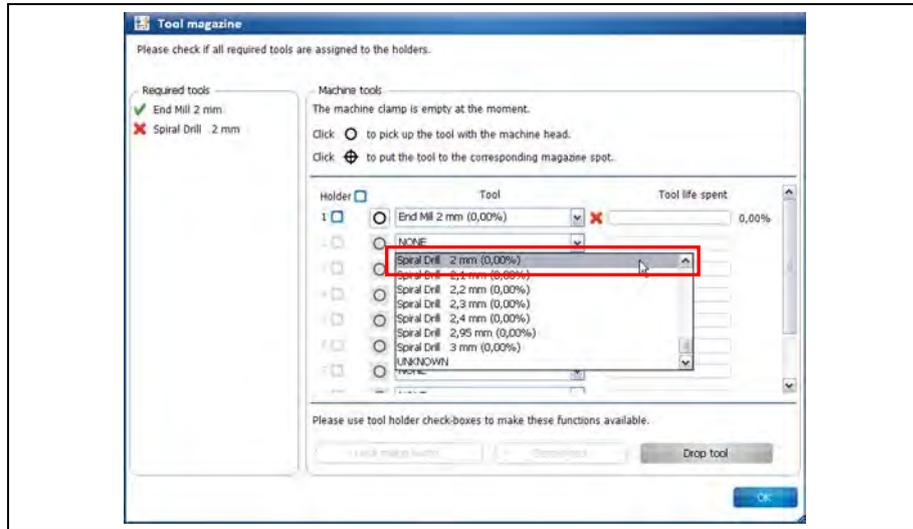
2. Insert the required tools into the tool magazine:

Fig. 68: Inserting the tool



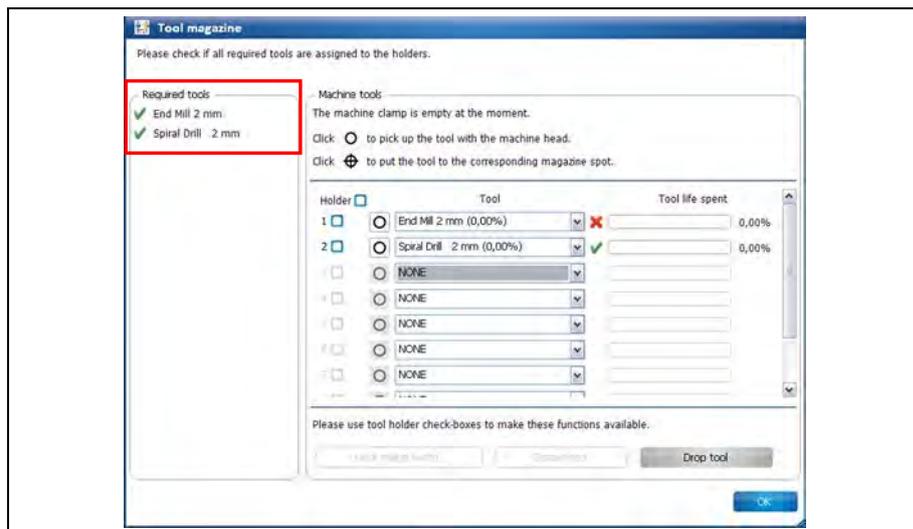
- In the dialog, assign the tools to the respective tool magazine positions used.

Fig. 69:
Assigning the tools in the dialog



- ➔ The tools which are displayed in the section “Required tools” are marked with a green checkmark:

Fig. 70: Assigned tools



- Make sure that the tool life spent is sufficient for the current job to be processed.



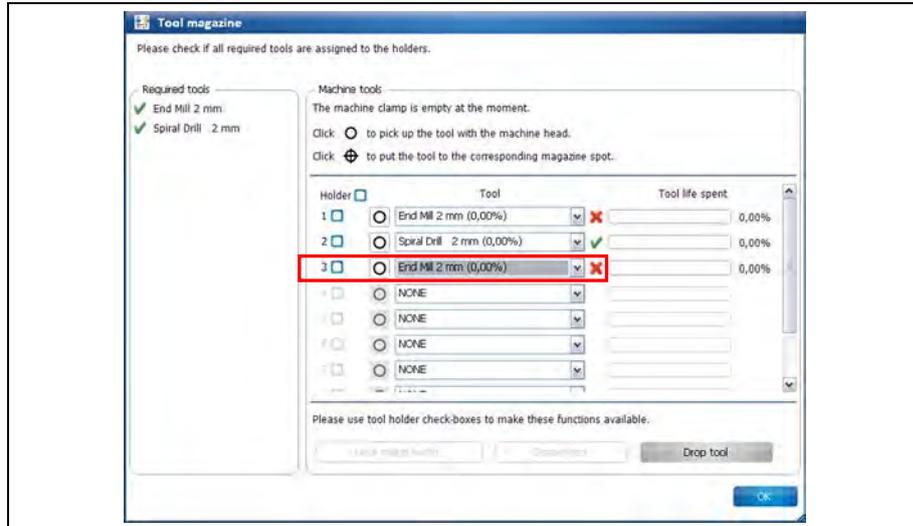
Note

If there is a red „X“ displayed beside the progress bar “Tool life spent” the endurance of the tool is insufficient for the job to be processed:

Holder	Tool	Tool life spent
1	End Mill 2 mm (0,00%)	0,00%
2	Spiral Drill 2 mm (0,00%)	0,00%

5. If necessary, insert additional tools in the tool magazine to reach the required tool endurance:

Fig. 71:
Assigning
additional tools



6. Click on [OK].
- ➔ The dialog is closed.
The tools are assigned to the corresponding tool holders:

Fig. 72: Tools in
the tool magazine

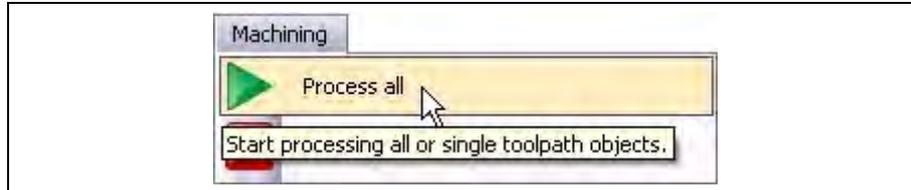


- ◆ The tools were loaded.

1.7 Starting processing

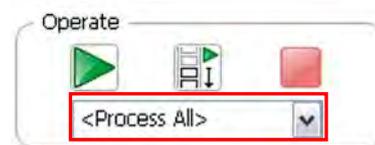
- Starting processing
 1. Click on Machining > Process all.

Fig. 73:
Machining >
Process all



Note

Make sure that <Process All> is selected in the combo box of the pane "Processing", so that all phases of the job are executed. The phases included in the current job are displayed in the section "Phases" of the pane "Toolpath".



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

After the production started, the machine will process following phases in order. The phases are displayed via prompts.

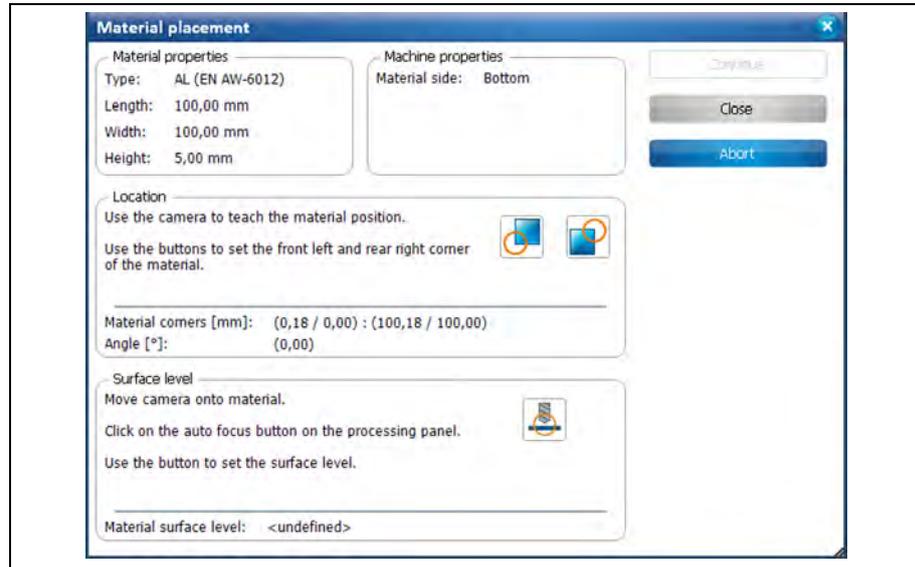
Phase "Mount Material"

1. Mount the material onto the processing area.
2. Fasten the material to the table using double-sided adhesive tape.
3. Click on [OK].

Phase “Placement Top”

➔ The following dialog is displayed:

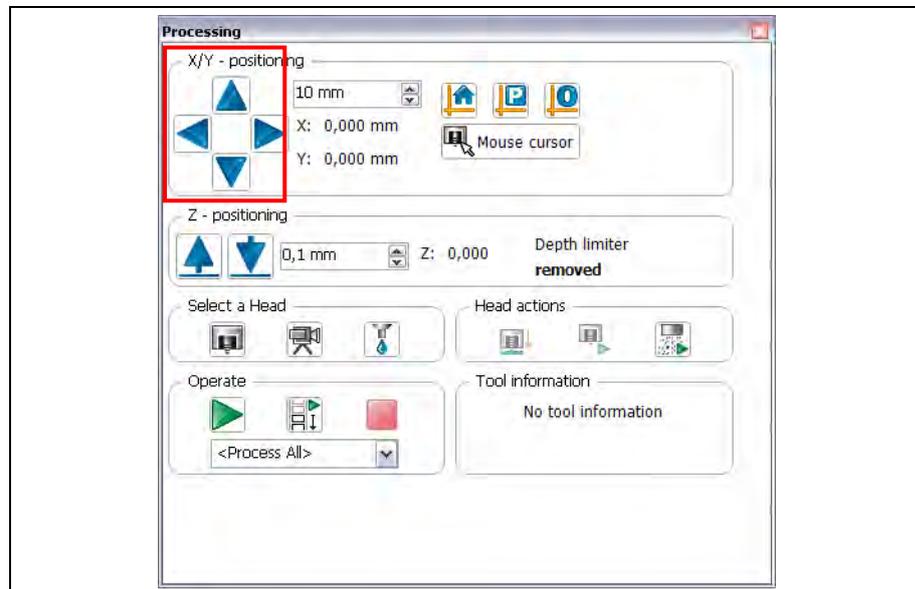
Fig. 74: Material placement



■ Defining the material corners

1. Move the camera head to the left corner of your base material:
 - a) Click on the “Processing” pane.
 - b) Use the arrow buttons in the X/Y section to move the camera head.

Fig. 75: Pane „Processing“



Note

The camera head is activated automatically. You are able to tell this by the green frame around the camera symbol in section “Select a head” of the “Processing” pane:



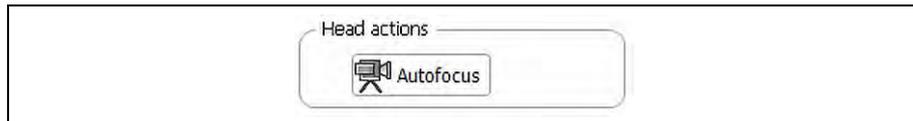


Tip

You are able to click on the corner of your material in the machining view. The camera moves automatically to this point. Use the X/Y arrow buttons to do the fine adjustment.

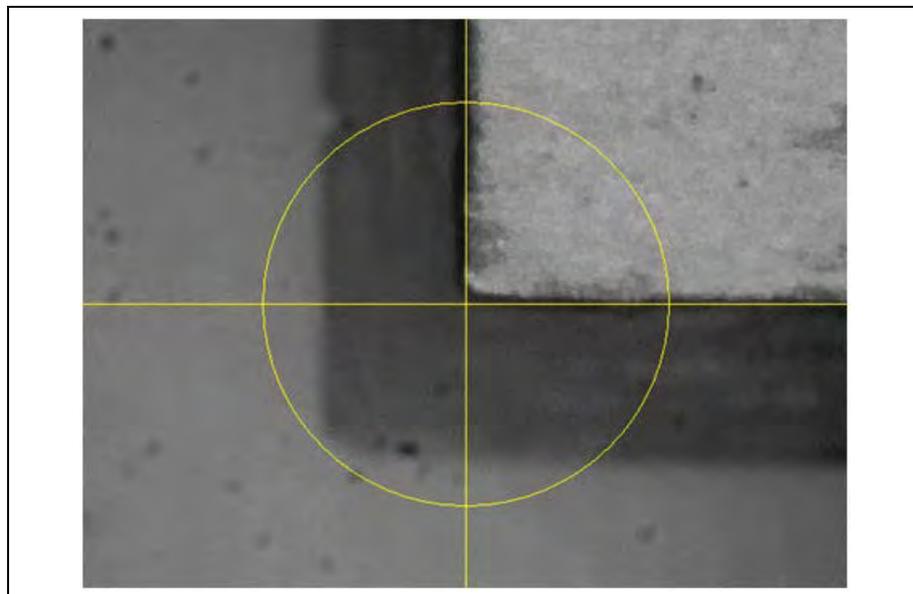
2. Use the auto focus function to align the material corners in the pane "Camera".
 - a) Click on [Autofocus] in the "Processing" pane:

Fig. 76:
Autofocus



3. Move the camera head, so that the left lower corner of your material is positioned right in the cross hair:

Fig. 77: Cross
hair of camera



Note

If the cross hair is not displayed, you are able to activate it by clicking on Camera > Overlay > Switch Crosshair State

4. Click on following icon in the dialog "Material Placement":

➔ The coordinates for the lower left corner were saved.



➔ Following message is displayed:

Fig. 78: Message
for second
position



5. Confirm the message by clicking on [Yes].
 - ➔ The camera moves automatically to the opposite corner of your material.
6. Place the cross hair right on the top of your material corner by using the X/Y arrow buttons.
7. Click on following icon in the dialog "Material Placement":
 - ➔ The coordinates for the upper right corner were saved.
 - ◆ The coordinates of the material corners were defined.

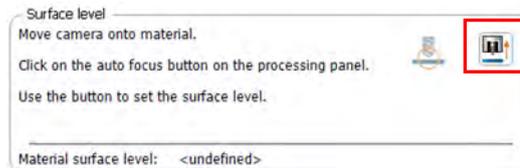


■ Defining the material surface level



Note

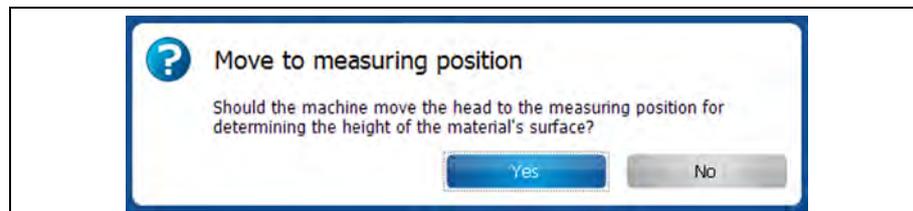
To define the material surface level the camera's Z offset must be determined first. If it is not already determined you are able to start this procedure in the dialog "Material placement" by clicking on the following icon:



For more information about teaching the Z focus offset please refer to the CircuitPro compendium.

After defining the coordinates of the material corners the following message is displayed:

Fig. 79: Move to measuring position



1. Confirm the message by clicking on [Yes].
 - ➔ The camera moves to the center of the material area and then performs an autofocus to determine the material height.
2. Check if the autofocus has been performed successfully and if the focused material surface is visible in the pane "Camera".



Note

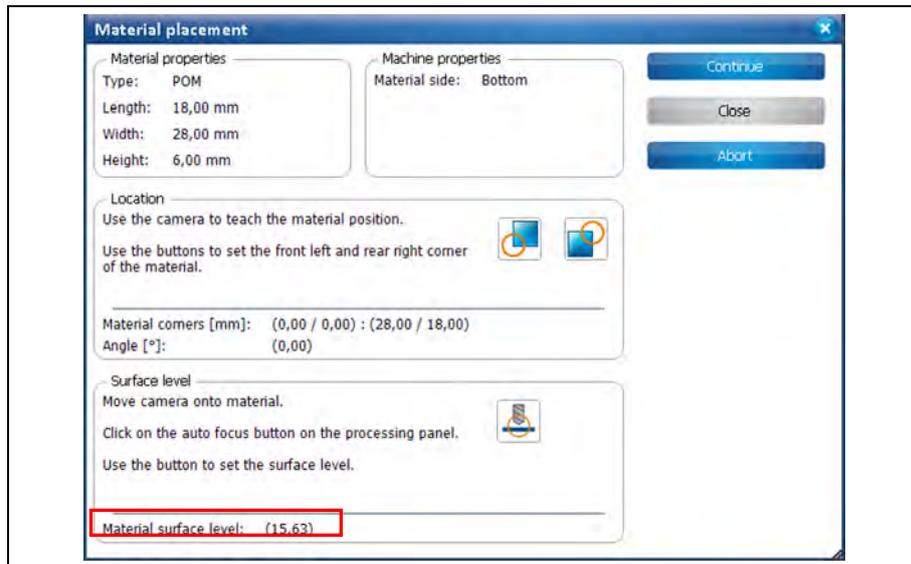
If the autofocus has not been performed successfully, perform it again by clicking on [Autofocus] in the pane "Processing".

3. Now click on the following icon for determining the material height:



➔ The material height is saved:

Fig. 80: Material surface level defined

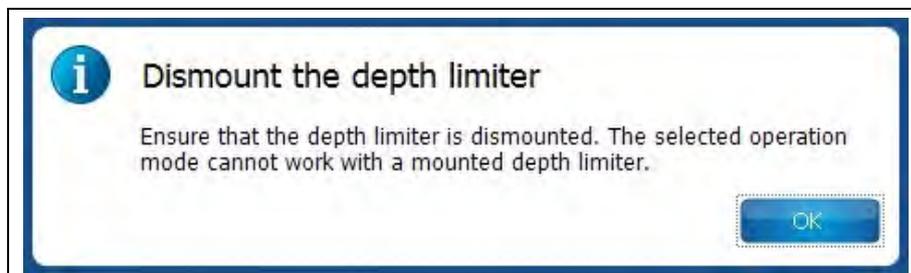


4. Click on [Continue].

◆ The material surface level was defined.

- If there is no depth limiter mounted, the phase "Processing 2.5D Top" is processed.
- If there is a depth limiter mounted, the following message is displayed that prompts you to dismount it:

Fig. 81: Dismount the depth limiter

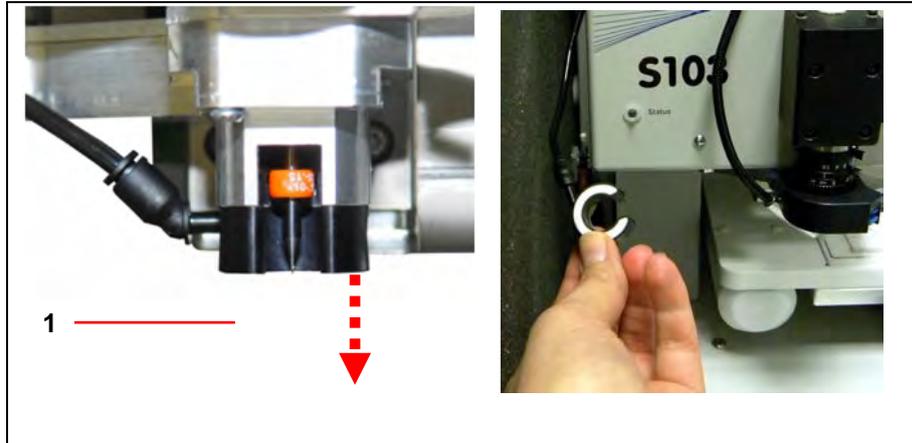


1. Confirm the message by clicking on [OK] and follow the instructions to dismount the depth limiter.

■ Dismounting the pneumatic depth limiter (S103)

1. Pull down the depth limiter:

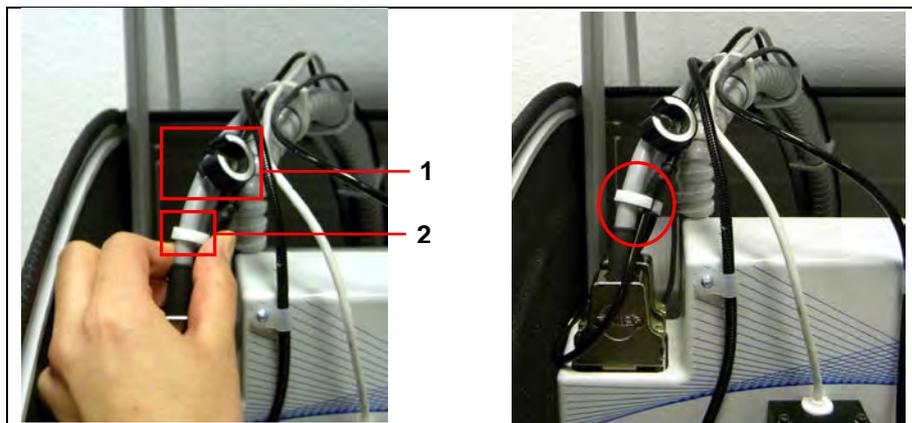
Fig. 82:
Pneumatic Depth
limiter



/1/ Pneumatic depth limiter

2. Fasten the depth limiter above the machine head in the hose clamp:

Fig. 83: Fasten
depth limiter



/1/ Pneumatic depth limiter

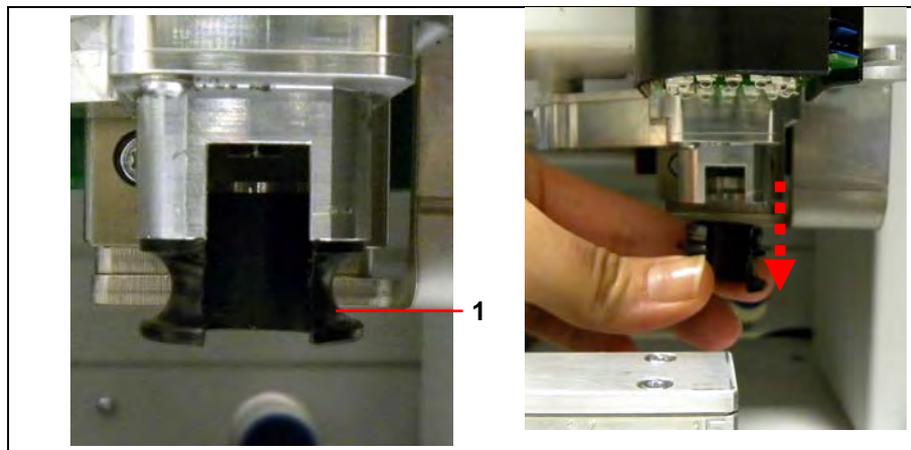
/2/ Hose clamp

◆ The pneumatic depth limiter was dismantled.

■ Dismounting the mechanical depth limiter (S63)

1. Pull down the mechanical depth limiter:

Fig. 84:
Mechanical depth
limiter



/1/ Mechanical depth limiter

2. Put the mechanical depth limiter close to the machine in order to mount it again later.
- ◆ The mechanical depth limiter was dismantled.

Phase „Processing 2.5D Top ”



If the spindle motor has not run before, a warm-up phase is started.

Note

- ➔ The required tools were picked up and the top side of the material is processed.

Phase „Board Production Finished”

- ➔ A message informs you that the board production has been finished.
- ◆ The 3D part was created.

Creating a 3D part from a STEP file

2 Creating a 3D part from a STEP file

In this section you will learn how to create a 3D part from a STEP file. The 3D part is created from a work piece with the dimensions 28 mm x 18 mm x 6 mm and the used material is POM. Different slots and drill holes are placed on the top and bottom side.



Note

To create a 3D part you need a ProtoMat S63 or S103!



Note

For this tutorial you need a base material with the dimensions 28 mm x 18 mm x 6 mm. You are able to work with other base materials. In this case make sure that your material has at least the dimensions 28 mm x 18 mm x 6 mm and a thickness of 6 mm.

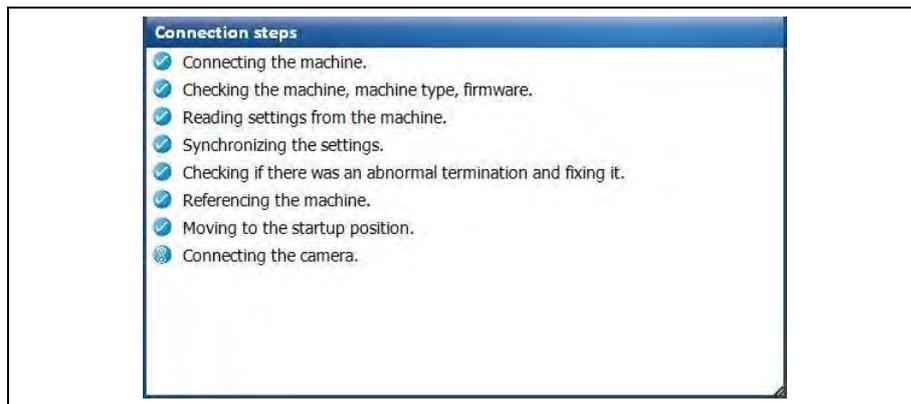
To create the 3D part you have to perform the following steps:

- i. Starting the machine and CircuitPro
- ii. Selecting the template and creating the new document
- iii. Importing the data
- iv. Creating 2.5D toolpaths
- v. Inserting fiducials
- vi. Creating toolpaths for the fiducials
- vii. Loading the tool magazine and assigning tools to holder positions
- viii. Start processing

2.1 Starting the machine and CircuitPro

- Starting the machine and CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed:

Fig. 85:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to its reference points and subsequently moves to the Pause position.

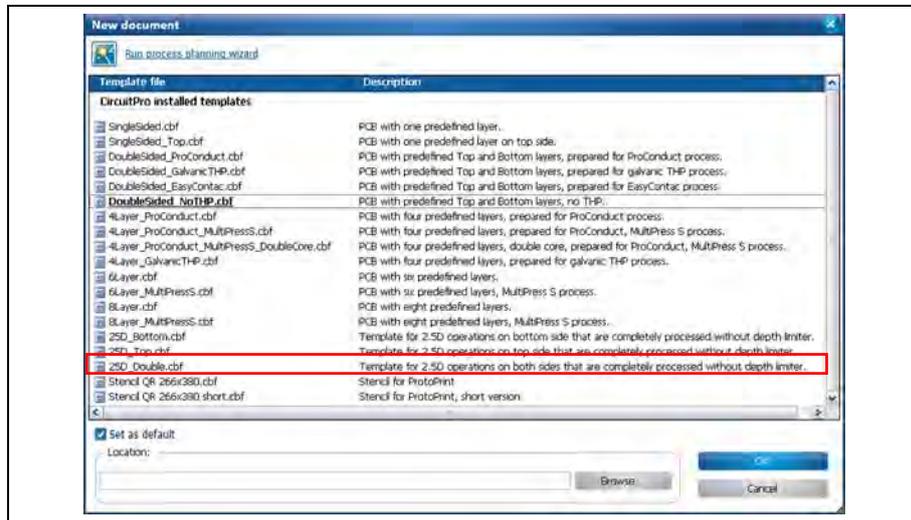
2.2 Selecting a template and creating a new document

■ Selecting a template and creating a new document

1. Click on File > New...

➔ The following dialog is displayed:

Fig. 86: New document

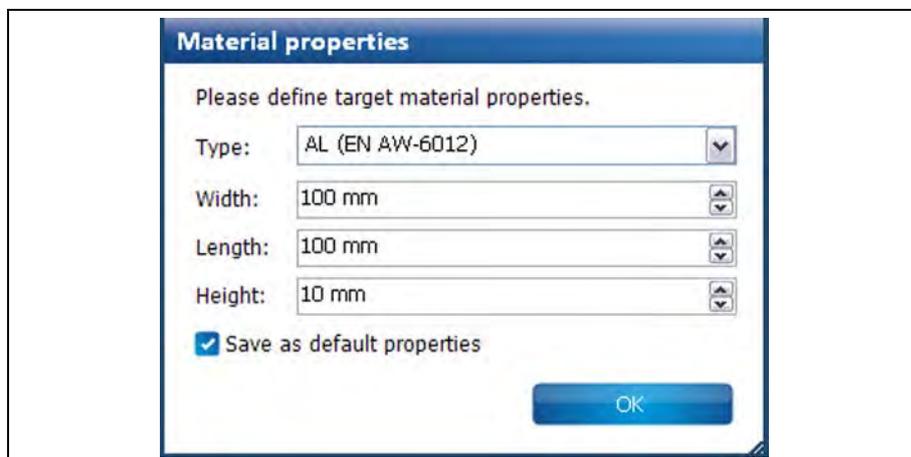


2. Select the template "25D_Double.cbf".

3. Click on [OK].

➔ The following dialog is displayed:

Fig. 87: Material properties



4. Enter the material type (POM).

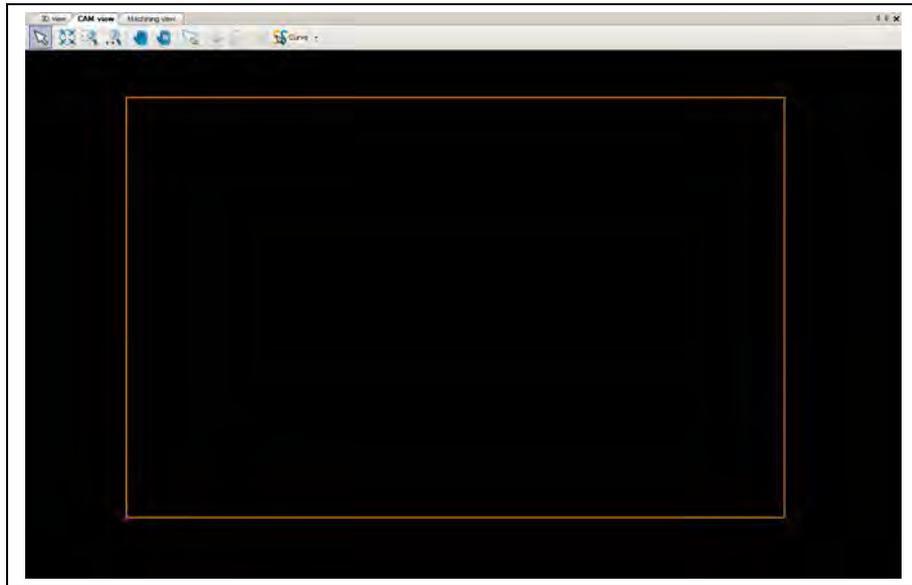
5. Adjust the values in the fields \Width\ and \Length\ according to your base material used.

6. Enter "6 mm" in the field \Height\.

7. Click on [OK].

➔ The CAM view is changing:

Fig. 88: CAM view



8. Click on File > Save As...
9. Enter a file name for the new file.
10. Click on [Save].

◆ The new document is created.



Note

You are able to change the material properties such as the dimensions of the base material. Therefore you have to right-click in the CAM view and click in the context menu on "Material properties...".

2.3 Importing data



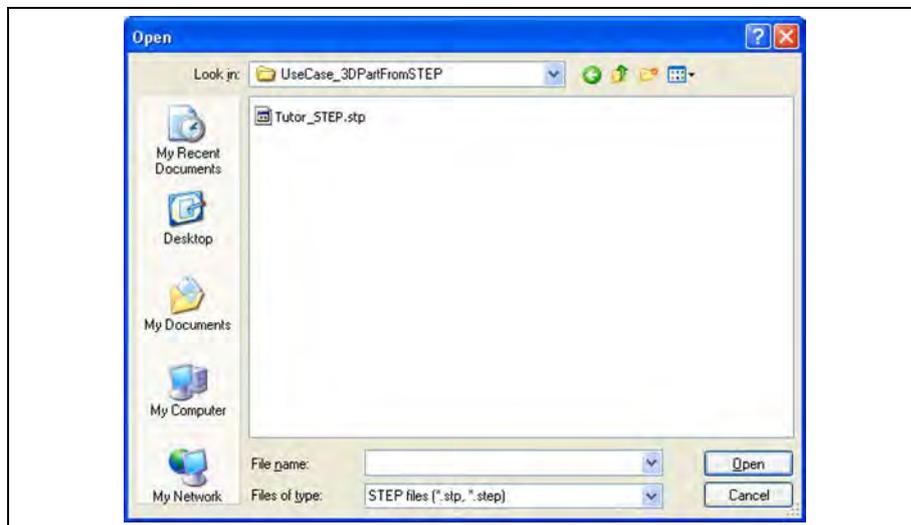
Tip

The LPKF tutor data is stored in "My Document\LPKF Laser&Electronics\LPKF CircuitPro 1.5\Example Data\UseCase_3DPartFromSTEP".

First the STEP file is imported in CircuitPro. Afterwards the imported data will be aligned to the processing volume.

- Importing data
- 1. Click on File > Import 3D shape...
- ➔ The following dialog is displayed:

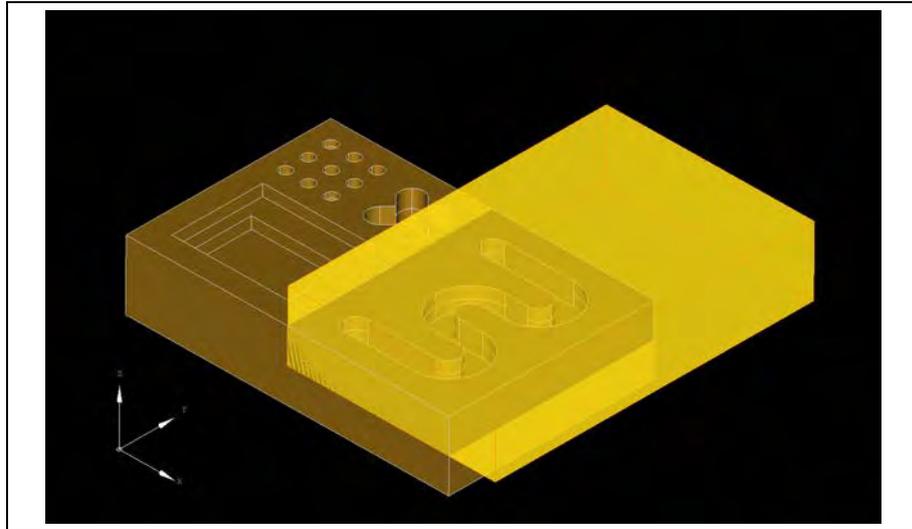
Fig. 89: Importing the STEP file



2. In the folder "UseCase_3DPartFromSTEP", select the file "Tutor_STEP.stp".
3. Click on [OK].

➔ The 3D view changes as follows:

Fig. 90: 3D view



◆ The data was imported.

■ Aligning the imported data to the processing volume

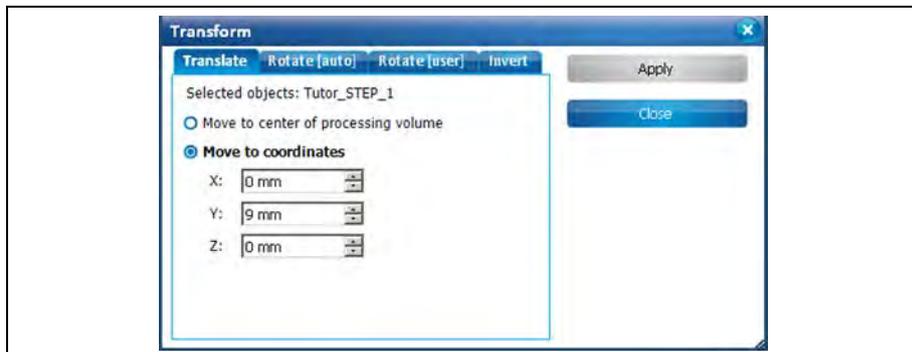
1. In the 3D view, right-click on the imported object.

➔ The context menu is shown.

2. In the context menu, click on "Transformation".

➔ The following dialog is displayed:

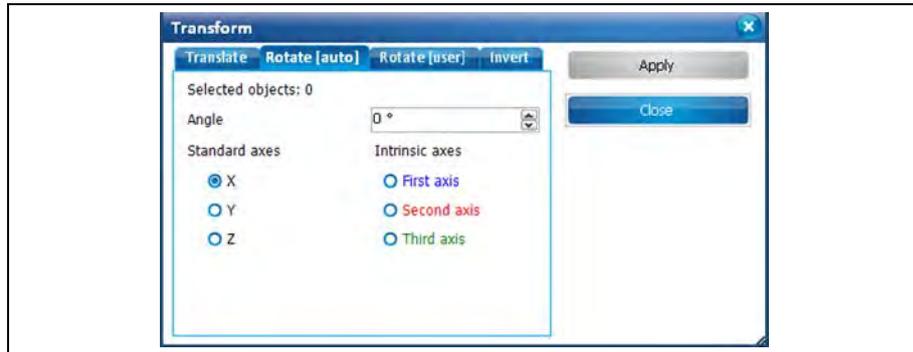
Fig. 91: Transformation



3. Click on "Rotate [auto]".

➔ The view changes as follows:

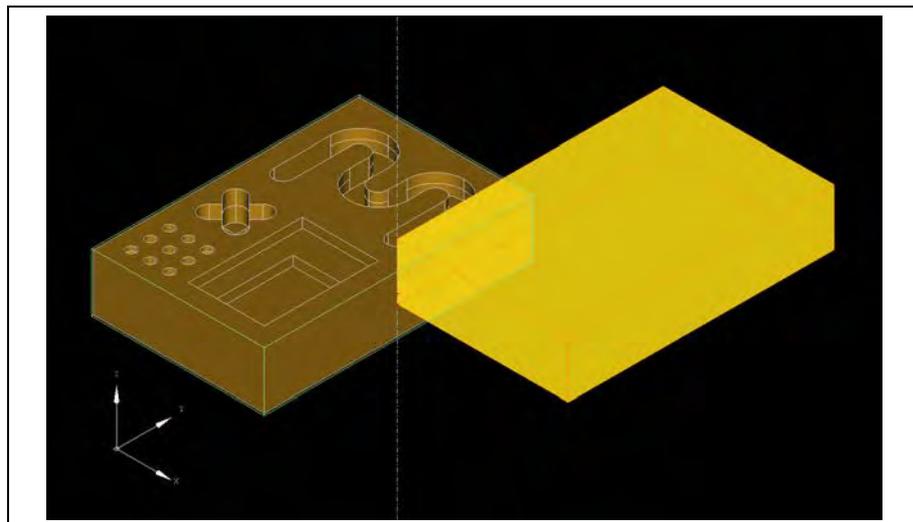
Fig. 92: Rotate
[auto]



4. Enter "90" in the field \Angle\.
5. Activate the standard axis "Z".
6. Click on [Apply].

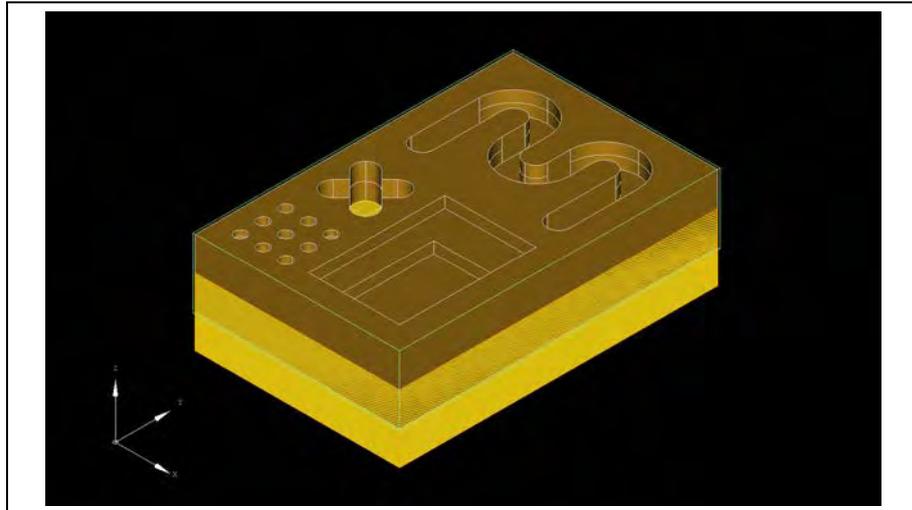
➔ The object is rotated around the Z axis at an angle of 90°. The 3D view changes as follows:

Fig. 93: 3D view
rotated around Z
axis



7. Click on the tab "Translate".
 8. Activate the option {Move to center of processing volume}.
 9. Click on [Apply].
- ➔ The object is moved in X and Y direction to the center of the processing volume. The 3D view changes as follows:

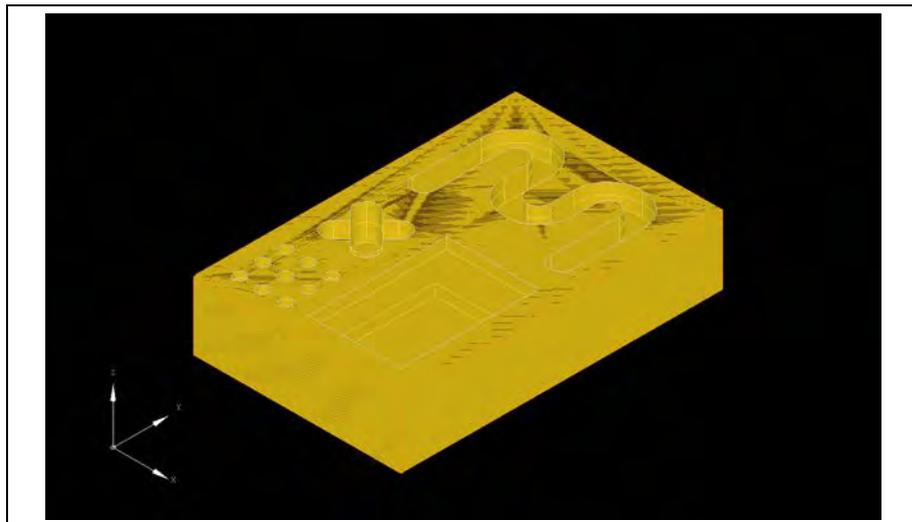
Fig. 94: Move in X
and Y direction



10. In the tab “Translate”, activate the option {Move to coordinates}.
11. Enter “3” in the field \Z\.
12. Click on [Apply].

➔ The object is moved in Z direction. The 3D view changes as follows:

Fig. 95: Move
object in Z
direction



13. Click on [Close].
- ➔ The dialog is closed.
 - ◆ The imported data was aligned to the processing volume.

2.4 Creating 2.5D toolpaths

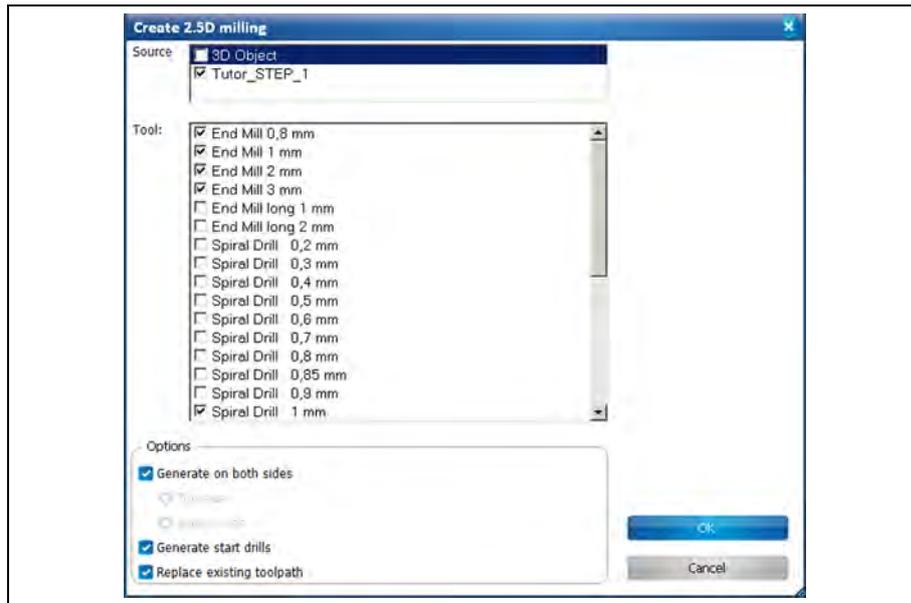
Before you insert the fiducials in your layout it is useful to create the 2.5D toolpaths first.

■ Creating 2.5D toolpaths

1. Click on Toolpath > 2.5D milling...

➔ The following dialog is displayed:

Fig. 96: Create 2.5D milling



2. In section \Source\, select the source data "Tutor_STEP_1" for creating the toolpath.
3. Select the tools that are available for the current job.



Note

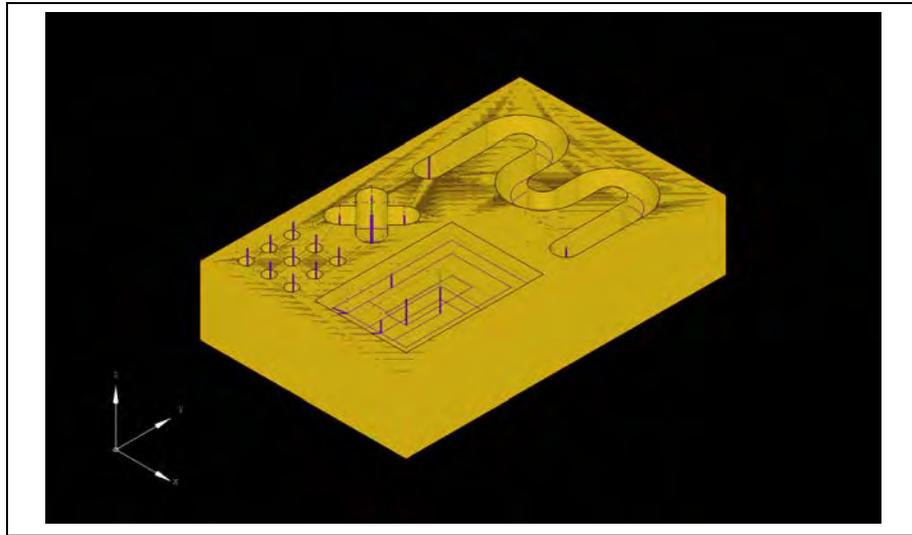
For creating uncontinuous drill holes you need an End Mill 0.8 mm. All other structures can be created with the End Mill 1 mm or 2 mm.

Also make sure that you have selected the appropriate drilling tools if you want to generate start drills for the End Mill tools.

4. Click on [OK] to create the toolpaths.

- ➔ The toolpaths are created and visible in the CAM and 3D view. The 3D view changes as follows:

Fig. 97: 3D view



Tip

You can rotate the object in the 3D view in any direction by activating this icon on your toolbar.



- ◆ The 2.5D toolpaths were created.

2.5 Inserting fiducials

For the exact alignment of the top and bottom side of the 3D part you need fiducials. Fiducials are optical marks on the surface of the machined part with a defined diameter of 1.5 mm. The Fiducials are drilled into the workpiece and are recognized by the camera of the ProtoMat systems.



Tip

Ideally you insert four fiducials for aligning the top and the bottom side.

You are also able to work with two fiducials. In this case it is useful to insert them diagonally into the layout.

■ Creating new phases for the fiducials

To be able to work with fiducials you have to create new phases to drill and to read the fiducials. To do this act as follows:

1. Click in the pane "Toolpath"
2. Click on the arrow button next to "Phases".

➔ The current job's phases are displayed:

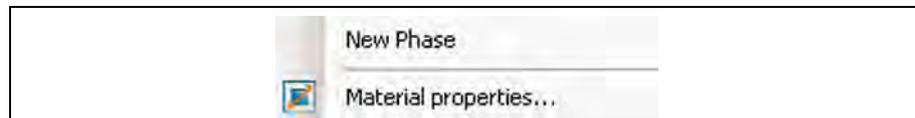
Fig. 98: Phases in pane "Toolpath"



3. Right-click on "Phases".

➔ The context menu is shown:

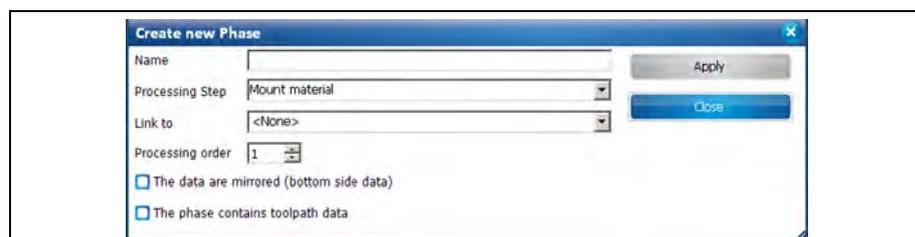
Fig. 99: Context menu



4. Click on "New Phase".

➔ The following dialog is shown:

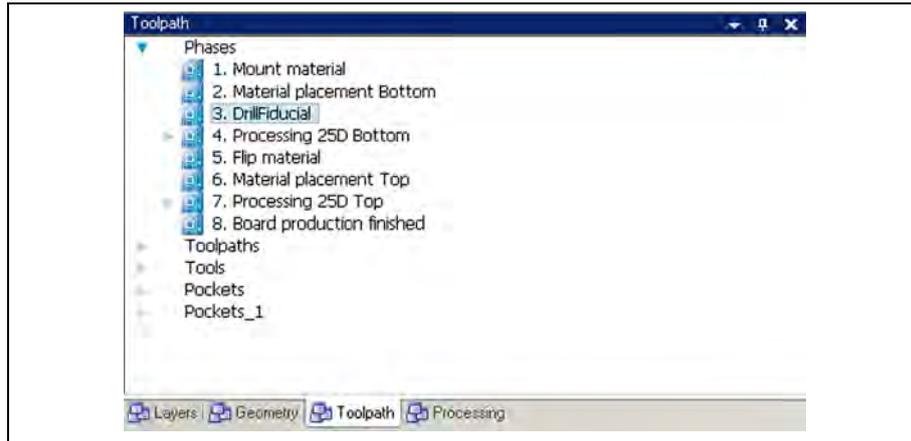
Fig. 100: Create new phase



5. Enter "DrillFiducial" in the field \Name\.
6. Select "DrillFiducials" in the drop down menu \Processing step\.
7. Enter "3" in the field \Processing order\.
8. Activate the option <The date is mirrored (bottom side)>.
9. Activate the option <The phase contains toolpath data>.
10. Click on [Apply].

➔ A new processing phase „DrillFiducial“ was created:

Fig. 101: New phase created



11. Create another processing phase using the following values:

- Name: ReadFiducialsTop
- Processing Step: Read Fiducials Top
- Link to: <None>
- Processing order: 7
- <The date is mirrored (bottom side)>: Not activated
- <The phase contains toolpath data>: Activated

12. Click on [Apply].

➔ A new processing phase „ReadFiducialsTop“ was created.

13. Click on [Close].

➔ The dialog is closed.

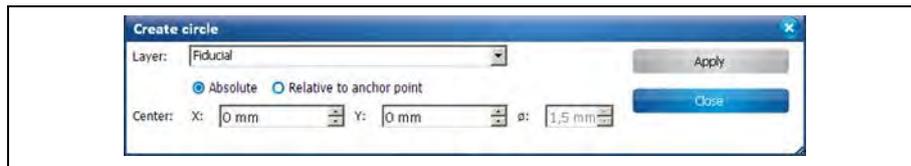
◆ The new phases for the fiducials were created.

■ Inserting fiducials

1. Click on Insert > Fiducial > Create new layer...

➔ A new layer "Fiducial" is created. The following dialog is displayed:

Fig. 102: Creating circles

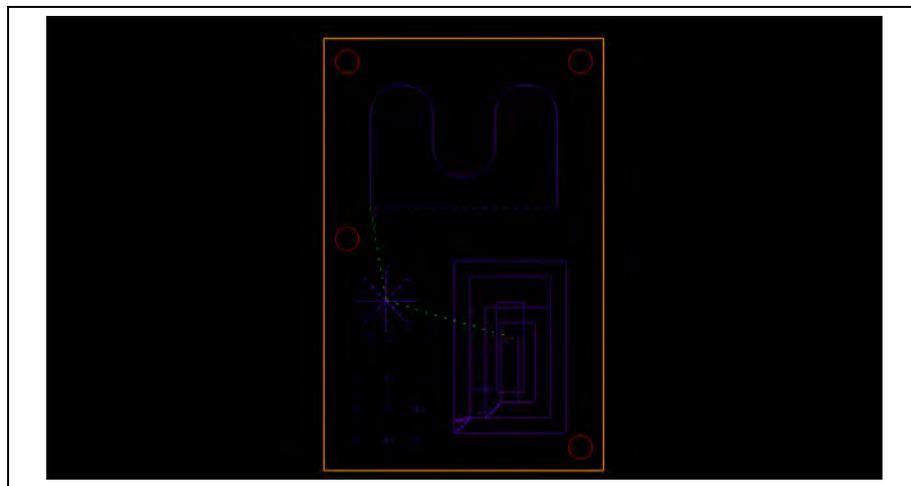


In the drop down menu "Layer" the new created layer „Fiducial“ is preselected.

Note

2. Activate the option "Absolute" (the center point is calculated from the zero point).
 3. Enter "1.5" in the field \Center X\.
 4. Enter "26.5" in the field \Center Y\.
 5. Click on [Apply].
- ➔ The fiducial is created.
6. Repeat the steps 3 to 5 using the following coordinates to create three more fiducials:
 - Center X: 1.5; Center Y: 15
 - Center X: 16.5; Center Y: 26.5
 - Center X: 16.5; Center Y: 1.5
- ➔ The Cam view now looks as follows:

Fig. 103: CAM view with fiducials





Note

You are able to create fiducials without entering any coordinate. Therefore left-click in the CAM view where you want to place the fiducials.

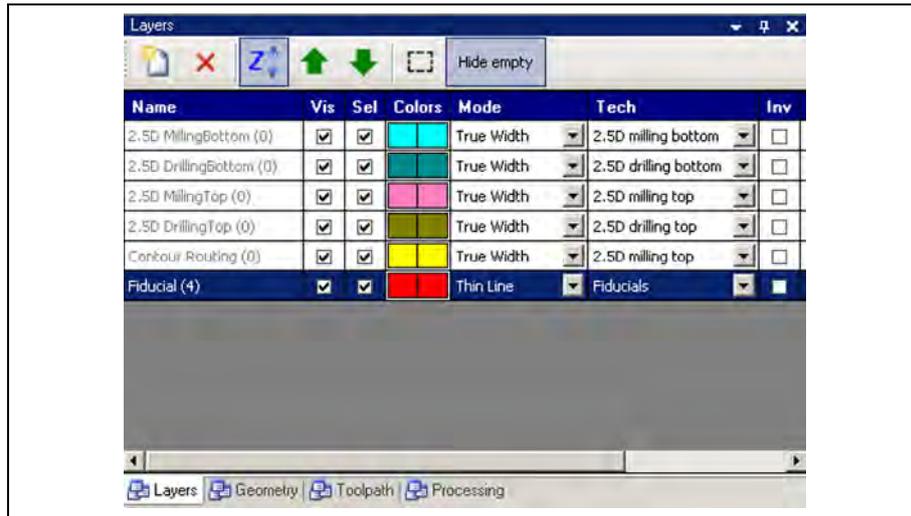
7. Click on [Close].
 - ➔ The dialog is closed.
 - ◆ The fiducials were inserted.

2.6 Creating toolpaths for fiducials

The toolpaths for the fiducials have to be created before starting the production of the 3D part. To do this the layer "Fiducial" has to be assigned to a processing phase first.

- Assigning the layer "Fiducial" to a processing phase
 1. Click on the pane "Layers".
 2. Select the row of the desired layer:

Fig. 104: Pane "Layers"



3. In the layer row, click on the arrow button of the column "Phase".

➔ The phase selection list is shown:

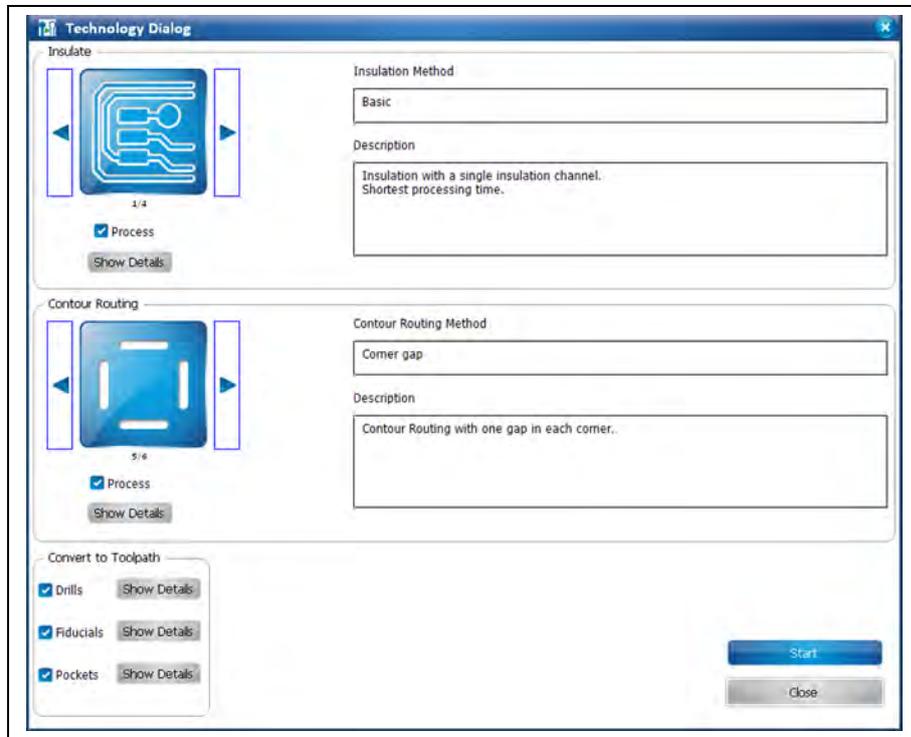
Fig. 105: Phase selection list



4. Assign the phase "DrillFiducial" to the layer "Fiducial".
 - ◆ The layer "Fiducial" was assigned to a processing phase.

- Creating toolpaths for the fiducials
1. Click on Toolpath > Technology Dialog...
- ➔ The following dialog is shown:

Fig. 106:
Technology
Dialog



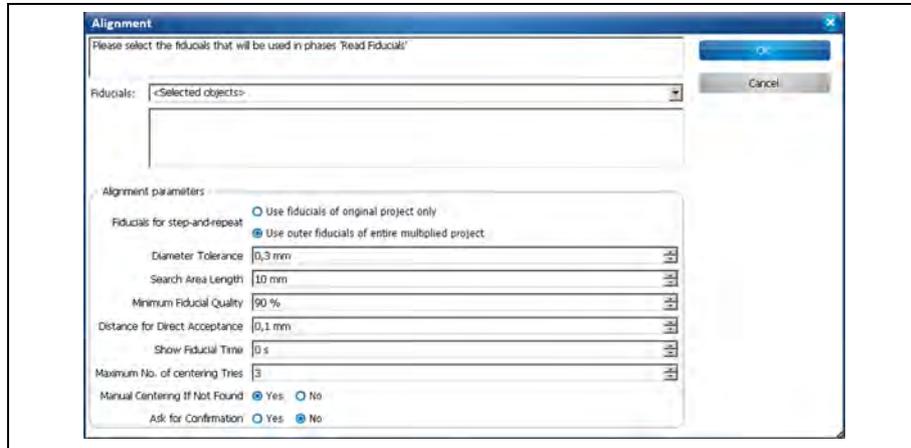
2. Deactivate the following functions:
 - Insulate
 - Contour Routing
 - Drills
 - Pockets
3. Click on [Start] to create the toolpaths.
 - ➔ The toolpaths are created and the computation results are displayed.
4. Click on [Close].
 - ➔ The dialog displaying the computation results is closed.
 - ◆ The toolpaths for the fiducials were created.

■ Defining the fiducials for the alignment

1. Click on Maching > Alignment...

➔ The following dialog is displayed:

Fig. 107:
Alignment

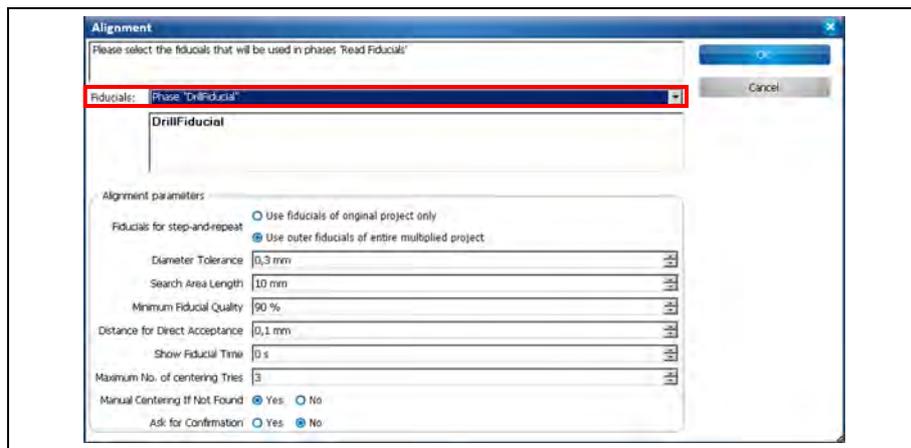


2. Click on the arrow button in the row "Fiducial".

➔ The selection list for the fiducials is shown.

3. Select the fiducials of the phase "DrillFiducial" for the alignment:

Fig. 108:
Selecting the
fiducials



4. Click on [OK].

➔ The dialog is closed.

◆ The fiducials are defined.

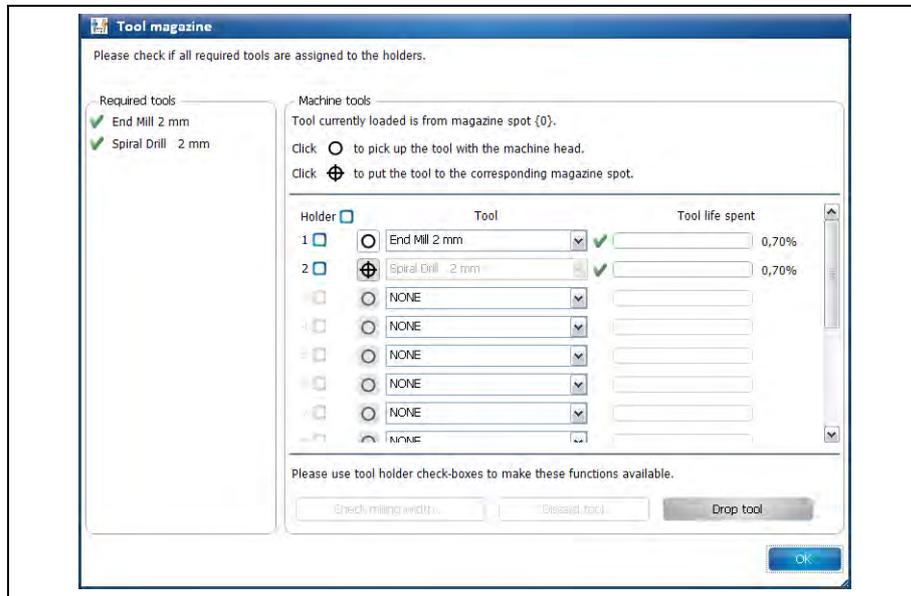
2.7 Loading the tool magazine and assigning tools to holder positions

- Loading the tool magazine and assigning the tools to holder positions

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

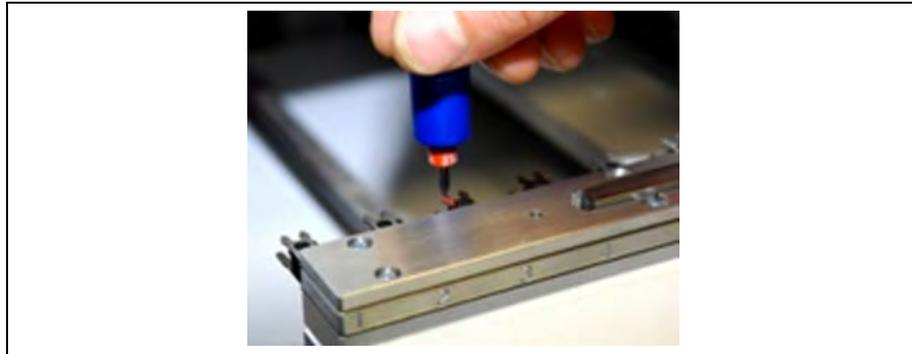
Fig. 109: Tool magazine



➔ The tools required for the job are displayed. Tools that are missing in the tool magazine are marked by a red "X".

2. Insert the required tools into the tool magazine:

Fig. 110: Inserting the tool

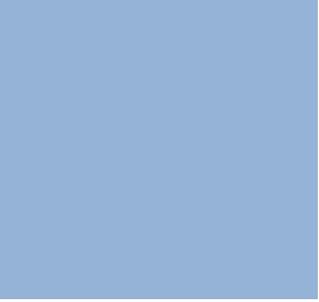


3. In the dialog, assign the tools to the respective tool magazine positions used.
4. Repeat the steps above until all required tools are assigned:

Fig. 111: Tools in the tool magazine



5. Click on [OK]
- ◆ The tools were loaded and assigned to their positions.



Creating a multi-layer PCB with galvanic through-hole plating



3 Creating a multi-layer PCB with galvanic through-hole plating

This tutorial shows you how to produce a 4-layer circuit board with galvanic through-hole plating.

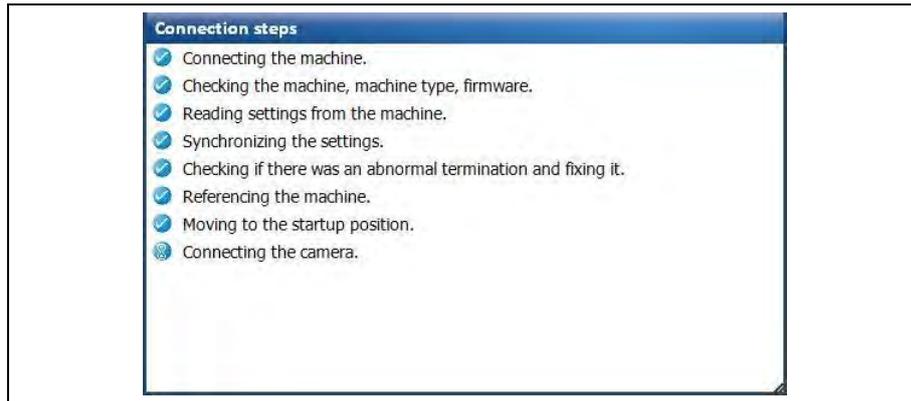
The following steps are necessary to complete the tutorial successfully:

- i. Starting the machine and CircuitPro
- ii. Selecting a template and creating a new document
- iii. Importing data
- iv. Inserting rubout areas
- v. Multiplying the design (if necessary)
- vi. Creating toolpaths
- vii. Loading the tool magazine and assigning tools to holder positions
- viii. Starting the processing

3.1 Starting the machine and CircuitPro

- Starting the machine and CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed:

Fig. 112:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to its reference points and subsequently moves to the Pause position.

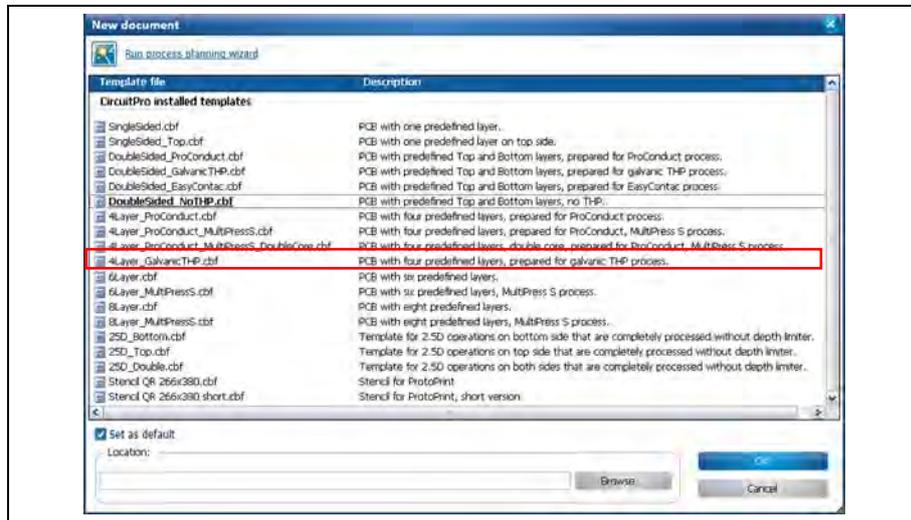
3.2 Selecting a template and creating a new document

■ Selecting a template and creating a new document

1. Click on File > New...

➔ The following dialog is displayed:

Fig. 113: New document

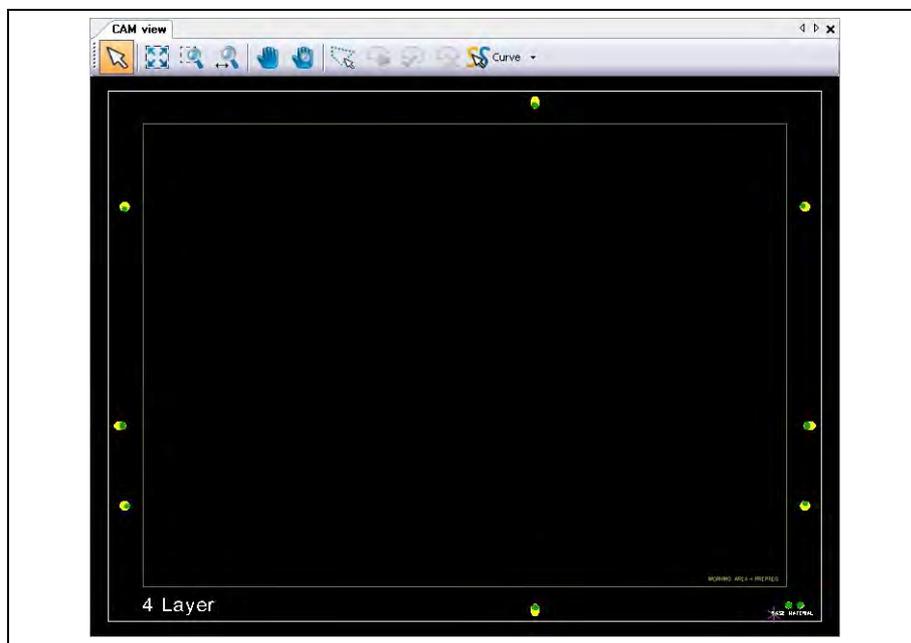


2. Select the template "4Layer_GalvanicTHP.cbf".

3. Click on [OK].

➔ The template is displayed in the CAM view:

Fig. 114: CAM view - multi-layer



**Note**

The template "4Layer_GalvanicTHP.cbf" already contains fiducials. These do **not** have to be added anew.

The template contains four slot holes that can be used to affix the individual layers to the press plate for the pressing step.

The template also contains two positioning holes for orientation during the alignment of the individual layers.

4. Click on File > Save As...
5. Enter a file name for the new file.
6. Click on [Save].

- ◆ The new document is created.

3.3 Importing data



Tip

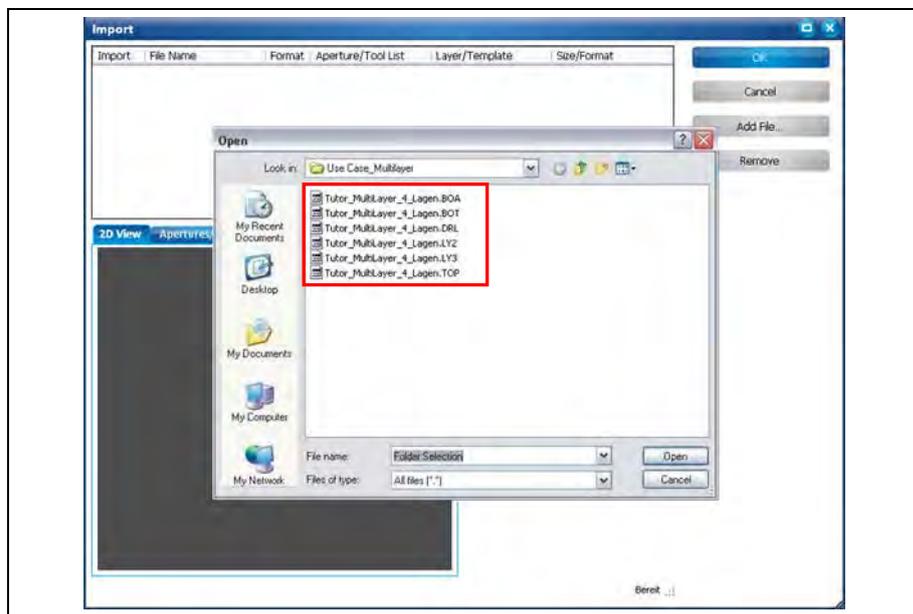
The LPKF tutor data are stored in folder “My documents\LPKF Laser & Electronics\LPKF CircuitPro 1.5\Example Data\UseCase_Multilayer”.

■ Importing data

1. Click on File > Import...

➔ The following dialog is displayed:

Fig. 115: Import



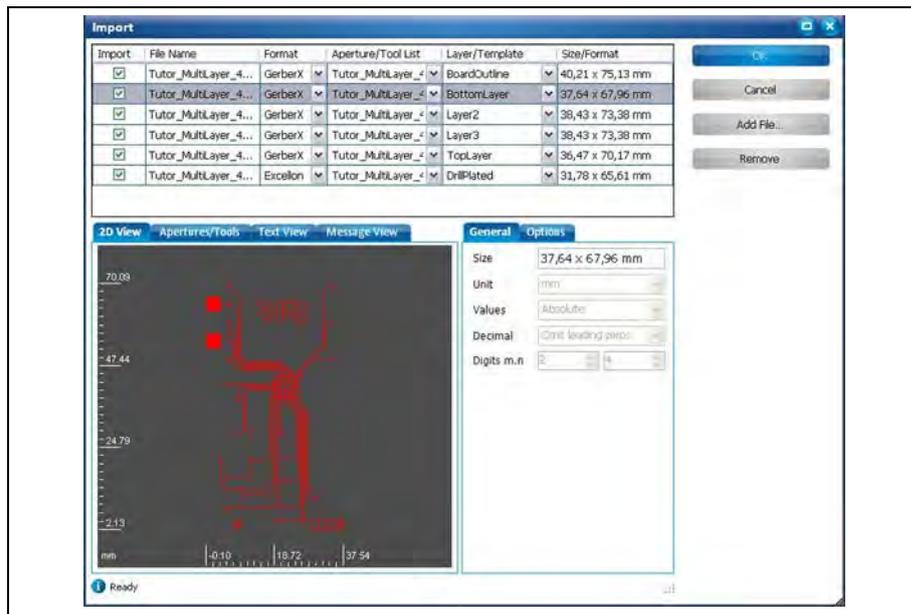
2. Select all files in the folder “UseCase_Multilayer”.

3. Click on [OK].

4. Assign the imported files to the corresponding layers (see table):

File extension	Layer
.BOA	Board Outline
.BOT	Bottom Layer
.LY2	Layer 2
.LY3	Layer 3
.TOP	Top Layer
.DRL	Drill Plated

Fig. 116:
Assigned layers



Note

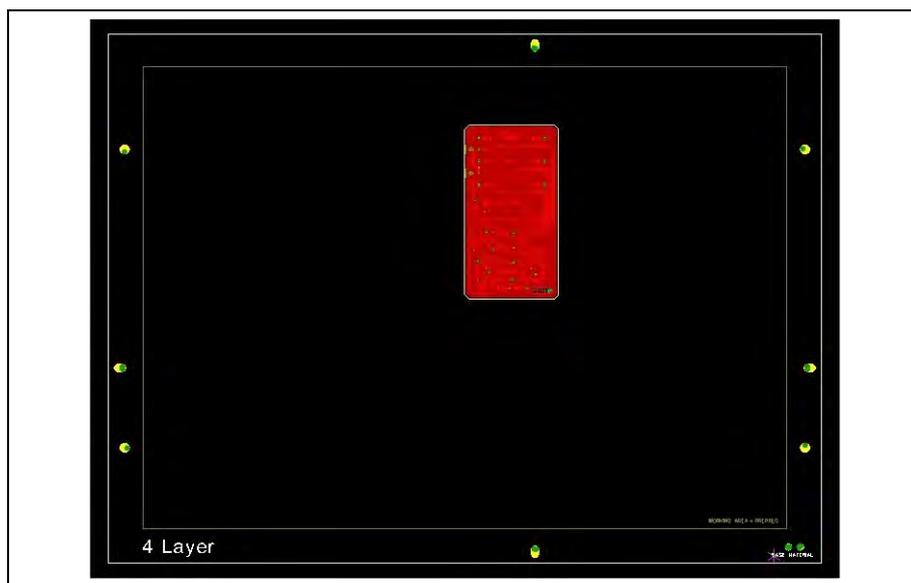
Instead of manually assigning the individual files to the layers, you can activate the options “Use layer name” and “Apply to all Gerber files”. Activate the corresponding checkboxes in the tab “Options”

If a file contains layer names these are automatically assigned. Please note that this is only available for Gerber files. All other files require assigning the layers manually via the drop-down menu.

5. Click on [OK].

➔ The data is shown in the CAM view:

Fig. 117: CAM
view



◆ The files are imported.

3.4 Inserting rubout areas

Inserting rubout areas is used for creating a most precise isolation in certain areas by removing the redundant copper partially or as a whole depending on the insulation method.

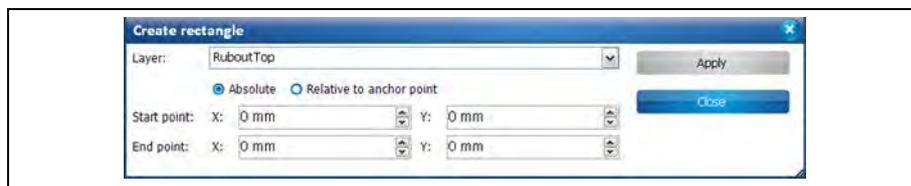
In this tutorial, a rubout area covering the whole circuit board is created on the Top and Bottom layers.

■ Inserting a rubout area

1. Click on Insert > Rubout area > RuboutTop.

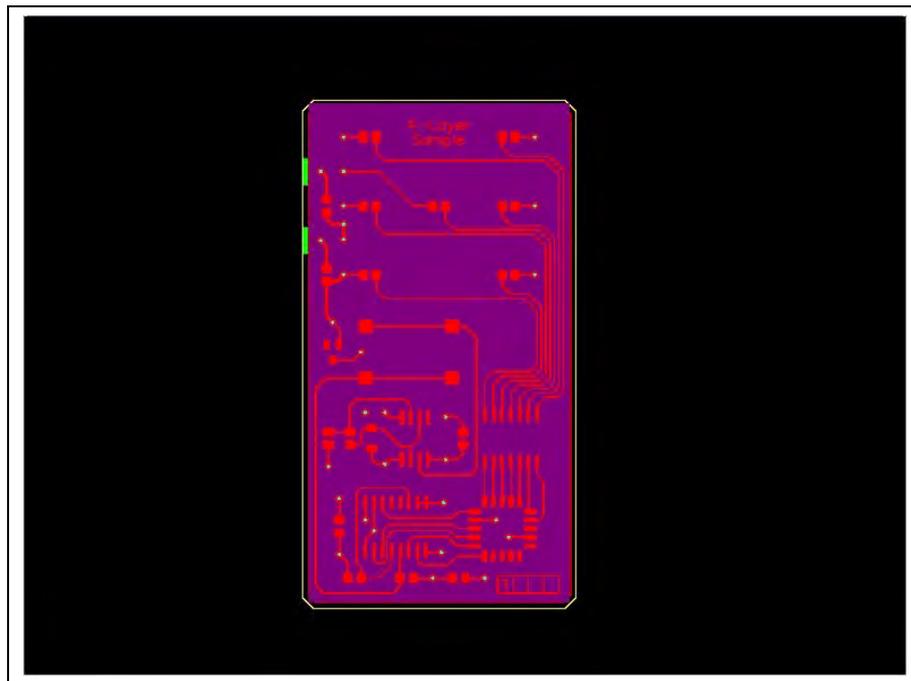
➔ The following dialog is displayed:

Fig. 118: Create Rectangle



2. Draw a rectangle across the whole area of the design using the mouse:

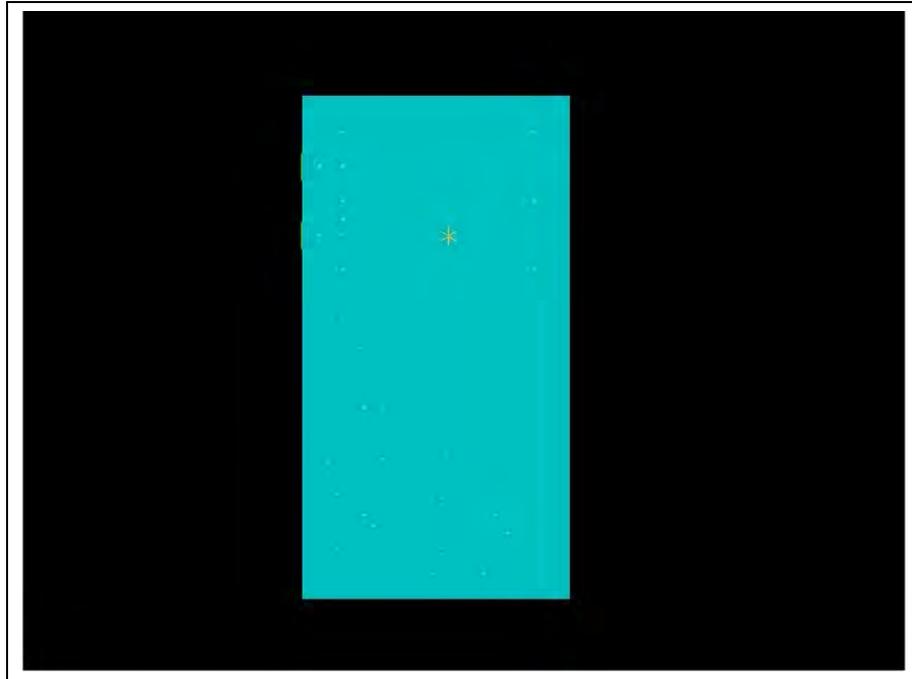
Fig. 119: Rubout Top



3. Select "RuboutBottom" in the "Layer" selection list of the dialog.

4. Draw another rectangle across the whole area of the design using the mouse:

Fig. 120: Rubout
Bottom



5. Click on [Close].
 - ➔ The dialog is closed.
 - ◆ The rubout areas are created.



Note

It may be helpful to hide other layers for drawing the rubout depending on which layer the rubout is to be created.

The display mode of the objects on the individual layers can be set in the “Layers” pane. You can select

- True width (area objects are filled, paths are displayed with their true width)
- Outline (outlines of the area objects and paths are displayed)
- Thin line (outlines of area objects are displayed and thin lines without defined width in case of paths)
- Unknown (used when importing .cam files with undefined objects)

3.5 Multiplying the design (if necessary)

The design can be multiplied and placed arbitrarily on the base material for producing a panel depending on the design's size and on the material's size.



Note

Please make sure, that all layers in the "Layer" pane are visible and selectable.

Otherwise it may cause incorrect results when multiplying the design.

- Multiplying the design
 1. Select the whole design.
 2. Click on Modify > Step & Repeat...
- ➔ The following dialog is displayed:

Fig. 121: Step & Repeat



3. Enter "2" in the \Repetition X\ field.
4. Enter "50" in the \Distance X\ field.



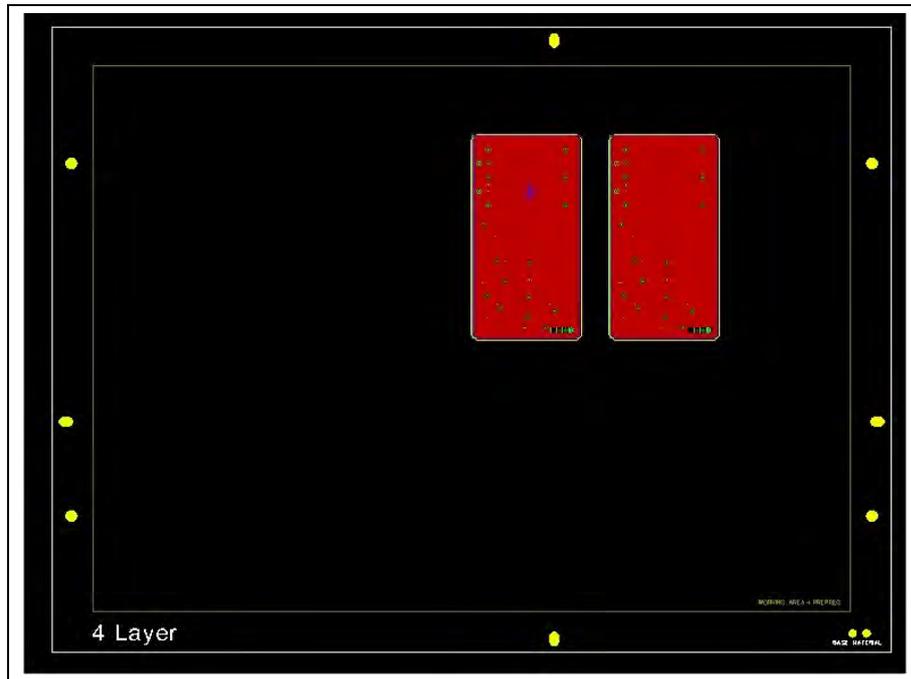
Note

The "Distance" values are automatically set to the size of the design. The value for spacing between the designs has to be added.

5. Click on [Apply].

➔ The design is multiplied in X direction:

Fig. 122: Design
in the CAM view



6. Click on [Close].

➔ The dialog is closed.

◆ The design is multiplied.

3.6 Creating toolpaths

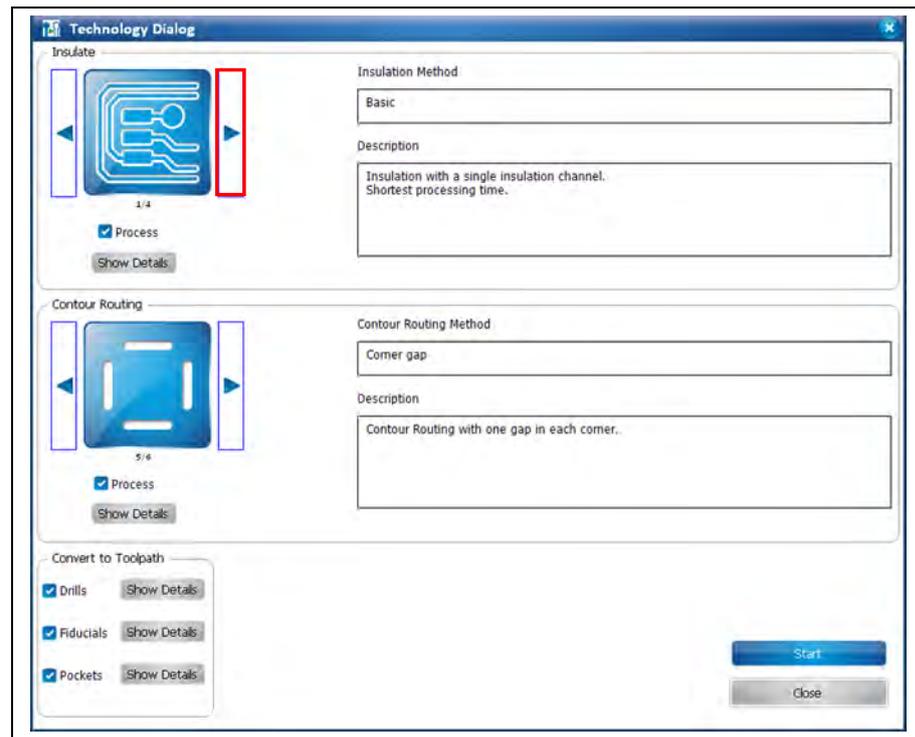
Before the circuit board can be produced, the toolpaths have to be generated for the imported data.

■ Creating toolpaths

1. Click on Toolpath > Technology dialog...

➔ The following dialog is displayed:

Fig. 123:
Technology
Dialog



Note

In the technology dialog, several settings can be modified by clicking on the [Show details] buttons.

For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

For this tutorial, the default values of the technology dialog are used.

2. Click on the right arrow button in the "Insulate" section until the isolation method "Partial rubout" is displayed.
3. Click on [Start].
- ➔ The computation results are displayed.
4. Click on [Close].
- ➔ The dialog with the computation results of the technology dialog is closed.
- ◆ The toolpaths are created.

3.7 Loading the tool magazine and assigning tools to holder positions



This chapter is only relevant if you are using a ProtoMat with automatic tool change (S63 or S103).

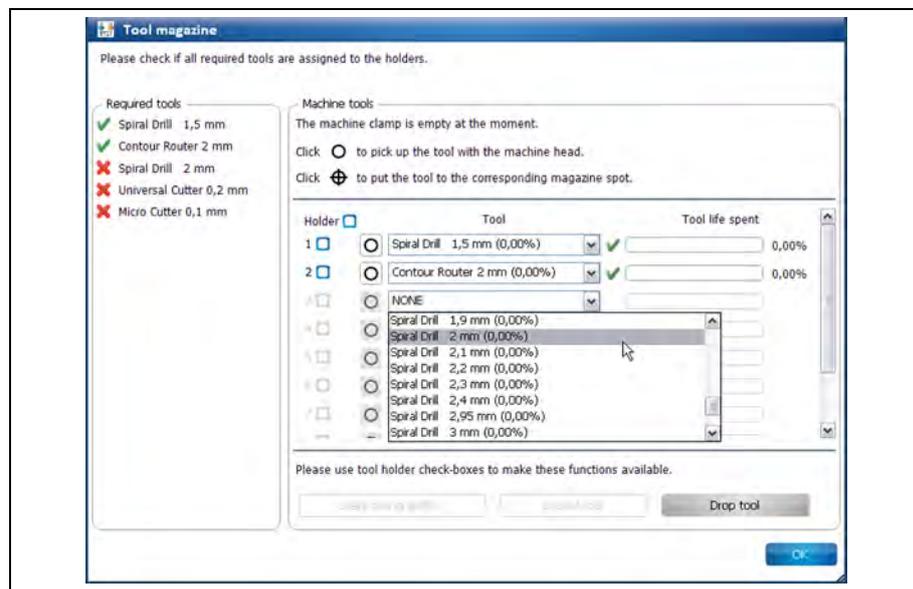
Note

■ Loading the tool magazine and assigning tools to holder positions

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

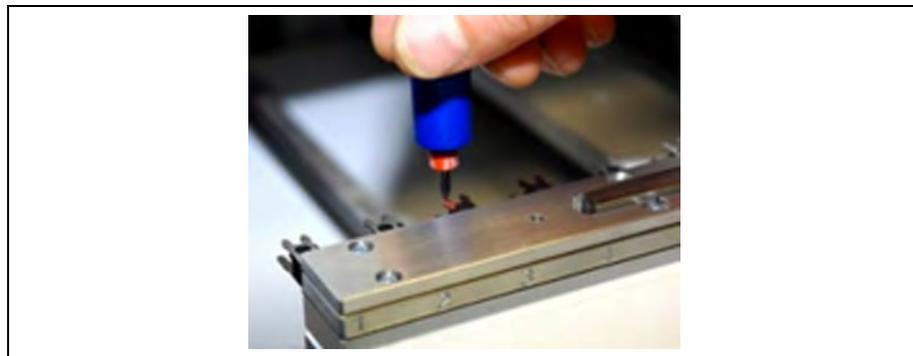
Fig. 124: Tool magazine



➔ The tools required for the job are displayed. Tools required for the job that are missing are marked by a red "X".

2. Insert the required tools into the tool holders of the machine:

Fig. 125: Inserting a tool



3. Assign the tools to the corresponding positions in the dialog.

➔ The tool holders of the machine are loaded:

**Fig. 126: Loaded
tool holder**



4. Click on [OK].

➔ The dialog is closed.

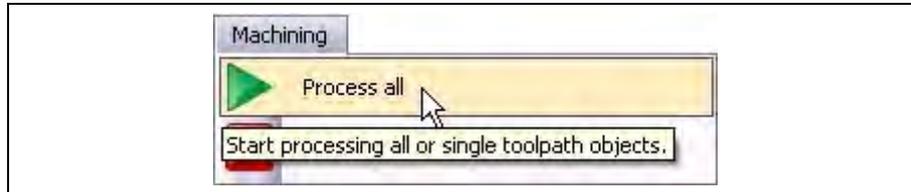
◆ The tool magazine is loaded and the tools are assigned to the corresponding holder positions.

3.8 Starting the processing

■ Starting the processing

1. Click on Machining > Process all.

Fig. 127:
Machining > Pro-
cess all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

Once you have started the processing, the ProtoMat machine executes the job in individual phases. The phases are displayed in messages:



Note

Depending on which ProtoMat you use the following phases could differ from the phases and messages displayed on your screen. Please follow the instructions on your screen.

For machines with manual tool exchange you are regularly asked to change the tool in the collet, for example.

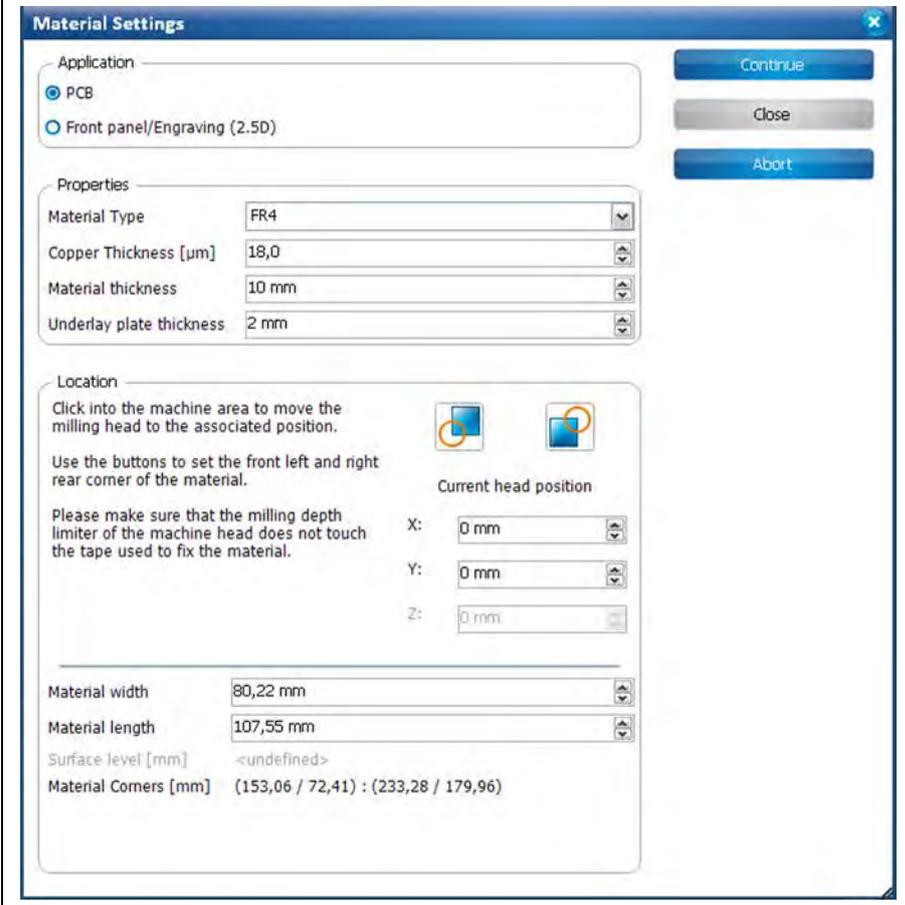
Phase "MountMaterial"

1. Mount the base material (multi-layer core) onto the processing area of the machine.
2. Fasten the material to the processing area using adhesive tape.
3. Click on [OK].

Phase "MaterialSettings"

➔ The following dialog is displayed:

Fig. 128: Material Settings



■ Entering the material settings

1. Enter the correct values for the base material used.

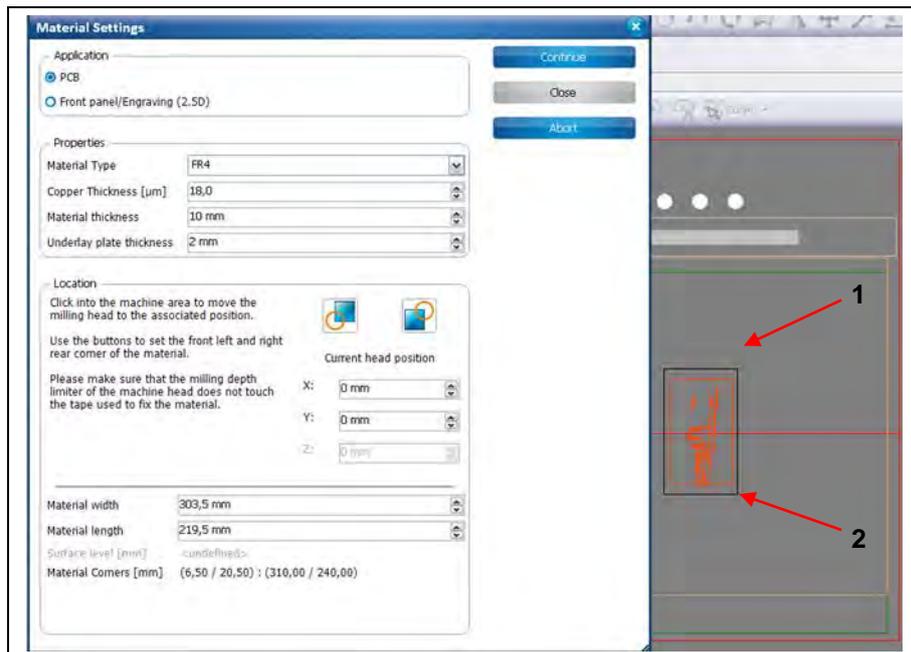


PCB is selected as default.

Note

2. Adapt the values for copper thickness and material thickness if necessary.
3. Define the processing area:
 - a) Move the "Material Settings" dialog off to the side.
 - b) Click on the position in the machining view that represents the right corner of your material:

Fig. 129: Defining
the left corner



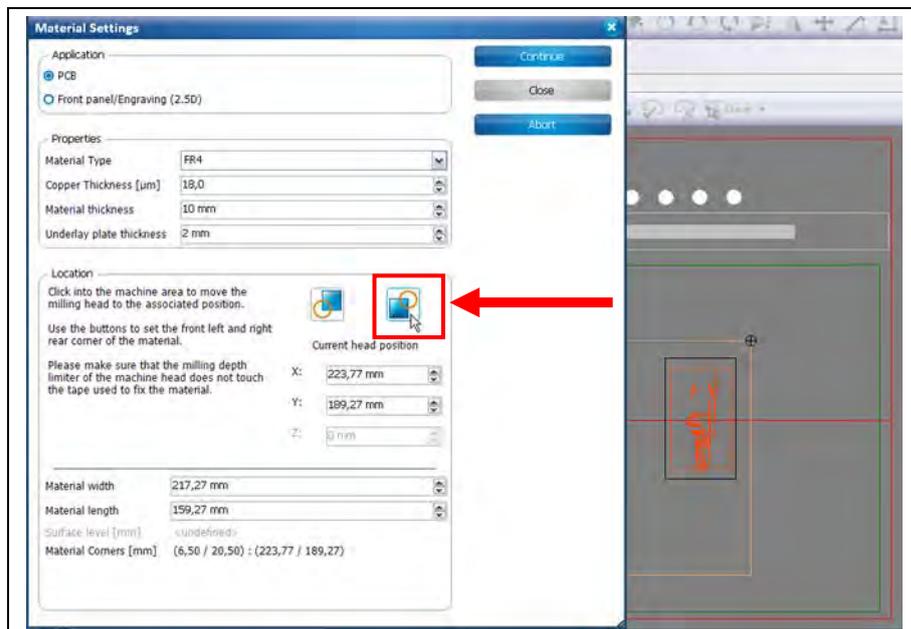
/1/ Click on machining area

/2/ Job

➔ The machine head moves to this position.

c) Click on the corresponding icon in the "Material Settings" dialog:

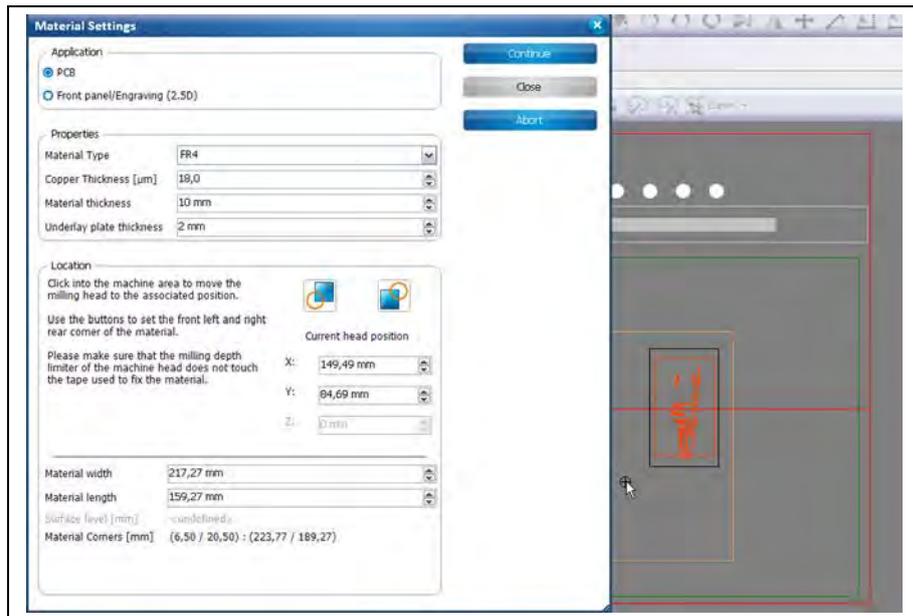
Fig. 130: Click on
the icon



➔ The machining area is adjusted.

d) Click on the position in the machining view that represents the right corner of your material:

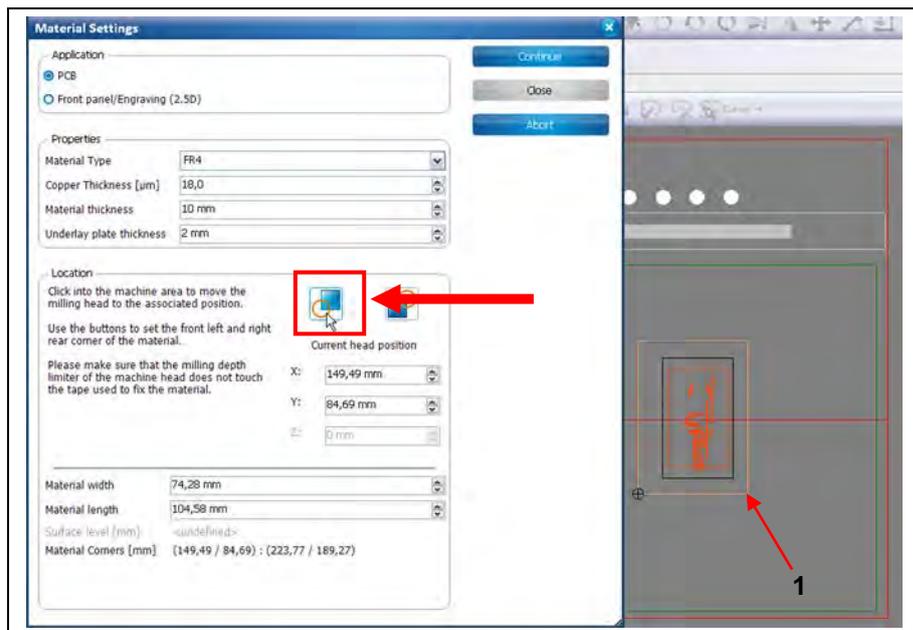
Fig. 131: Defining the left corner



➔ The machine head moves to this position.

e) Click on the corresponding icon in the “Material Settings” dialog:

Fig. 132: Defined processing area



/1/ Defined processing area

➔ The processing area has been fit to the material.

4. Click on [Continue].

◆ The material settings were entered.

Phase “Placement”

In this phase, the job can be placed arbitrarily on the base material, be rotated and be multiplied if necessary.



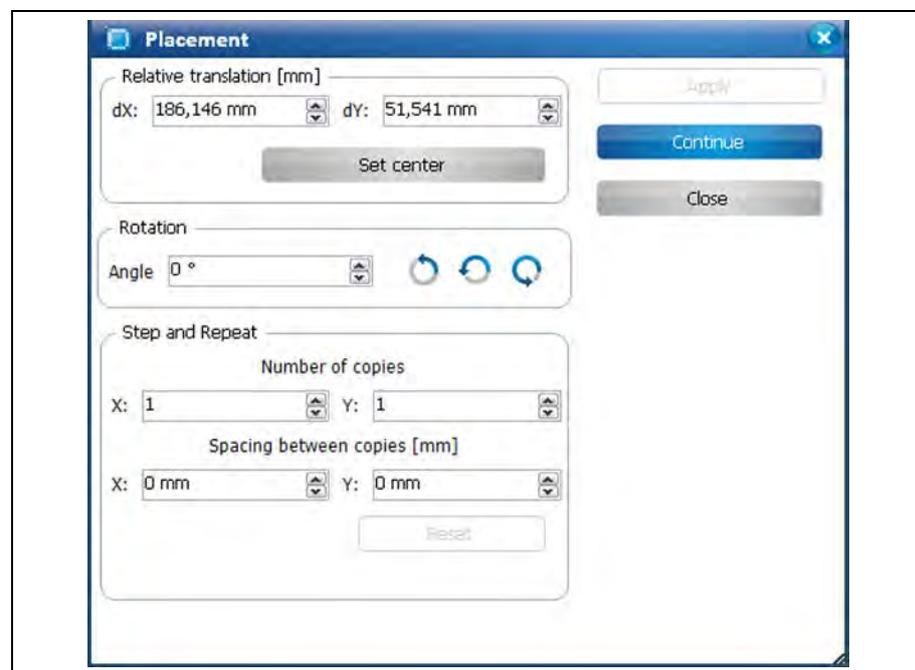
Note

At this point, the whole circuit board design including the template and the toolpaths possibly contained in the template are moved.

If only the design is to be moved within the template, you have to use the menu item Modify > Transform....

➔ Following dialog is displayed:

Fig. 133:
Placement



1. Click on the job and drag it to the desired position using the mouse.
- Or
1. Enter the new position in the dialog.
 2. If desired, multiply the job data by entering the number of copies and spacing values in X and Y direction in the corresponding fields (Step and Repeat section).

Phase “DrillFiducial”

- ➔ The Spiral Drill 1.5 mm is picked up to drill the fiducials.



If the spindle motor has not run before, a 2-minute warm-up phase is started.

Note

Phase “Prepare Core”

- ➔ Four slot holes are created near the edges of the circuit board.

Phase “Milling Layer3”

- ➔ The tools are picked up as required and Layer3 is processed.

Phase “Flip Material”

A message prompts you to turn the material over.

1. Turn the base material over.



Turn the material over around the X axis if you are using an S43, S63, or S103 machine.

Note

Turn the material over around the Y axis if you are using an E33 machine.

2. Click on [OK].



The display in the machine view changes. The position of your design is adapted to the material. The material side to be processed now is Top.

Note

Phase “Read Fiducials Layer2”

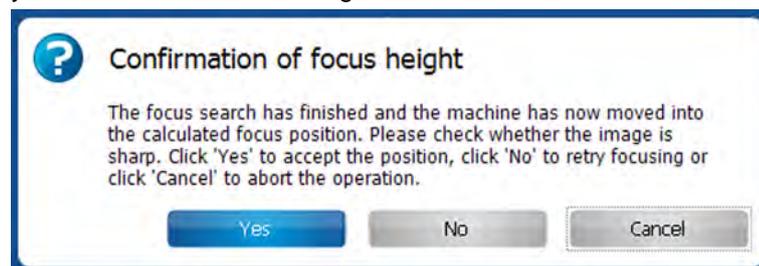
S43, S63, and S103



If the fiducial search is performed for the first time (after having started CircuitPro) the camera is performing an autofocus five times.

Note

Afterwards the following message is displayed which prompts you to confirm the focus height:

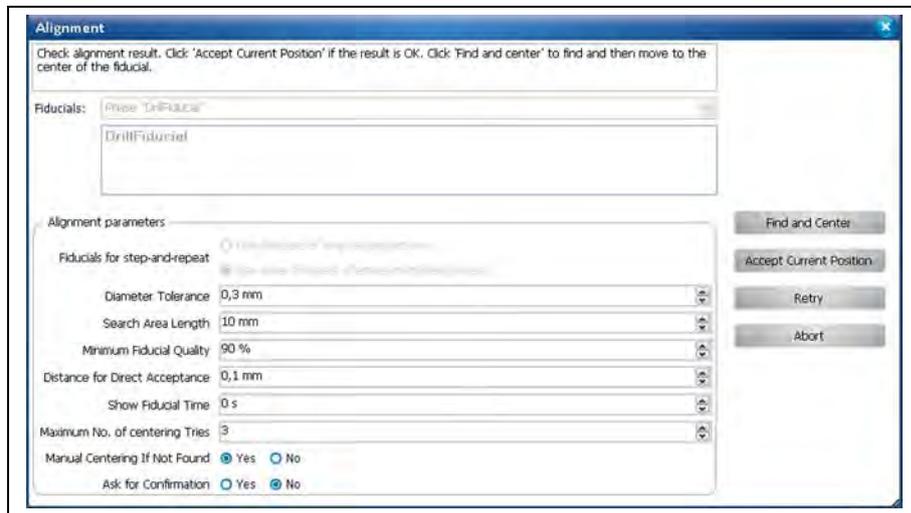


- ➔ The camera moves to the positions of the fiducials and determines the exact position.

The Top side is thus aligned to the Bottom side.

The following dialog is displayed if the fiducials have not been recognized automatically:

Fig. 134:
Alignment



1. Enlarge the search area by increasing the value of the field \Search Area Length\.
2. Restart the search by clicking on [Start].
3. Repeat steps one and two if necessary.



Note

Enlarging the search spiral increases the time required for searching the fiducials. Try to put the material at the same position as before when turning the material over (if this is not predetermined by reference pins).

E33, S43 without camera

If no camera is available for processing the “Read Fiducials_Top” phase, the Top side is aligned to the Bottom side using the reference pins. The “Read Fiducials_Top” phase is not processed in this case.

Phase “Milling Layer2”

- ➔ The tools are picked up as required and Layer2 is processed.

Phase “Dismount Material”

1. Dismount the base material from the machine.
2. Click on [OK].

Phase “Mount Laminate”

1. Mount the laminate for the Bottom side of the multi-layer PCB onto the processing area of the machine.
2. Fasten the material to the processing area using adhesive tape.
3. Click on [OK].

Phase “MaterialSettings”

1. Adapt the values for copper thickness and material thickness of the laminate if necessary.
2. Set the material size.
3. Click on [OK].

Phase “Prepare Laminate Bottom”

- ➔ The slot holes, position markers and fiducial exposures are processed.

Phase “Dismount Material_1”

1. Dismount the laminate for the Bottom side of the multi-layer PCB from the machine.
2. Click on [OK].

Phase “Mount Laminate_1”

1. Mount the laminate for the Top side of the multi-layer PCB onto the processing area of the machine.
2. Fasten the material to the processing area using adhesive tape.
3. Click on [OK].

Phase „Prepare Laminate Top”

- ➔ The slot holes, position markers and fiducial exposures are processed.

Phase “Dismount Material_2”

1. Dismount the laminate for the Top side of the multi-layer PCB from the machine.
2. Click on [OK].

Phase “Press Layer Top_Bottom”

- ➔ The outer layers Top and Bottom have to be laminated onto the core base material. This can be done using an LPKF MultiPress S system for example.

Phase “Mount Material Top”

1. Mount the laminated material onto the processing area of the machine, the Top side facing upwards.



Note

The Top and Bottom sides can be distinguished by the position markers. The Top side has the position markers in the right front corner.

2. Fasten the material to the processing area using adhesive tape.

Phase “MaterialSettings”

1. Adapt the values for copper thickness and material thickness of the laminated material as required.
2. Define the material area.
3. Click on [OK].

Phase “Read Fiducials Top”

- ➔ See Phase “Read Fiducials Layer2” above.

Phase “Marking Drills”

- ➔ The Universal Cutter is picked up and the drill positions are marked.

Phase “Drilling Plated”

- ➔ The tools are picked up as required and the holes are drilled.



Note

This phase may use several drill tools according to the drill diameters.

Phase “Dismount Material”

1. Dismount the laminated material from the machine.
2. Click on [OK].

Phase “Through Hole Plating”

- ➔ The hole have to be plated. This can be done using an LPKF Contac RS system for example.

Phase “Mount Material Bottom”

1. Mount the laminated material onto the processing area of the machine, the Bottom side facing upwards.



Note

The Top and Bottom sides can be distinguished by the position markers. The Top side has the position markers in the right front corner.

2. Fasten the material to the processing area using adhesive tape.

Phase “MaterialSettings”

1. Adapt the values for copper thickness and material thickness if necessary.



Note

Please note that the copper thickness has increased during the through-hole plating process. The additional copper thickness depends on the parameters of the through-hole plating process.

2. Define the material area.
3. Click on [OK].

Phase “Read Fiducials Bottom”

- ➔ See Phase “Read Fiducials Layer2” above.

Phase “Milling Bottom”

- ➔ The tools are picked up as required and the circuit board is milled.

Phase “Flip Material”

A message prompts you to turn the material over.

1. Turn the material over.



Note

Turn the material over around the X axis if you are using an S43, S63, or S103 machine.

Turn the material over around the Y axis if you are using an E33 machine.

2. Click on [OK].



Note

The display in the machine view changes. The position of your design is adapted to the material. The material side to be processed now is Top.

Phase “Read Fiducials Top”

- ➔ See Phase “Read Fiducials Layer2” above.

Phase “Milling Top”

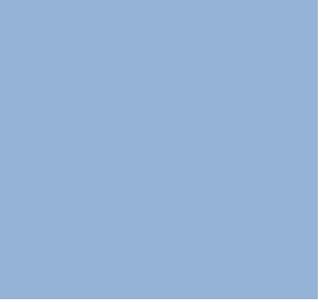
- ➔ The tools are picked up as required and the Top side of the circuit board is milled.

Phase “Contour Routing”

- ➔ The tools are picked up as required and the circuit board is drilled and routed.

Phase “Board Production Finished”

- ➔ A message informs you that the production is finished.
- ◆ The multi-layer circuit board is finished.



Creating a front panel

4 Creating a front panel

In this section you will learn how to create a front panel that contains following elements:

- Cutouts for switches and a fan
- Holes for fastening LED's
- Labels
- Markings

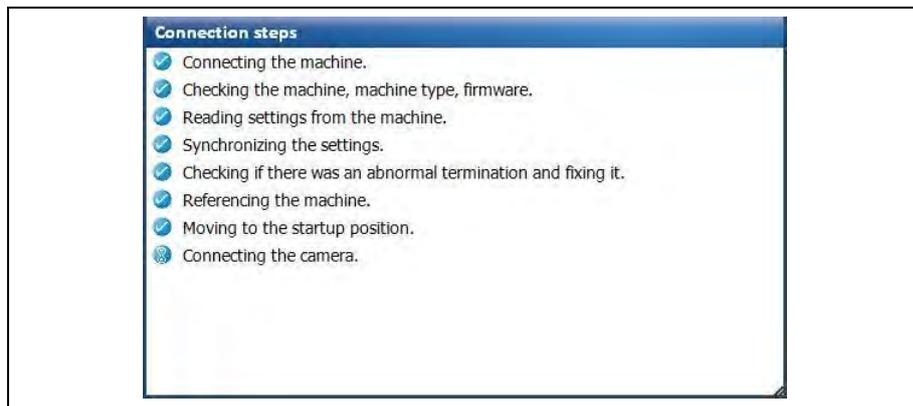
Therefore you have to perform following steps:

- i. Starting the machine and CircuitPro
- ii. Selecting a template and creating a new document
- iii. Creating the front panel design
- iv. Creating toolpaths
- v. Loading the tool magazine and assigning tools to holder positions
- vi. Starting processing

4.1 Starting the machine and CircuitPro

- Starting the machine and CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed:

Fig. 135:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to its reference points and subsequently moves to the Pause position.

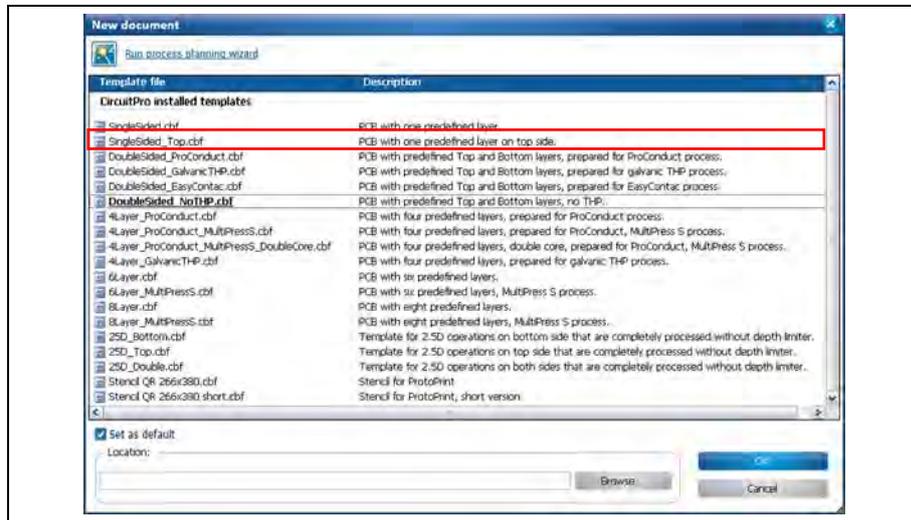
4.2 Selecting a template and creating a new document

■ Selecting a template and creating a new document

1. Click on File > New.

➔ The following dialog is displayed:

Fig. 136: New document



2. Select the template "SingleSided_Top.cbf".

3. Click on [OK].

4. Click on File > Save as...

5. Enter a new name for the file.

6. Select the storage location.

7. Click on [Save].

◆ The new document was created.

4.3 Creating the front panel design

To create the front panel design, following steps are required:

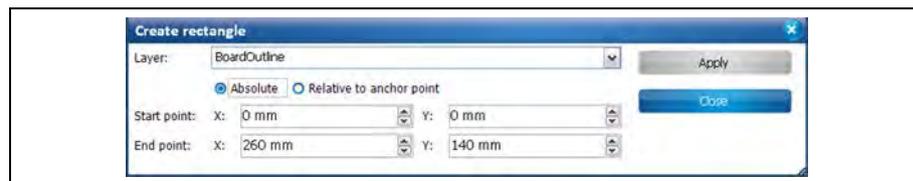
- i. Defining the dimensions of the front panel
- ii. Inserting cutouts for switches
- iii. Inserting holes for LEDs
- iv. Inserting markings
- v. Creating legends
- vi. Creating holes for mounting the front panel
- vii. Creating cutouts for a fan

■ Defining the dimensions of the front panel

The dimensions of the front panel are defined by inserting a rectangle on the “BoardOutline” layer with the dimensions of 260 mm x 140 mm.

1. Click on Insert > Rectangle.
- ➔ Following dialog is displayed:

Fig. 137: Create rectangle



2. Select the layer BoardOutline.
3. Ensure that {Absolute} is selected.
4. Enter “0” into the \Start point X\ and \Start point Y\ fields.
5. Enter “260” into the \End point X\ field.
6. Enter “140” into the \End point Y\ field.
7. Click on [Create].

➔ The rectangle is created.

8. Click on [Close].

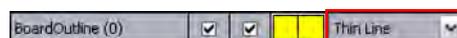
➔ The dialog is closed.

◆ The dimensions of the front panel are defined.



Note

In order to see the drawn objects in the CAM view that lie within the board outline, set the mode of layer “BoardOutline” in the “Layers” pane to “Thin Line”:



Note

The lower left corner of the board outline is now on the zero point. Thus, you can place all the subsequent objects at specific positions in relation to the board outline.

If you have the need to place objects in relation to a specific object, move the zero point as follows:

1. Click on the lower left corner of the reference object. The anchor point moves to this position.
2. Click on Edit > Set zero point. The zero point is moved to the anchor point and thus to the lower left corner.

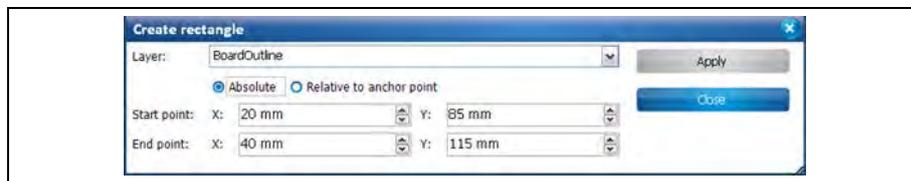
■ Inserting cutouts for switches

Two cutouts are needed for switches.

1. Click on Insert > Rectangle.

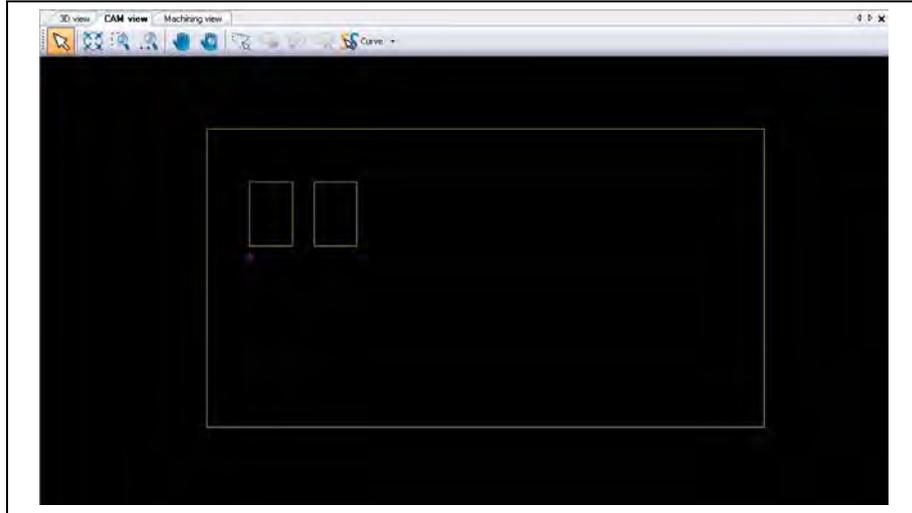
➔ Following dialog is displayed:

Fig. 138: Create rectangle



2. Select the layer "BoardOutline".
3. Select {Absolute} (thus, the start and end point values are in relation to the zero point).
4. Enter "20" into the \Start point X\ field.
5. Enter "85" into the \Start point Y\ field.
6. Enter "40" into the \End point X\ field.
7. Enter "115" into the \End point Y\ field.
8. Click on [Create].
- ➔ The rectangle is created.
9. Repeat the steps above to create another rectangle with a start point of X=50 and Y=85 and an end point of X=70 and Y=115.
- ➔ The second rectangle is created.
10. Click on [Close].
- ➔ Both rectangles are displayed in the CAM View:

Fig. 139:
Rectangles in the
CAM view



Note

The objects have to be converted into closed paths to enable CircuitPro to generate the contour routing toolpaths in the Technology dialog.



Tip

Converting the polygons into closed paths

1. Select the rectangles that you have created.
2. Click on Modify > Convert to path
→ The rectangles are converted to closed paths.

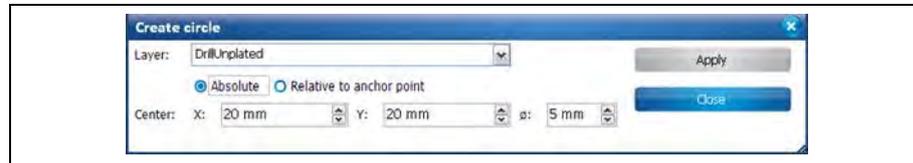
- ◆ The cutouts for the switches were inserted.

■ Inserting holes for LEDs

1. Click on Insert > Circle.

➔ Following dialog is displayed:

Fig. 140: Create circle



2. Select the layer "DrillUnplated".

3. Select {Absolute}.

4. Enter "20" into the \Center X\ field.

5. Enter "20" into the \Center Y\ field.

6. Enter "5" into the \Center ϕ\ diameter field.

7. Click on [Apply] to create the circle.

➔ The circle is created.

8. Click on [Close].

➔ The dialog is closed.

9. Select the drill hole to be multiplied.

10. Click on Modify > Step & Repeat.

11. Enter "9" into the \Repetition X\ field.

12. Enter "7" into the \Distance X\ field.

13. Click on [Apply] to multiply the drill hole.

➔ The drill holes were multiplied.

Another series of drill holes is needed, so that we have to insert circles again:

1. Click on Insert > Circle.

2. Select the layer "DrillUnplated".

3. Select {Absolute}.

4. Enter "105" into the \Center X\ field.

5. Enter "30" into the \Center Y\ field.

6. Enter "5" into the \Center ϕ\ diameter field.

7. Click on [Apply] to create the circle.

➔ The circle is created.

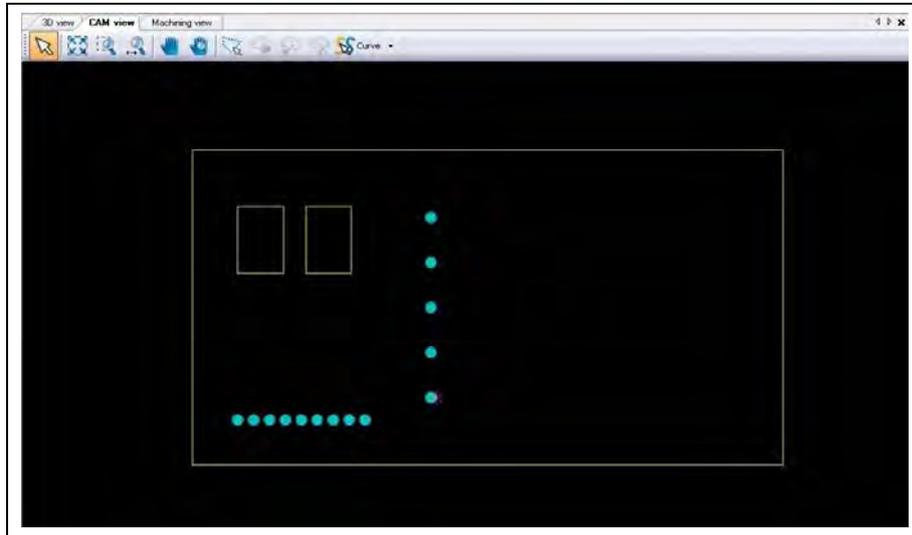
8. Click on [Close].

➔ The dialog is closed.

9. Multiply the drill hole in Y direction by using the “Step & Repeat” function.
10. Enter “5” into the \Repetition Y\ field.
11. Enter “20” into the \Distance Y\ field.
12. Click on [Apply] to multiply the drill holes in Y direction.

➔ The design now looks as follows:

Fig. 141: Drill holes



- ◆ The holes for the LEDs were inserted.

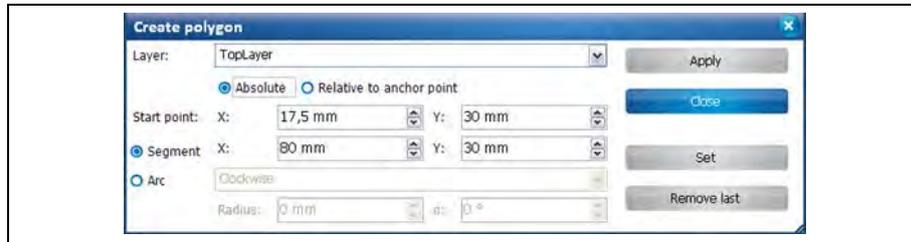
■ Inserting markings

An indication of the charge status is to be created above the lower row of LEDs.

1. Click on Insert > Polygon.

➔ Following dialog is displayed:

Fig. 142: Create Polygon



2. Select the layer "Top Layer".

3. Select {Absolute}.

4. Enter "17.5" into the \Start point X\ field.

5. Enter "30" into the \Start point Y\ field.

6. Select {Segment}.

7. Enter "80" into the \X\ field.

8. Enter "30" into the \Y\ field.

9. Click on [Set] to create the segment.

10. For the next segment, enter "80" into the \End point X\ field.

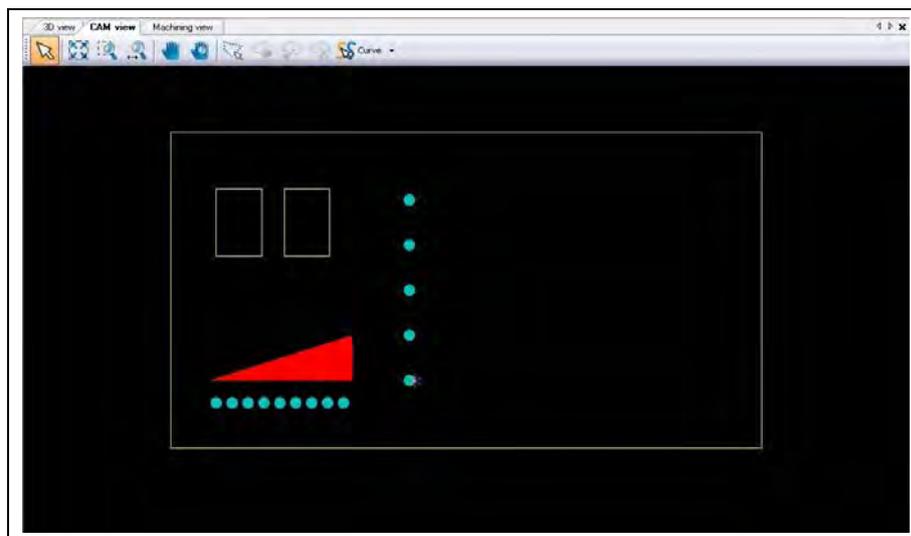
11. Enter "50" into the \End point Y\ field.

12. Click on "Set" to create the second segment.

13. Click on "Apply" to create the polygon.

➔ CircuitPro automatically connects the end of the last segment with the start of the first segment:

Fig. 143: Polygon



◆ The polygon is created.

- Creating circular marks around the drill holes
 1. Click on Insert > Circle.
 2. Select the layer "Top Layer".
 3. Select {Absolute}.
 4. Enter "105" into the \Center X\ field.
 5. Enter "110" into the \Center Y\ field.
 6. Enter "15" into the \Center \varnothing \ diameter field.
 7. Click on [Apply] to create the circle.

➔ The circle is created.

 8. Click on [Close].

➔ The dialog is closed.

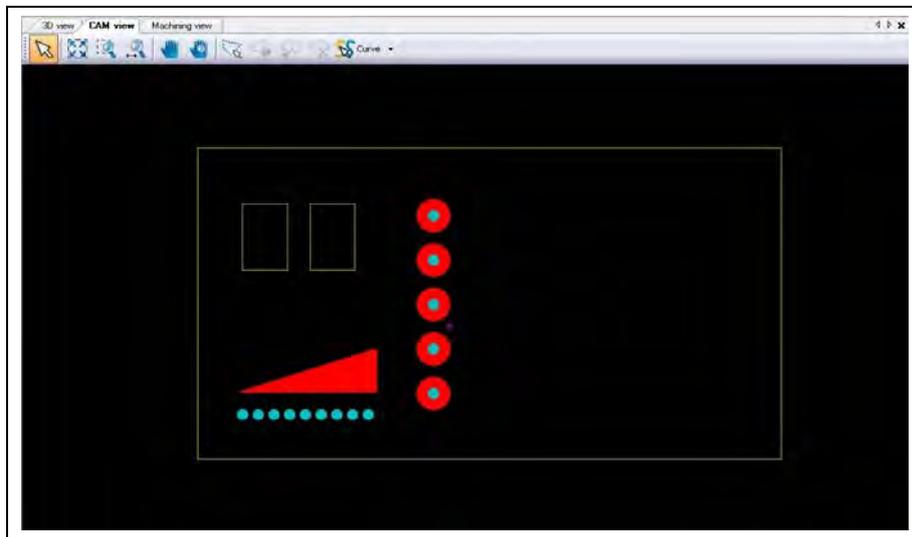
 9. Multiply the circles created in Y direction by using the "Step & Repeat" function.
 10. Enter "5" into the \Repetition Y\ field.
 11. Enter "-20" into the \Distance Y\ field.
 12. Click on [Apply] to multiply the circles in Y direction.

➔ The circle is multiplied in Y direction.

 13. Click on [Close].

➔ The dialog is closed and the design now looks as follows:

Fig. 144: Circular marks



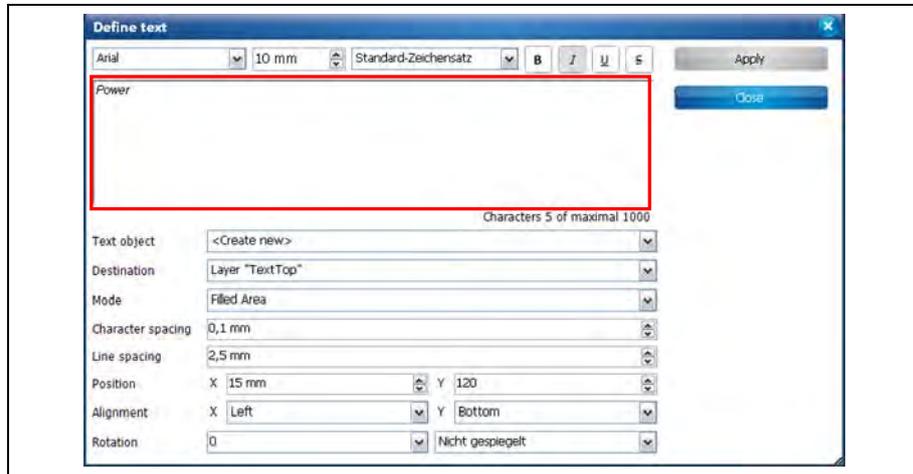
- ◆ The circular marks are created around the drill holes.

■ Creating legends

1. Click on Insert > Text.

➔ Following dialog is displayed:

Fig. 145: Define text



2. Enter the text “Power” in the red highlighted text field. 3. Select the layer “TextTop” in field \Destination\.

4. Modify the text attributes if necessary (in this example size=6 mm, bold, italic).

5. Enter “15” into the \Position X\ field.

6. Enter “120” into the \Position Y\ field.

7. Click on [Apply] to create the text object.

➔ The text object is displayed in the CAM view.

8. Click on [Close].

➔ The dialog is closed.

9. Click on an empty space in the CAM view.

➔ The text object is no longer selected.

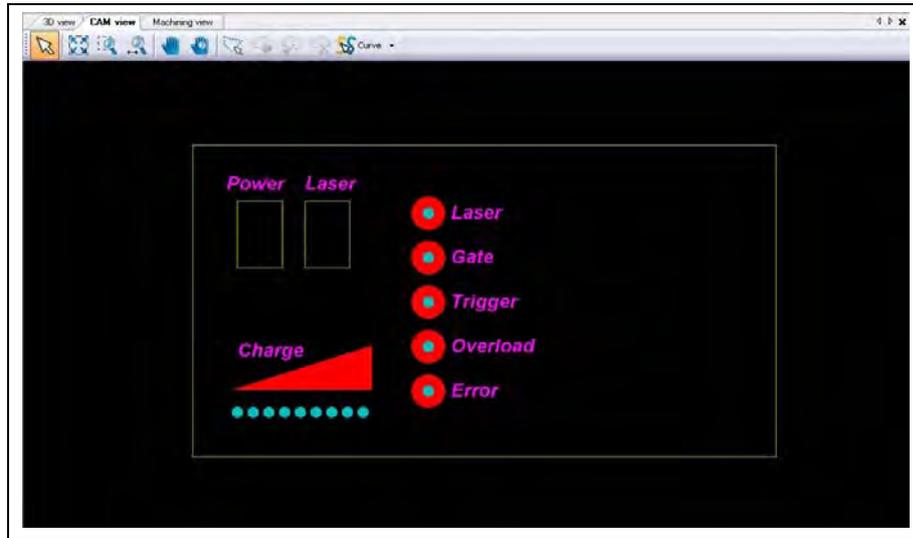
10. Repeat the steps above to create other legends using the values listed in the following table:

Table 1: Values for text legends

X position	Y position	Text
50	120	Laser
20	45	Charge
115	107	Laser
115	87	Gate
115	67	Trigger
115	47	Overload
115	27	Error

➔ The design now looks as follows:

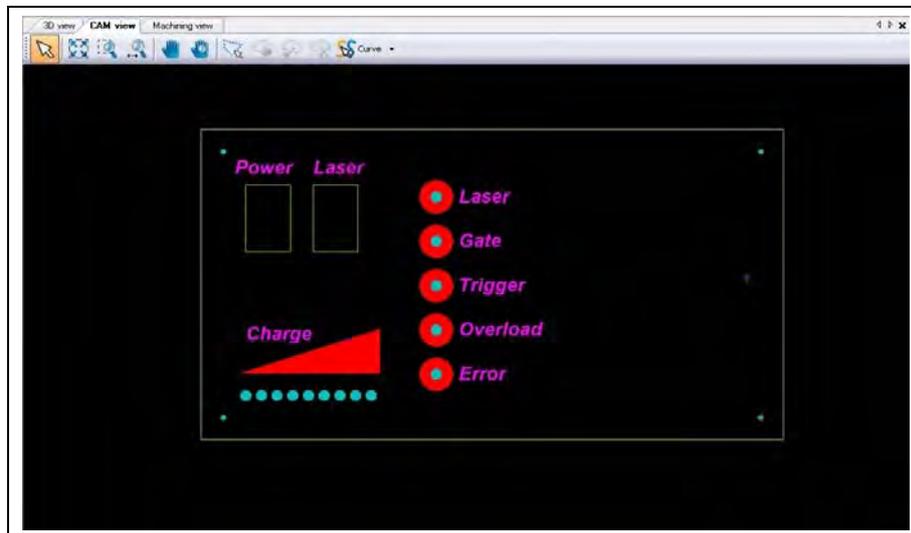
Fig. 146: Text in CAM view



◆ The legends were created.

- Creating holes for mounting the front panel
 1. Click on Insert > Circle.
 2. Select the layer "DrillUnplated".
 3. Select {Absolute}.
 4. Enter "10" into the \Center X\ field.
 5. Enter "10" into the \Center Y\ field.
 6. Enter "2.4" into the \Center \varnothing \ diameter field.
 7. Click on "Apply" to create the circle.
 8. Create another three circles using the following center points:
 - X=10; Y=130
 - X=250; Y=130
 - X=250; Y=10
- ➔ The new design now looks as follows:

Fig. 147: Holes in CAM view



- ◆ The holes for mounting the front panel were created.

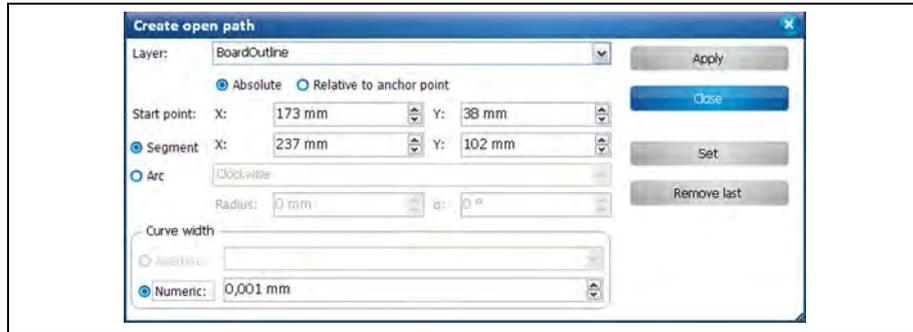
■ Creating cutouts for a fan

This task is to create cutouts for a fan. The required objects are created by using the function “Open path”.

1. Click on Insert > Open Path.

➔ Following dialog is displayed:

Fig. 148: Create closed path



2. Select the Layer “BoardOutline”.

→ Thus, contour routing toolpaths are generated for the cutouts in the technology dialog.

3. Select {Absolute}.

4. Enter “173” into the \Start point X\ field.

5. Enter “38” into the \Start point Y\ field.

6. Enter “237” into the \Segment X\ field.

7. Enter “102” into the \Segment Y\ field.

8. Click on [Apply] to create the path.

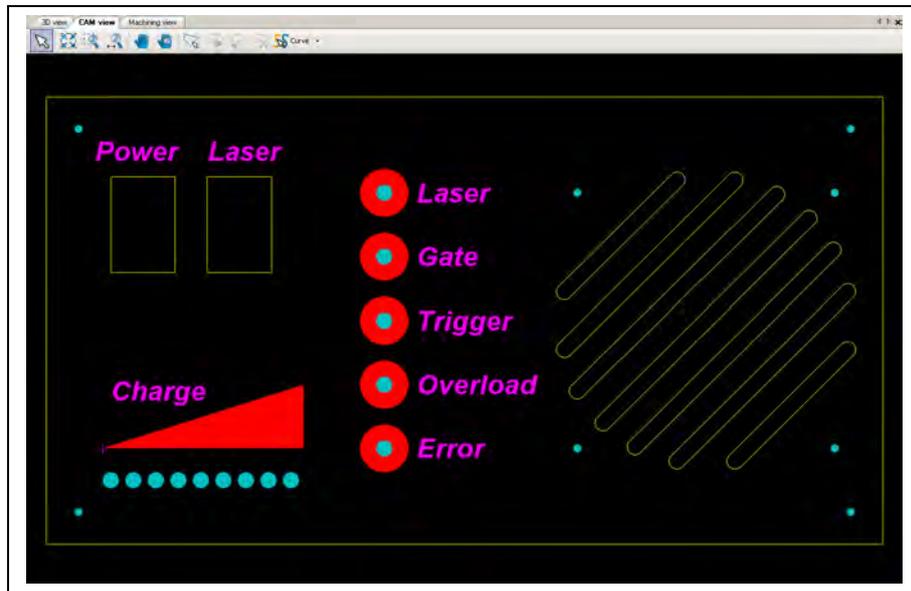
9. Repeat the steps 4-8 to create six more paths using the following values (the setting “Absolute” remains unchanged for the individual paths):

Table 2: Values for creating arcs

Start point X	Start point Y	Segment X	Segment Y
165,5	48	227	109,5
161	61	214	114
161	79	196	114
183	30,5	244,5	92
196	26	249	79
214	26	249	61

➔ The new design now looks as follows:

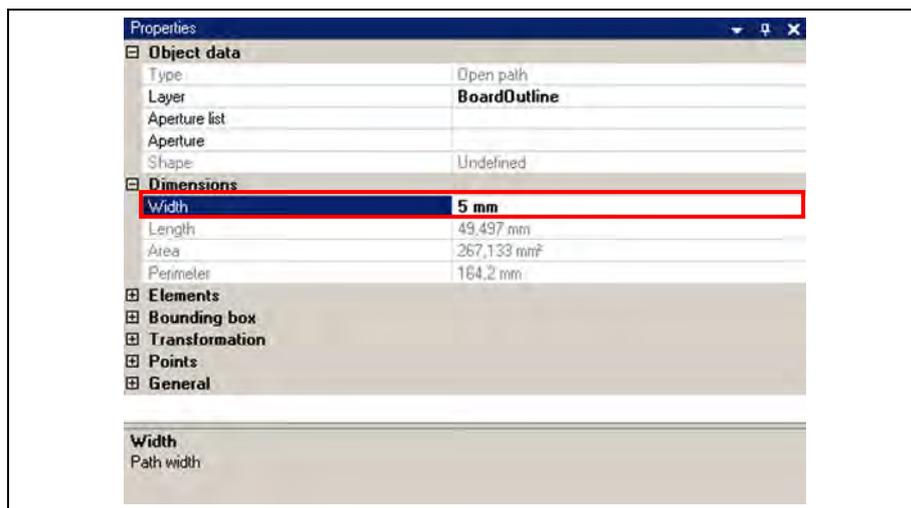
Fig. 149: Cutouts for the fan



Now the width of the open paths has to be changed to create the cutouts.

10. Select one of the created paths.
11. Click in the pane "Properties".
12. In the pane "Properties" select the field \Width\.
13. Overwrite the value by entering 5 mm.
14. Press "Enter" on your keyboard.

Fig. 150: Pane „Properties“



➔ The line width has changed.



Note

The line width of open or closed paths gets visible if switching to mode "Outline" in the pane "Layers". If mode "Thin Line" is selected, the open or closed path is always shown as a thin line.

15. Repeat the steps 10-14 to change the line width of the remaining six open paths to 5 mm.

◆ The cutouts for the fan were created.

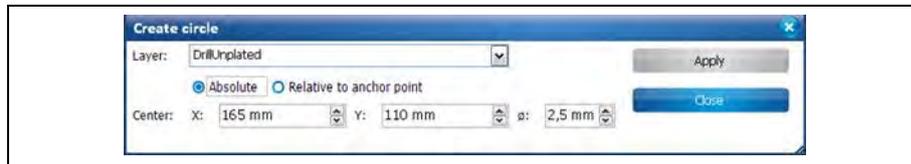
■ Creating holes for mounting the fan

This task is to create holes for mounting the fan. The holes are created using the function „Create circle“.

1. Click on Insert > Circle.

➔ The following dialog is displayed:

Fig. 151: Create Circle



2. Select the layer „DrillingUnplated“.

3. Select {Absolute}.

4. Enter „165“ into the \Center X\ field.

5. Enter „110“ into the \Center Y\ field.

6. Enter „2,5“ into the \ø\ field.

7. Click on [Apply] to create the circle.

➔ The circle was created.

8. Repeat the steps 4-7 using the following values to create three more circles:

Table 3: Values for the circles

X Position	Y Position	Durchmesser
165	30	2,5
245	110	2,5
245	30	2,5

◆ The holes were created.

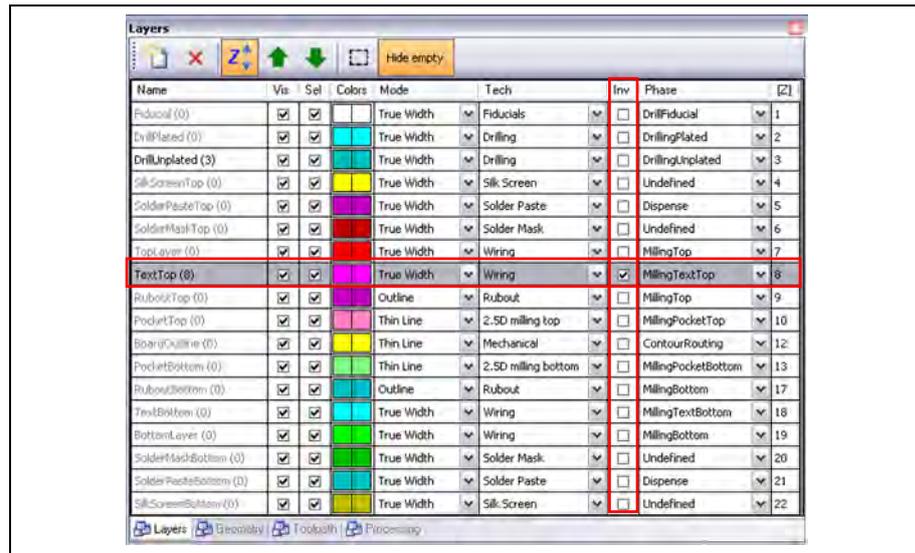
4.4 Creating toolpaths

Before we start creating the toolpaths, we have to create filled characters instead of outlined characters.

If the characters of the legends are to be milled fully, the settings of the layer have to be modified.

In the pane “Layers” check the check box in the “Inv” column in the row of the “TextTop” layer:

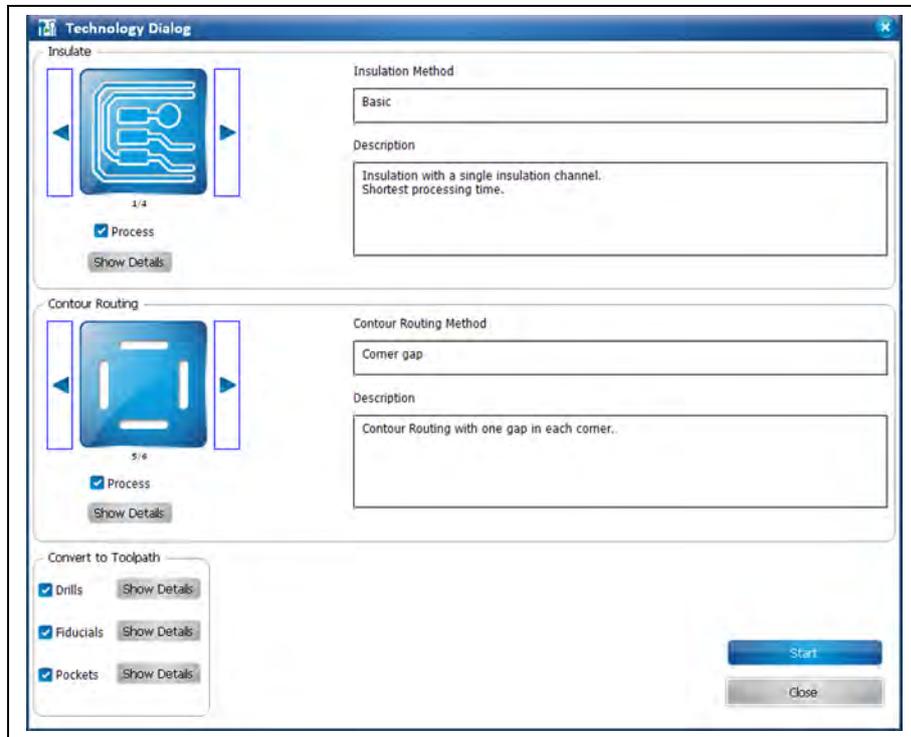
Fig. 152: Pane “Layers”



This means for a “Wiring” layer (see column “Tech”) that the inside of the object is filled with milling toolpaths instead of creating an insulating toolpath around the object when the toolpaths are generated.

- Creating toolpaths
1. Click on Toolpath > Technolog Dialog.
- ➔ The following dialog is displayed:

Fig. 153:
Technology
Dialog



Note

In the technology dialog, several settings can be modified by clicking on the [Show details] buttons.

For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

This example just requires the “Basic” contour routing method.

2. Click on the left-pointing arrow button in the “Contour Routing” section until the “Basic” contour routing method is selected.
 3. Click on [Start].
- ➔ The results of the toolpath generation are displayed.

4. Click on [Close].

➔ The dialog is closed.

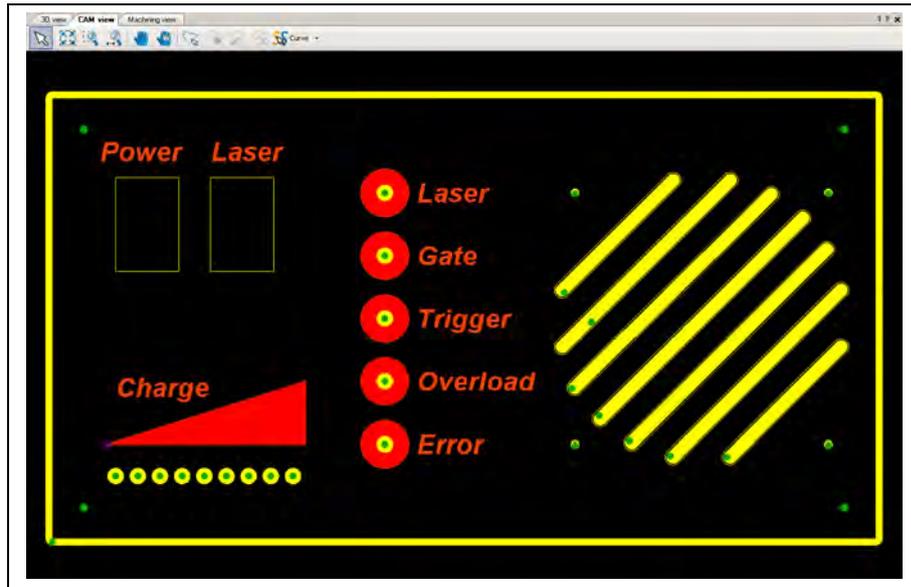


Note

CircuitPro processes circular cutouts up to a diameter of 2.4 mm as drill holes. All larger cutouts are processed as inner contours and are cut using the contour router tool.

➔ The design now looks as follows:

Fig. 154: Created toolpaths



◆ The toolpaths were created

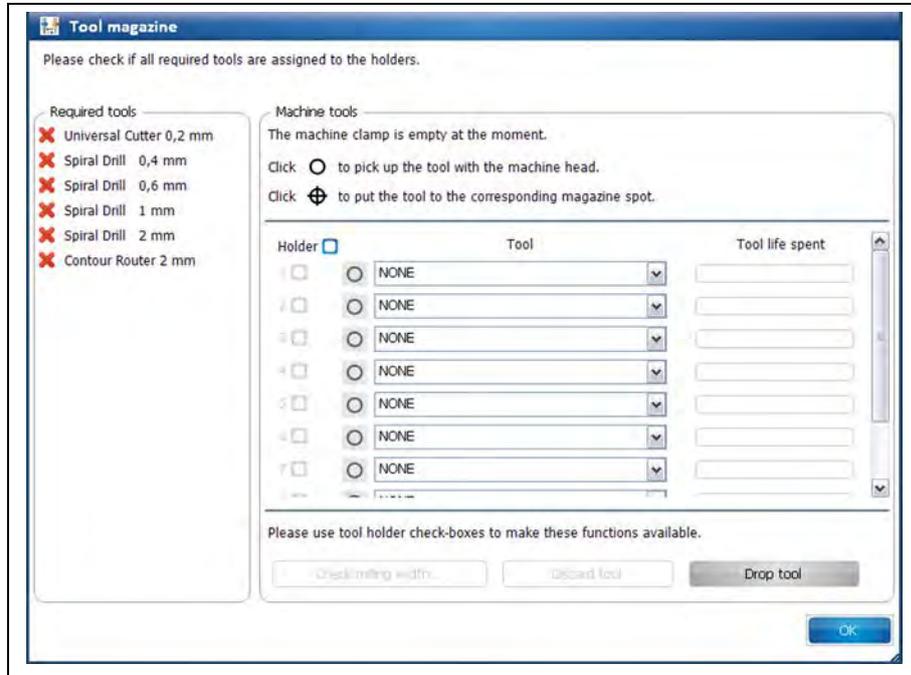
4.5 Loading the tool magazine and assigning tools to holder positions

- Loading the tool magazine and assigning the tools to holder positions

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

Fig. 155: Tool magazine



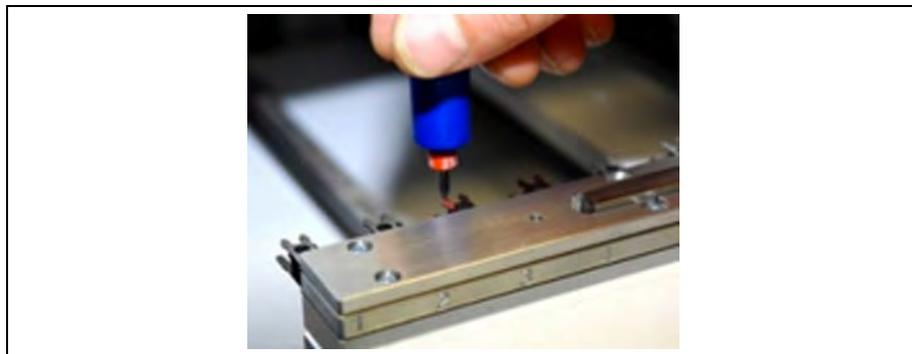
Note

The tools shown in the tool magazine dialog must not correspond to your needed tools. These tools are examples.

➔ The tools required for the job are displayed. Tools that are missing in the tool magazine are marked by a red "X".

2. Insert the required tools into the tool magazine:

Fig. 156: Inserting a tool



3. In the dialog, assign the tools to the respective tool magazine positions used.

➔ The tool holders of the machine are loaded:

Fig. 157: Loaded tool holder



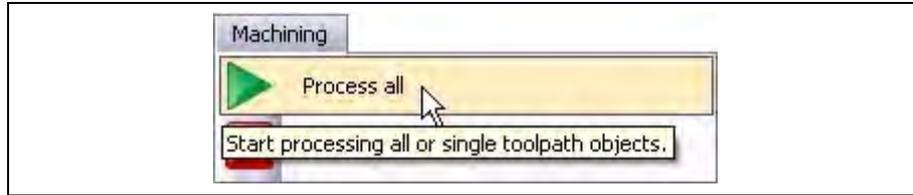
4. Click on [OK].
- ➔ The dialog is closed.
- ◆ The tools were assigned and the tool magazine was loaded.

4.6 Start processing

■ Start processing

1. Click on Machining > Process all.

Fig. 158:
Machining >
Process all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

Once you have started the processing, the ProtoMat machine executes the job in individual phases. The phases are displayed in messages:



Note

Depending on which ProtoMat you use the following phases could differ from the phases and messages displayed on your screen. Please follow the instructions on your screen.

For machines with manual tool exchange you are regularly asked to change the tool in the collet, for example.

Phase "Mount Material"

1. Mount the material onto the processing area.
2. Fasten the material to the processing area using adhesive tape.
3. Click on [OK].

Phase "Material Settings"

➔ The following dialog is displayed:

Fig. 159: Material settings

■ Entering the material settings

1. Enter the correct values for the material used.

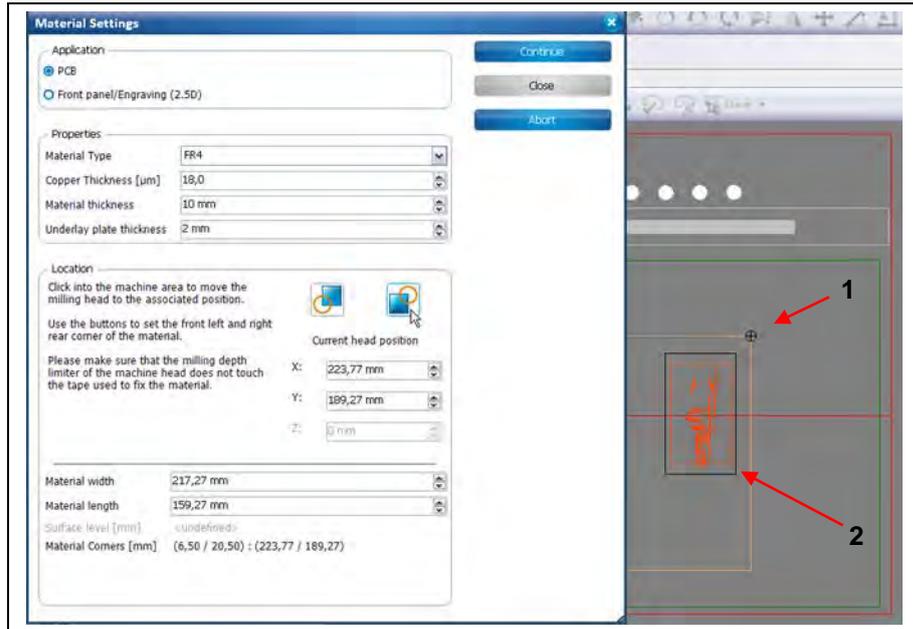


PCB is selected as default.

Note

2. Change the values of copper thickness and material thickness necessary.
3. Define the processing area:
 - a) Move the dialog „Material Settings“ off to the side.
 - b) Using your mouse in the machining view, click on the right corner of your material:

Fig. 160: Right corner



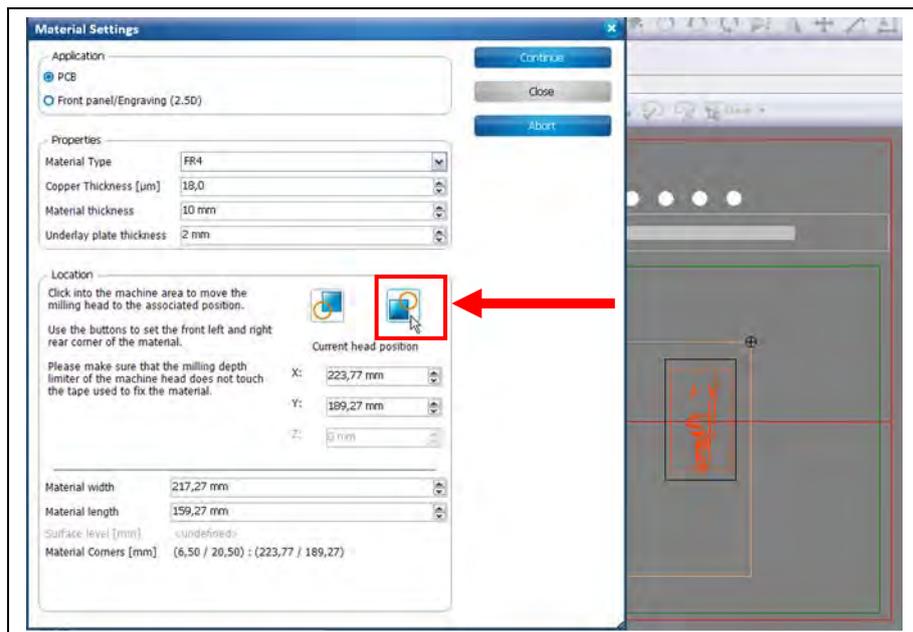
/1/ Click in the machining view

/2/ Material

➔ The machine head moves to this position.

c) Now click on the corresponding button in the dialog "Material Settings":

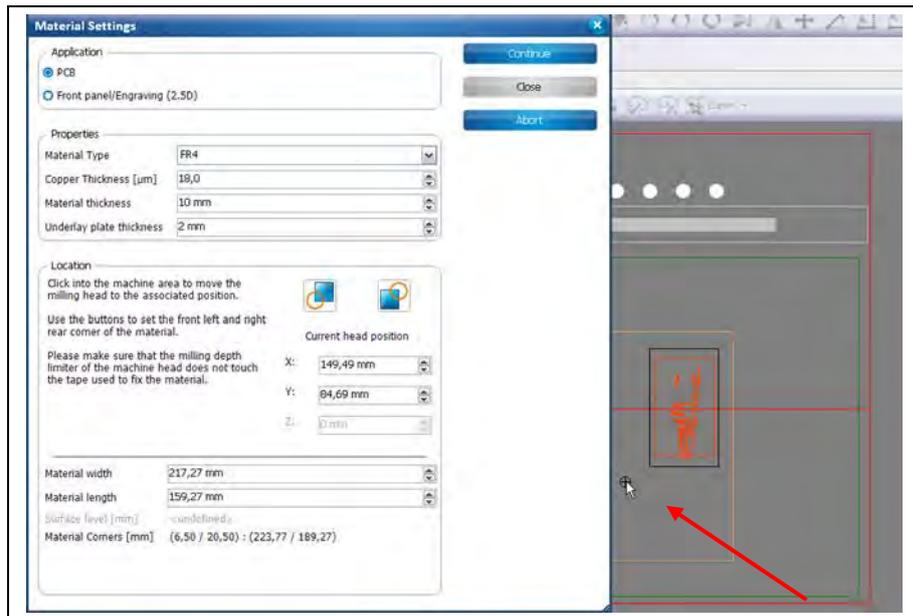
Fig. 161: Click on button



➔ The processing area is adapted.

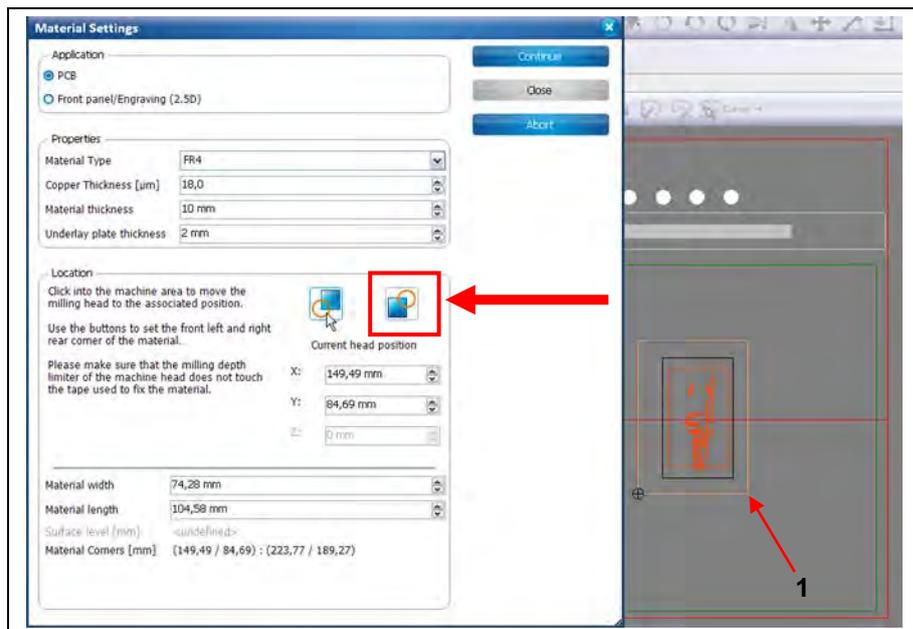
d) Using your mouse in the machining view, click on the left corner of your material:

Fig. 162: Left corner



- ➔ The machine head moves to this position.
- e) Now click on the corresponding button in the dialog "Material Settings":

Fig. 163: Defined processing area



/1/ Defined processing area

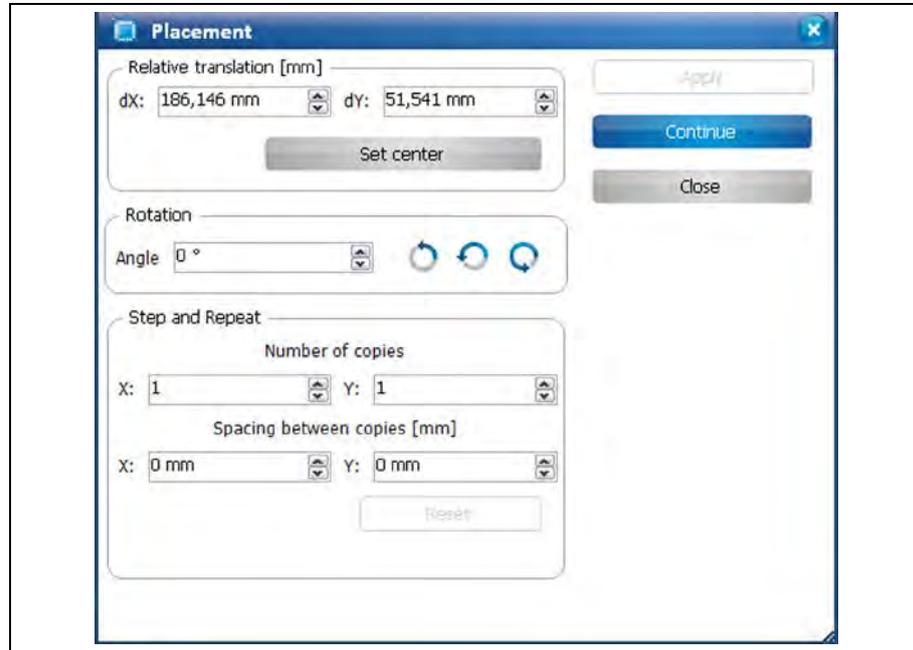
- ➔ The processing area was adapted to the material.
- 4. Click on [Continue].
- ◆ The material settings were entered.

Phase “Placement”

In this phase, the job can be positioned, rotated and multiplied within the processing area.

➔ Following dialog is displayed:

Fig. 164:
Placement



1. Drag the job to the desired position or use the dialog to position the job.
2. If desired, rotate the job by entering a rotation angle.
3. If desired, multiply the job by using the settings in the “Step and Repeat” section of the dialog.
4. Click on [Continue].

Phase “Marking Drills”

➔ The “Universal Cutter” tool is picked up and the drill positions are marked.

Phase “Drilling Unplated”

➔ The tools are picked up as required and the holes are drilled.

Phase “Milling Top”

➔ The tools are picked up as required and the material (Top side) is milled.

Phase “Contour Routing ”

➔ The tools are picked up as required and the material is drilled and routed.

Phase “Board Production Finished”

- ➔ A message informs you that the production is finished.
- ◆ Your front panel is created.

Creating a polyimide stencil

5 Creating a polyimide stencil

This tutorial shows you how to produce an SMT stencil made of polyimide that you can use in the later process stages to apply solder paste (using e.g. the LPKF ProtoPrint system).

The following steps are necessary to complete the tutorial successfully:

- i. Starting the machine and CircuitPro
- ii. Selecting a template and creating a new document
- iii. Importing data
- iv. Processing data
- v. Creating toolpaths
- vi. Loading the tool magazine and assigning tools to holder positions
- vii. Starting the processing



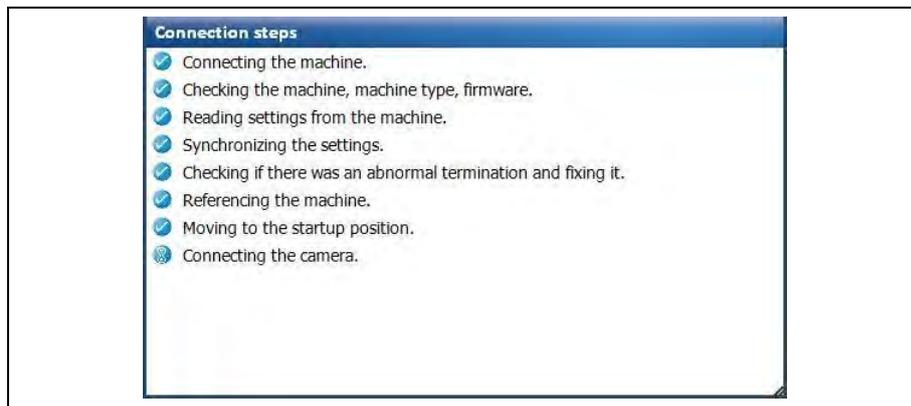
Note

In order to create a stencil, the polyimide film must be fixed on the underlay with a spray adhesive.

5.1 Starting the machine and CircuitPro

- Starting the machine and CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed:

Fig. 165:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to its reference points and subsequently moves to the Pause position.

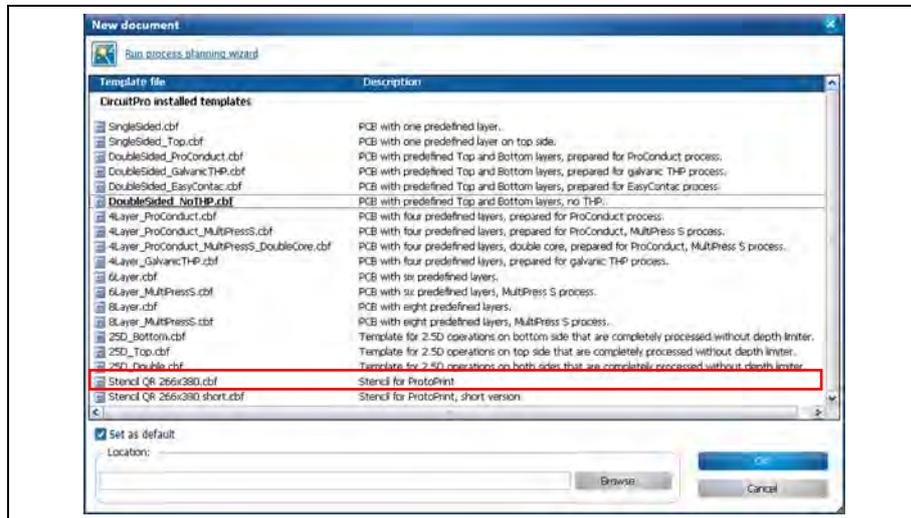
5.2 Selecting a template and creating a new document

■ Selecting a template and creating a new document

1. Click on File > New...

➔ The following dialog is displayed:

Fig. 166: New document



2. Select the template "Stencil QR 266x380.cbf".

3. Click on [OK].

4. Click on File > Save As...

5. Enter a file name for the new file.

6. Select a memory location.

7. Click on [Save].

◆ The new document is created.

5.3 Importing data



Tip

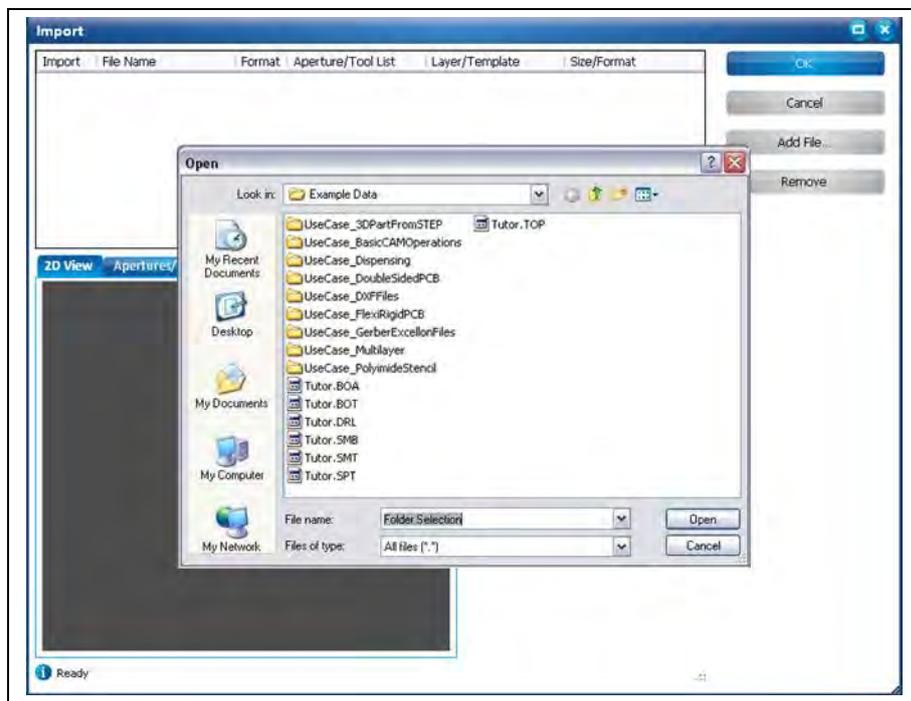
The LPKF tutor data are stored in folder “My documents\LPKF Laser & Electronics\LPKF CircuitPro 1.5\Example Data\UseCase_PolyimideStencil”.

■ Importing data

1. Click on File > Import...

➔ The following dialog is displayed:

Fig. 167: Import



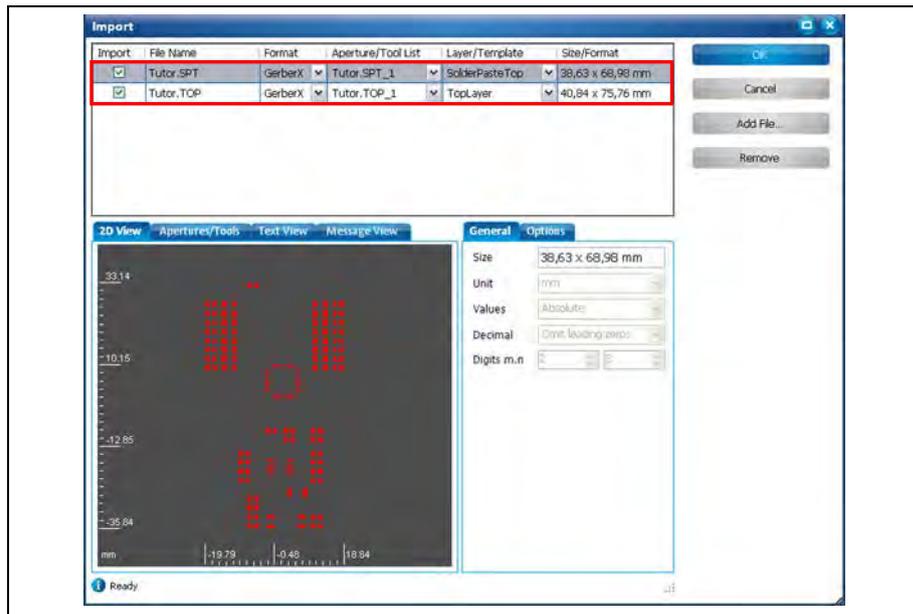
2. In the folder “UseCase_PolyimideStencil”, select the files “Tutor.SPT” and “Tutor.TOP”.

3. Click on [OK].

4. Assign the imported files according to the following table:

File extension	Layer
.SPT	SolderPasteTop
.TOP	Top Layer

Fig. 168: Assign layers



Note

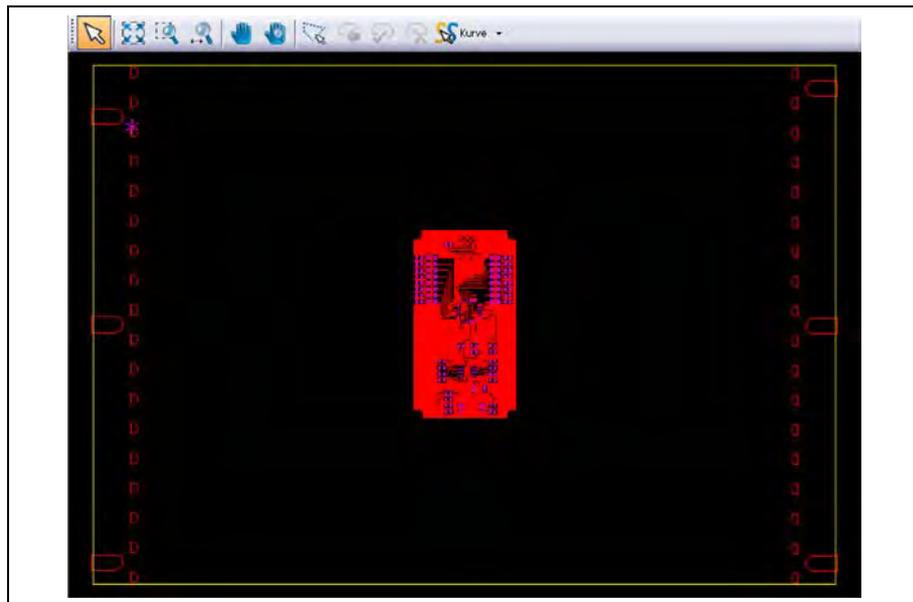
Instead of manually assigning the individual files to the layers, you can activate the options “Use layer name” and “Apply to all Gerber files”. Activate the corresponding checkboxes in the tab “Options”.

If a file contains layer names these are automatically assigned. Please note that this is only available for Gerber files. All other files require assigning the layers manually via the drop-down menu.

5. Click on [OK].

➔ The data are displayed in the CAM view:

Fig. 169: CAM view



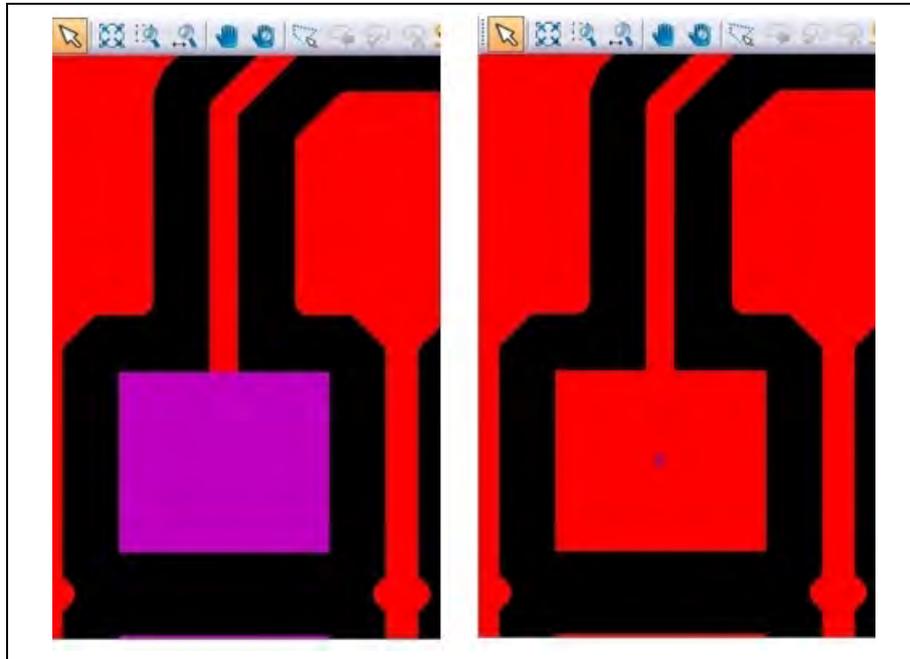
◆ The data is imported.

5.4 Processing data

In the CAM view you are able to check the proportion between the solder paste pads and the copper pads.

The imported data shows you, that both pads on the SolderPaste-Layer and on the Top-Layer have the same proportion:

Fig. 170: Identical proportion



In order to achieve better result when applying solder paste, LPKF recommends to minimize the pad dimension for the solder paste. This will prevent the solder to flow over the copper pads.

- Minimizing the pad dimension of the solder paste pads
- 1. Highlight the data on the layer "SolderPasteTop":
 - a) Click in the pane "Layer" on the layer "SolderPasteTop".
 - b) Now click in the pane "Layer" on the adjoining icon "Select objects on layer", to highlight all objects on the layer.
- 2. Click on Modify > Convert to polygon.



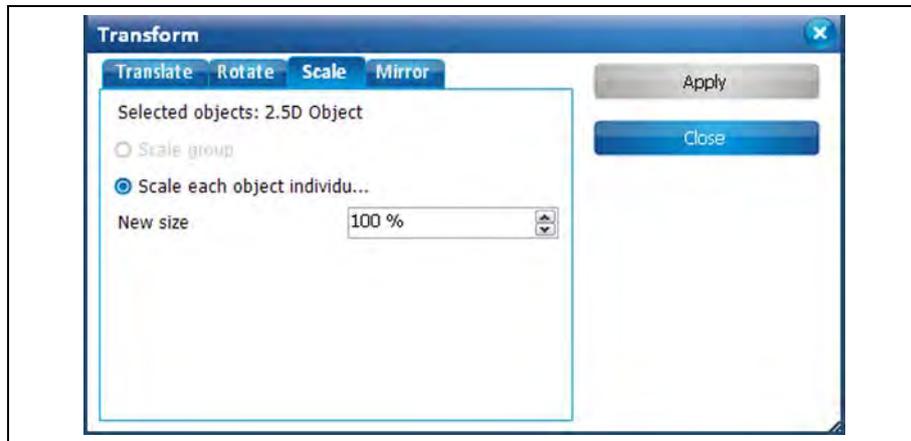
Note

The objects on the layer "SolderPasteTop" are flash objects. These cannot be scaled and must be converted into polygons first.

3. Click on Modify > Transformation.

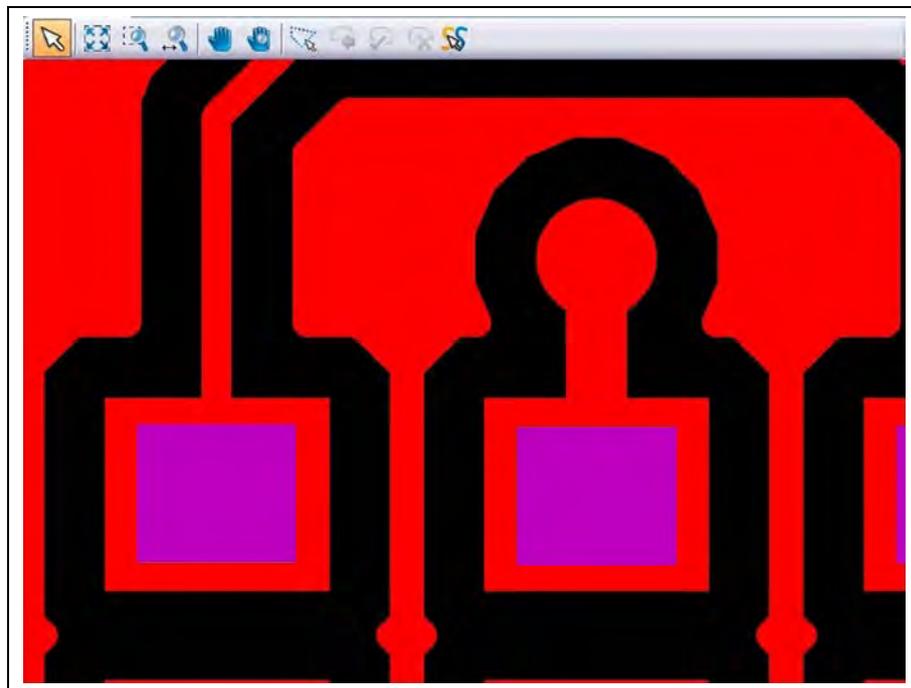
- ➔ The “Transformation” dialog is shown.
- 4. Select the tab “Scale”:

Fig. 171: Transformation >Scale



- 5. If not active, enable the option “Scale each object individually”.
- 6. Enter “70” in the \New size in percent\ field.
- 7. Click on [Apply].
- 8. Click on [Close].
- ➔ The dimension of the solder paste pads on the layer “SolderPasteTop” has changed:

Fig. 172: Dimensions changed



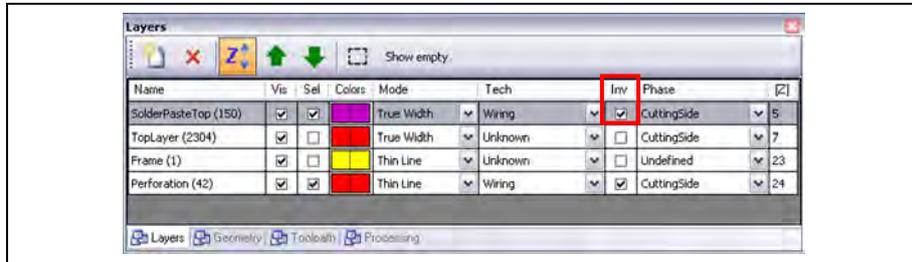
- ◆ The dimension of the solder paste pads is minimized.

5.5 Creating toolpaths

The objects of layer “SolderPasteTop” displayed in the CAM view require an inner insulation in order to remove the material inside these areas.

This requires that the check box in the “Inv” column of the “SolderPasteTop” row of the “Layers” pane is activated:

Fig. 173: “Layers” pane



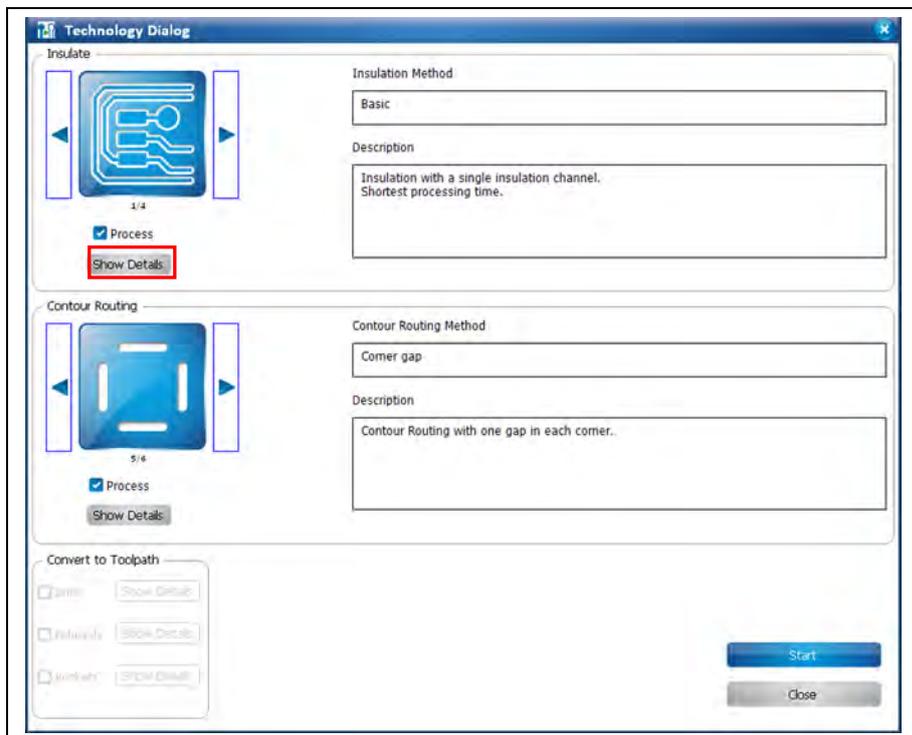
This means for a “Wiring” layer (see column “Tech”) that the inside of the object is filled with milling toolpaths instead of creating an insulating toolpath around the object when the toolpaths are generated.

■ Creating toolpaths

1. Click on Toolpath > Technology dialog...

➔ The following dialog is displayed:

Fig. 174: Technology Dialog





Note

In the technology dialog, several settings can be modified by clicking on the [Show details] buttons.

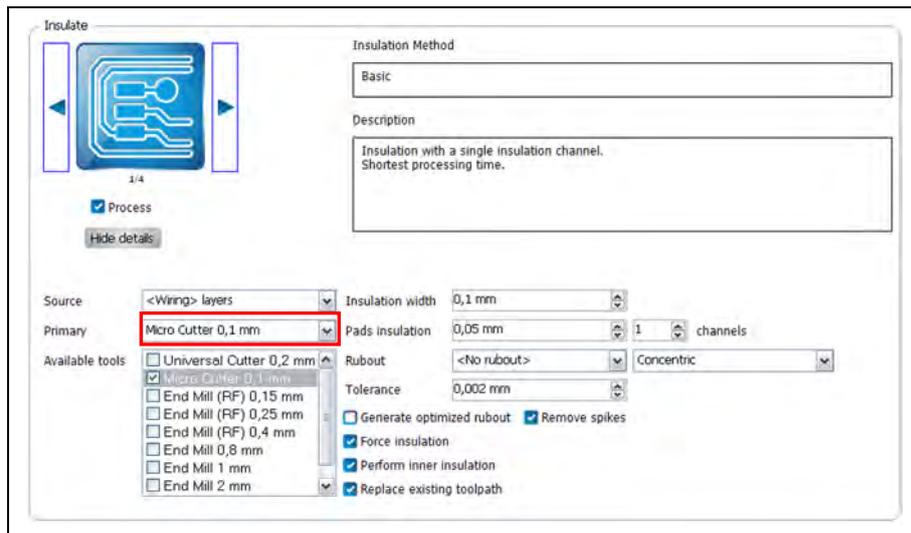
For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

This example just requires the “Basic” contour routing method.

2. Click on [Show Details] in the “Insulate” section.

➔ The details for insulation are displayed.

Fig. 175:
Insulation details



3. In the drop down list “Primary”, select the tool “Micro Cutter 0.1 mm



Note

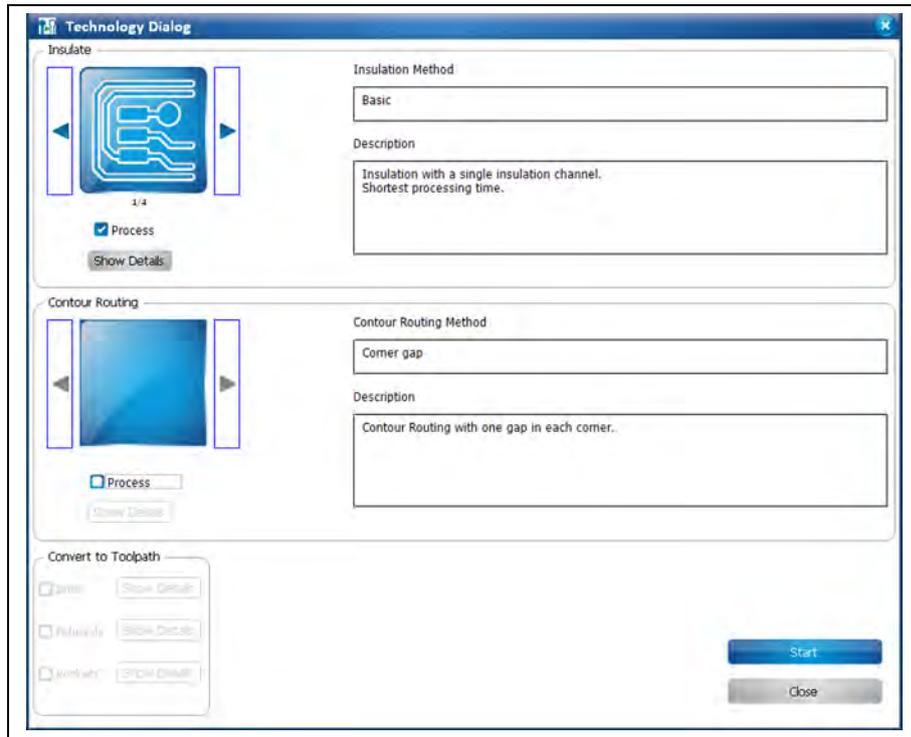
All other tools are not qualified for milling the polimide stencil. Deactivate all other tools in the drop down list „Available tools“.

4. Click on [Hide details].

➔ The details are hidden.

5. Deactivate all other functions of the technology dialog except insulation. To achieve this, click on the check marks next the individual sections:

Fig. 176:
Deactivating
functions



- ➔ The functions are deactivated.
6. Click on [Start].
- ➔ CircuitPro creates the toolpaths.
 - ➔ The dialog with the computation results is displayed.
7. Click on [Close].
- ➔ The dialog is closed.
 - ◆ The toolpaths are created.

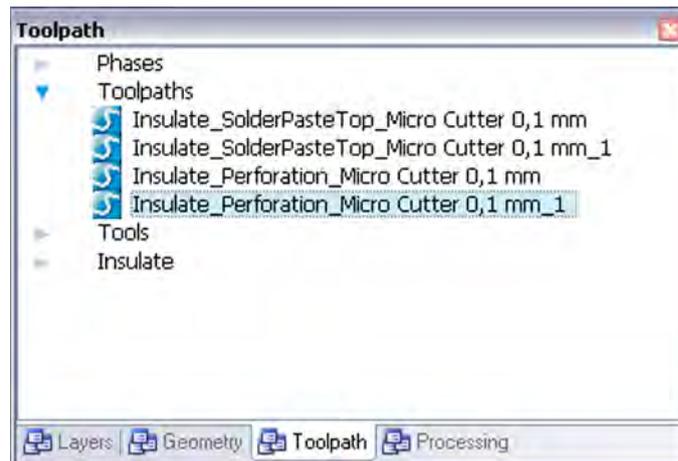


Note

CircuitPro automatically creates several toolpaths for an inner insulation. There are two distinct kinds of toolpaths:

- Toolpath for insulating the inside of the selected objects.
- Toolpath for removing the material inside the insulated area.

The toolpath that removes the material inside the insulated area is marked with “_1” in the “Toolpath” pane.



The material is cut through when insulating the objects for producing the stencil. Thus the toolpath for removing the material inside the insulation (Insulate_SolderPasteTop_Micro Cutter 0.1mm_1“) is not necessary and can be deleted.

Deleting a toolpath

1. Select the desired toolpath in the “Toolpath” pane.
2. Press the “Del” key or select “Delete” in the context menu.

5.6 Loading the tool magazine and assigning tools to holder positions



This chapter is relevant, if you use a ProtoMat with automatic tool change (S63 or S103).

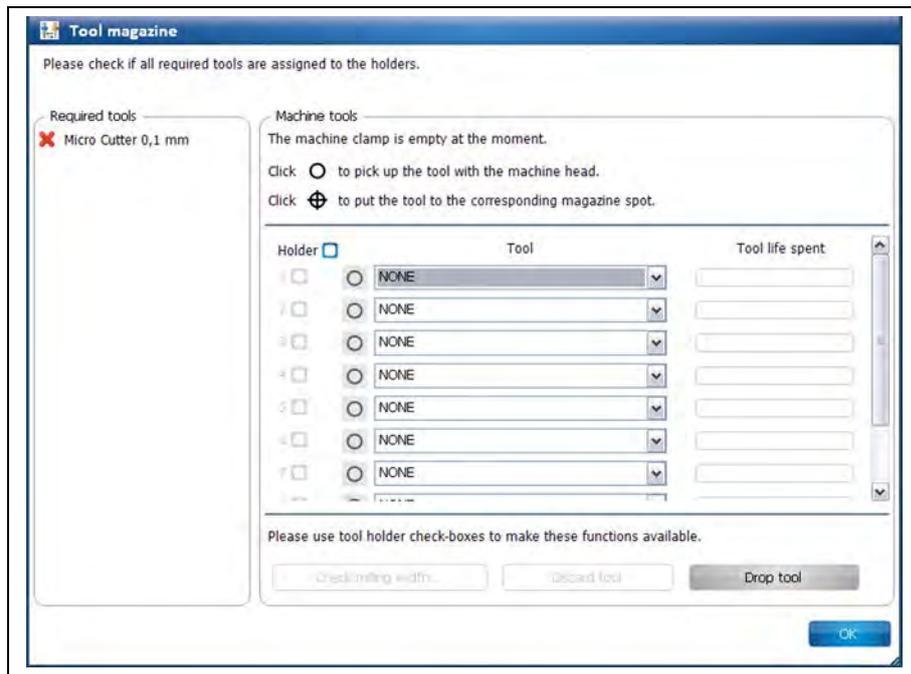
Note

■ Loading the tool magazine and assigning tools to holder positions

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

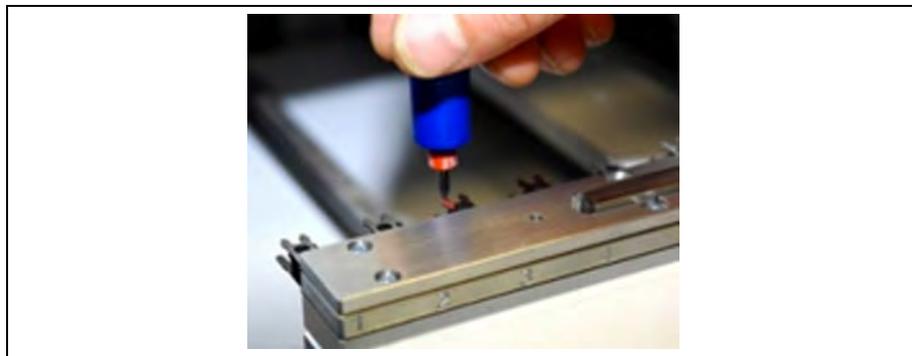
Fig. 177: Tool magazine



➔ The tools required for the job are displayed. Tools required for the job that are missing are marked by a red "X".

2. Insert the required tools into the tool holders of the machine:

Fig. 178: Insert tool



3. Assign the tools to the corresponding positions in the dialog.

➔ The tool holder in the machine is loaded:

Fig. 179: Loaded tool holder



4. Click on [OK].

➔ The dialog is closed.

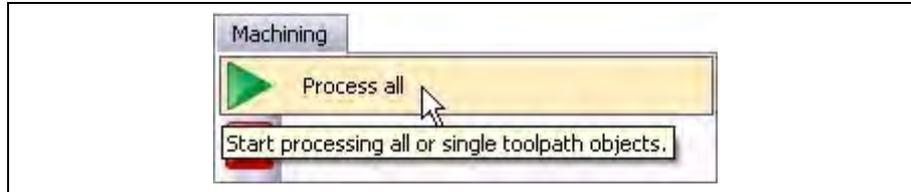
◆ The tools are inserted into the tool holders and assigned accordingly.

5.7 Starting the processing

■ Starting the processing

1. Click on Machining > Process all.

Fig. 180:
Machining >
Process all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

Once you have started the processing, the ProtoMat machine executes the job in individual phases. The phases are displayed in messages:



Note

Depending on which ProtoMat you use the following phases could differ from the phases and messages displayed on your screen. Please follow the instructions on your screen.

For machines with manual tool exchange you are regularly asked to change the tool in the collet, for example.

Phase "MountMaterial"

1. First spray adhesive (e.g. 3M type 75) onto the surface of the underlay.
2. Place the polyimide film centered on the underlay.
3. Press the polyimide film on the underlay.
4. Use a rubber roll to remove all air blisters:
5. Put the underlay on the working table using the dowel pins if a vacuum table is **not** assembled.



Or

5. Fasten the underlay on the vacuum table by using adhesive tape.
6. Click on [OK].

Phase "Material Settings"

➔ The following dialog is displayed:

Fig. 181: Material Settings

- Entering the material settings
 1. Enter the correct values for the material used.



PCB is selected as default.

Note

2. Enter the thickness of your material (polyimide film) in the field \Copper Thickness\
3. If a vacuum table is **not** assembled: Enter "0" in the field \Material Thickness\.

Or

3. If a vacuum table is assembled: Enter the thickness of your underlay in the field \Material Thickness\.

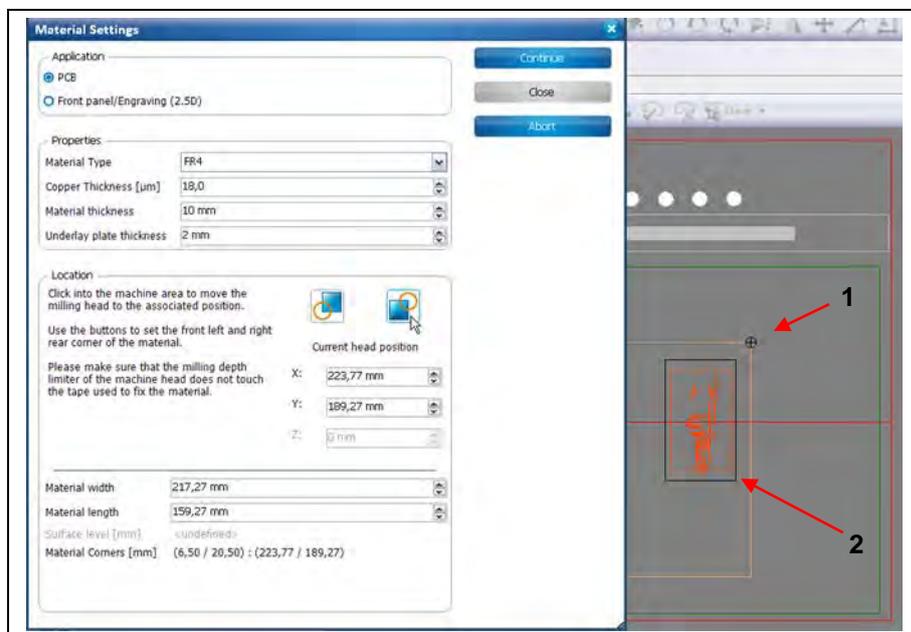


The polyimide film has no copper layer. Since the material is cut with an insulation tool, the thickness of the material (polyimide film) has to be entered in the field for the copper thickness.

Note

4. Define the processing area:
 - a) Move the dialog „Material Settings“ off to the side.
 - b) Using your mouse in the machining view, click on the right corner of your material:

Fig. 182: Right corner

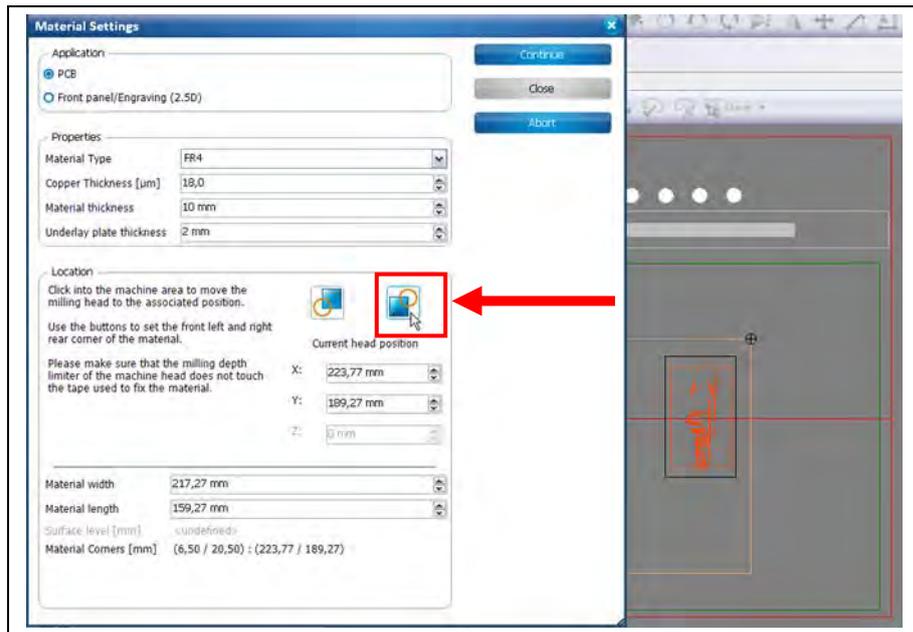


/1/ Click in the machining view

/2/ Material

- ➔ The machine head moves to this position.
 - c) Now click on the corresponding button in the dialog “Material Settings“:

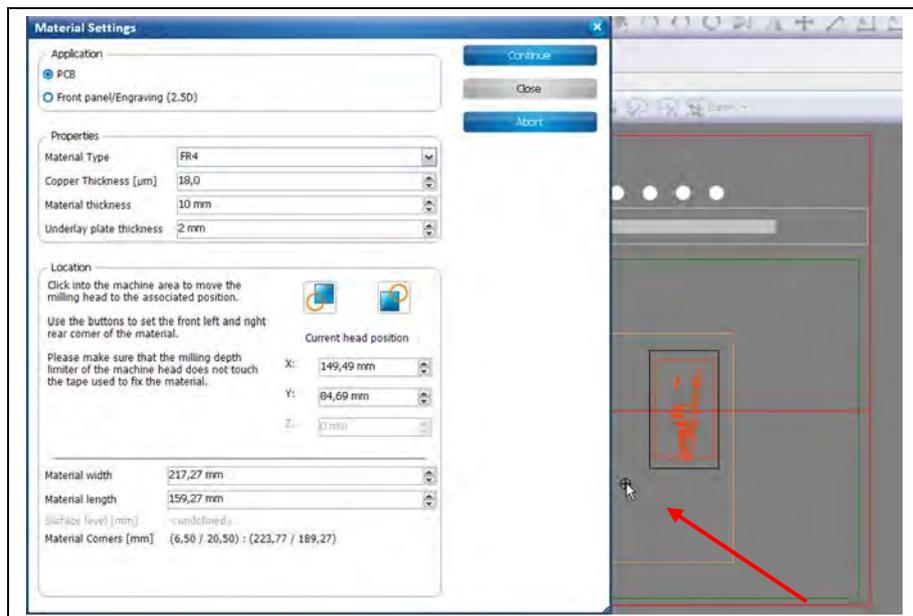
Fig. 183: Click on button



➔ The processing area is adapted.

d) Using your mouse in the machining view, click on the left corner of your material:

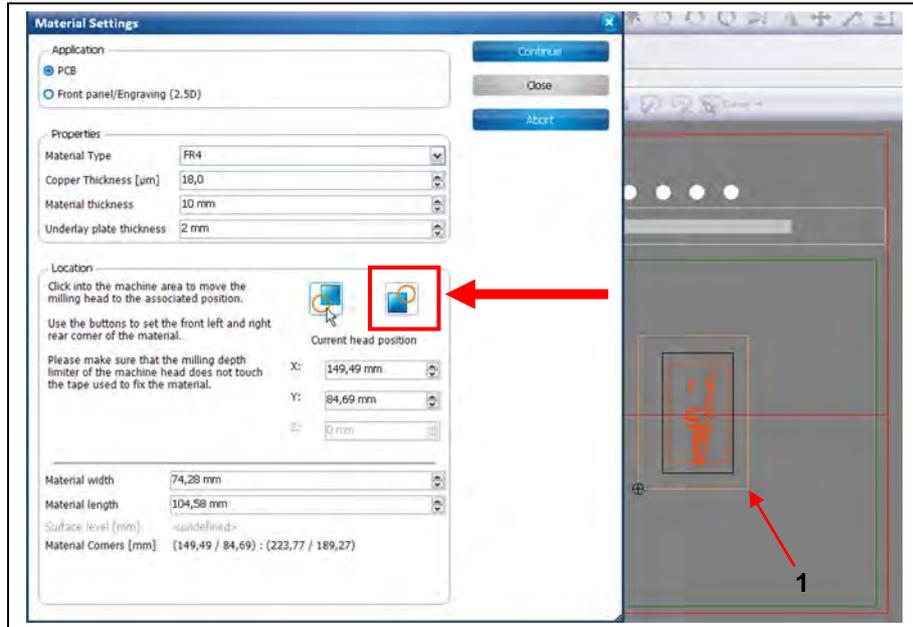
Fig. 184: Left corner



➔ The machine head moves to this position.

e) Now click on the corresponding button in the dialog "Material Settings":

Fig. 185: Defined processing area



/1/ Defined processing area

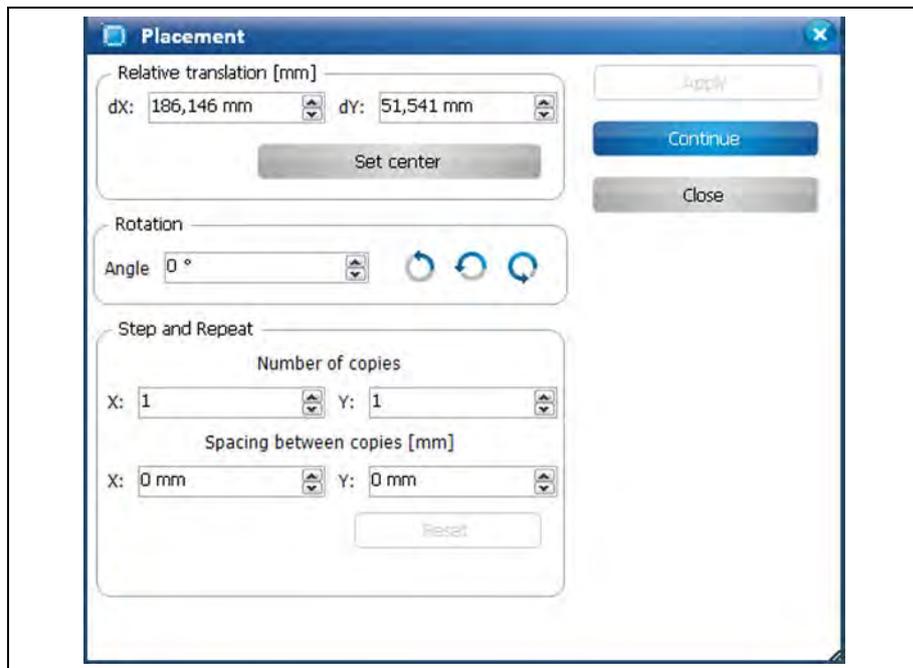
- ➔ The processing area was adapted to the material.
- 5. Click on [Continue].
- ◆ The material settings were entered.

Phase “Placement”

In this phase, the job can be placed arbitrarily on the base material and be multiplied if necessary.

- ➔ Following dialog is displayed:

Fig. 186: Placement



1. Click on the job and drag it to the desired position using the mouse.
- Or
2. Enter the new position in the dialog.



Note

At this point, the whole circuit board design including the template and the toolpaths possibly contained in the template are moved.

If only the design is to be moved within the template, you have to use the menu item Modify > Transform....

Phase "Milling Top"

- ➔ The Micro Cutter is picked up. The stencil is milled.

Phase "Board Production Finished"

- ➔ A message informs you that the production is finished.

1. Remove the stencil from the underlay:
2. Put the stencil on a table with the adhesive side facing upwards.
3. Take a lint-free cloth.
4. Moisten the cloth with rubbing alcohol.
5. Remove any remaining adhesive from the stencil.



- ◆ The polyimide stencil is created.



Tip

At a later stage of the process, you can clamp the stencil in an LPKF ProtoPrint and apply solder paste onto the circuit board with high precision using the stencil.

You can also cut the stencil to size for other solder paste processes.

Creating a flexi-rigid PCB

6 Creating a flexi-rigid PCB

This tutorial shows you how to produce a flexi-rigid circuit board.

The following steps are necessary to complete the tutorial successfully:

- i. Starting the machine and CircuitPro
- ii. Selecting a template and creating a new document
- iii. Importing data
- iv. Drilling and milling the individual physical layers
- v. Bonding the individual physical layers
- vi. Routing the bonded PCB



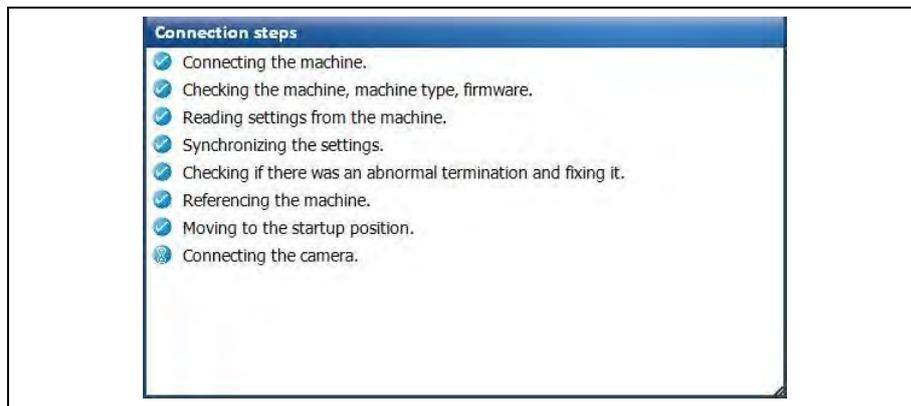
Note

Thin materials like the flexible polyimide material and the prepreg can only be machined using a vacuum table and air-cushioned milling-depth limiter.

6.1 Starting the machine and CircuitPro

- Starting the machine and CircuitPro
 1. Switch on the machine.
 2. Start CircuitPro.
- ➔ CircuitPro automatically connects to the machine. The connection steps are displayed:

Fig. 187:
Connection steps



- ➔ CircuitPro reads the settings from the machine.
- ◆ The machine moves to its reference points and subsequently moves to the Pause position.

6.2 Selecting a template and creating a new document

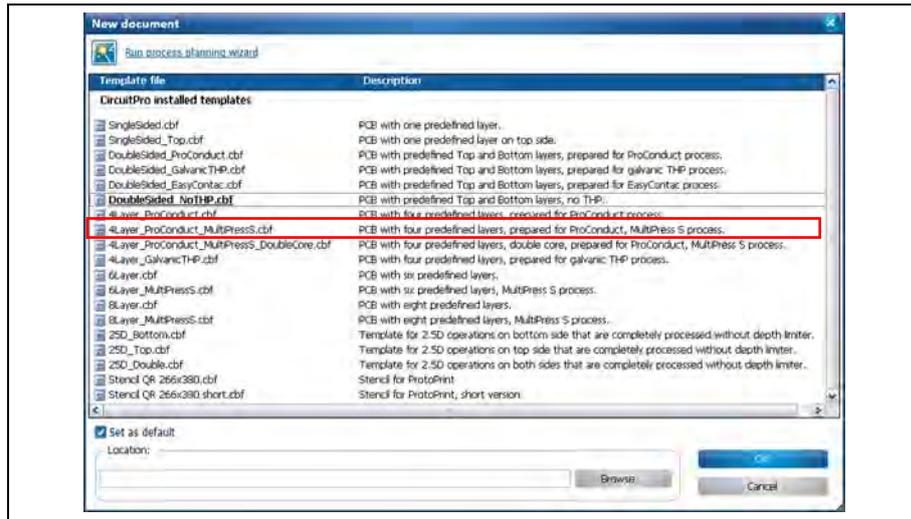
This section shows you how to create the new document move the fiducials into the prepreg area

■ Selecting a template and creating a new document

1. Click on File>New...

➔ The following dialog is displayed:

Fig. 188: New document



2. Select the template "4Layer_ProConduct_MultiPressS.cbf".



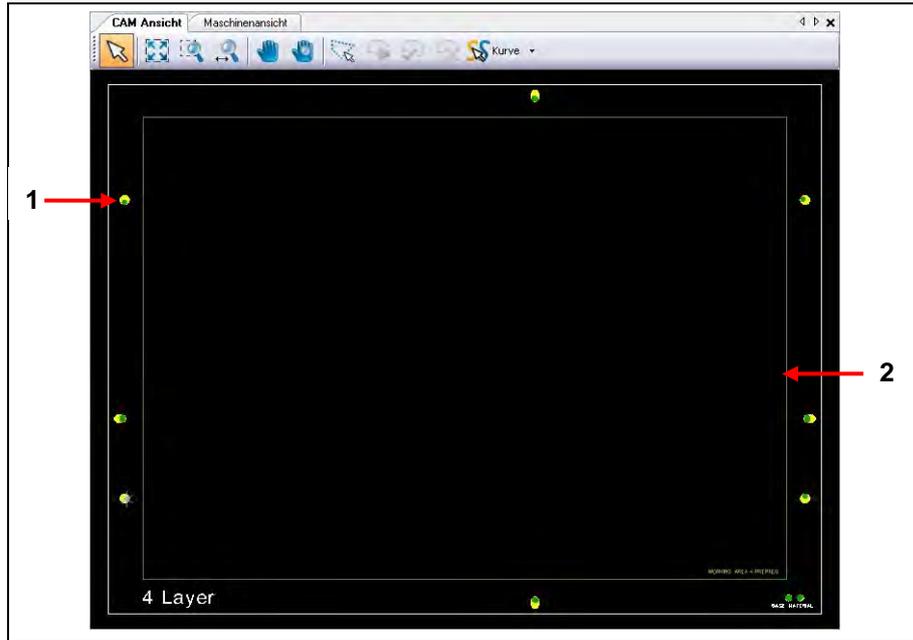
Note

You need to select a template that contains registration holes for the LPKF MultiPress S system because creating a flexi-rigid PCB requires bonding of rigid and flexible materials. Thus, the different physical layers can be aligned for bonding using the registration system of the MultiPress S system.

3. Click on [OK].

➔ The template is displayed in the CAM view:

Fig. 189: CAM view



/1/ Fiducial

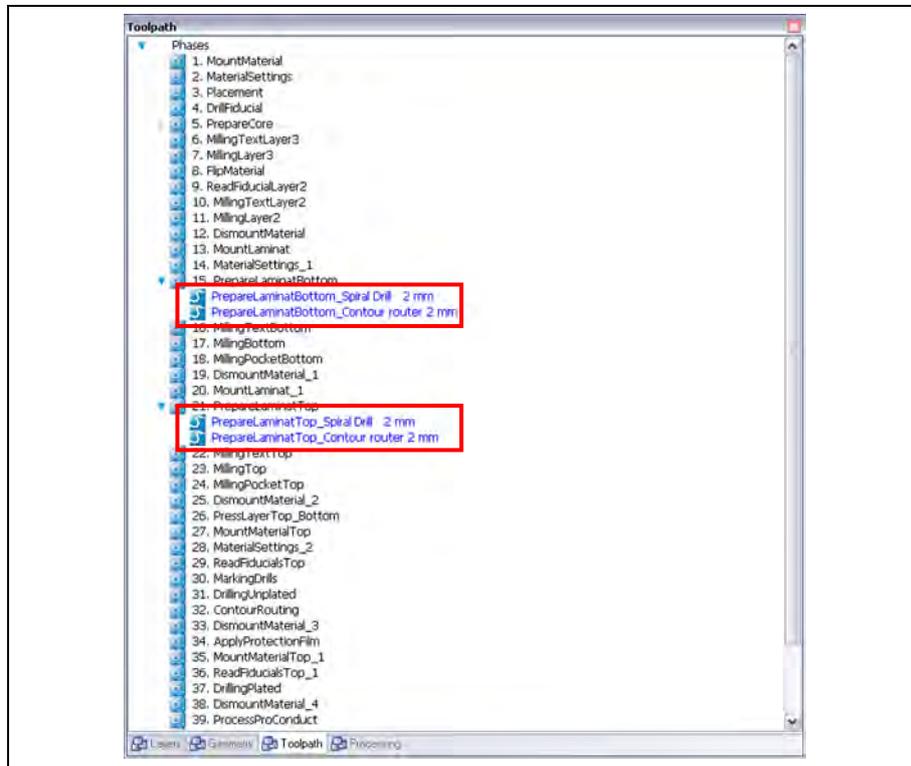
/2/ Prepreg

The fiducials in the selected template are outside the prepreg's area (see figure above) as the prepreg usually is not machined in a multi-layer application and thus does not need to be aligned to the other layers.

This tutorial's flexi-rigid PCB, however, requires a cutout in the prepreg. Thus, the prepreg has to be aligned to the other physical layers for bonding. Follow these steps:

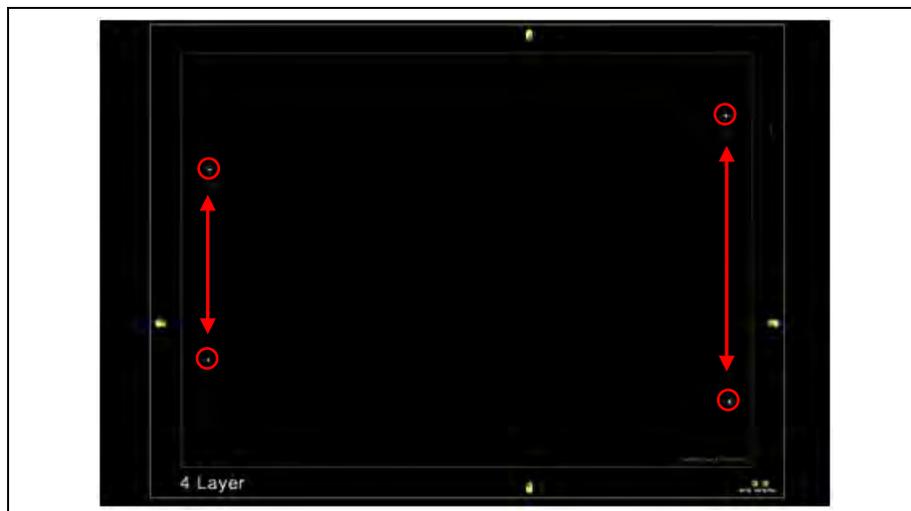
- Moving the fiducials into the prepreg area:
 1. Click on the “Toolpath” pane.
 2. Click on the plus symbol next to “Phases”.
- ➔ The phases are displayed:

Fig. 190: Phases



3. Delete the toolpaths of the phases “PrepareLaminatBottom” and “PrepareLaminatTop” marked in the figure above.
4. Move the fiducial into the prepreg area:
 - Activate the following “Move selected objects” icon: 
 - Click on the fiducial to be moved.
 - Drag the fiducial while pressing the left mouse button to the desired position in the CAM view.

Fig. 191: Phases





Note

Modify the distance between the fiducials on one side (see figure above) in order to orient and align the prepreg correctly to the other materials for bonding.

- ➔ The fiducials have been moved into the prepreg area.
 - 5. Click on File > Save As...
 - 6. Enter a file name for the new file.
 - 7. Select the memory location.
 - 8. Click on [Save].
-
- ◆ The new document is created.

6.3 Importing data

In this section you will pass through following steps:

- i. Importing data
- ii. Deleting unnecessary production phases
- iii. Assigning imported data to production phases

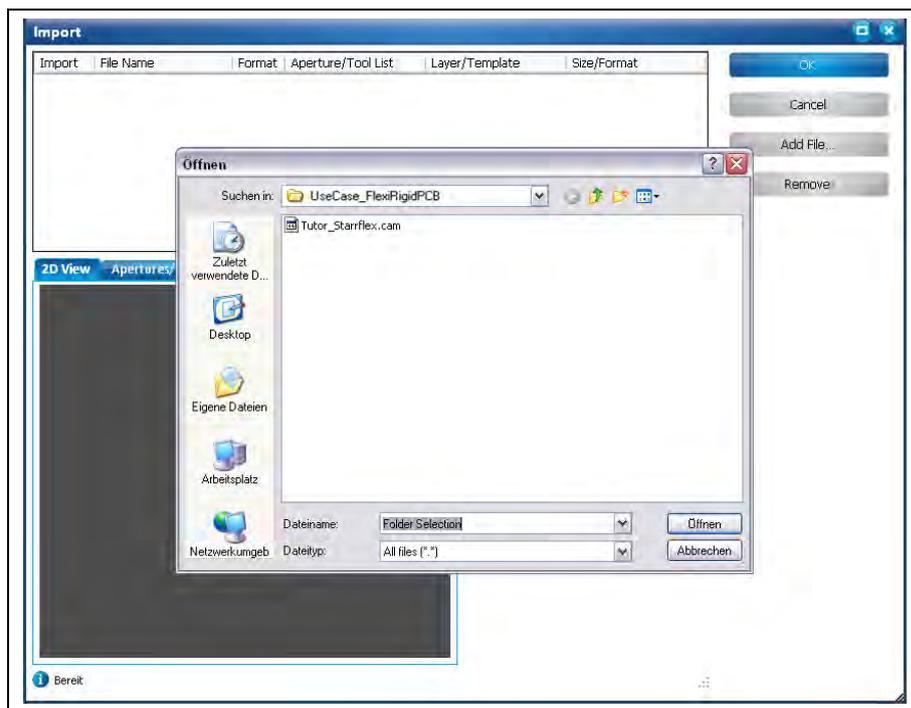


Tip

The LPKF tutor data are stored in folder “My documents\LPKF Laser & Electronics\LPKF CircuitPro 1.5\Example Data\UseCase_FlexiRigidPCB”.

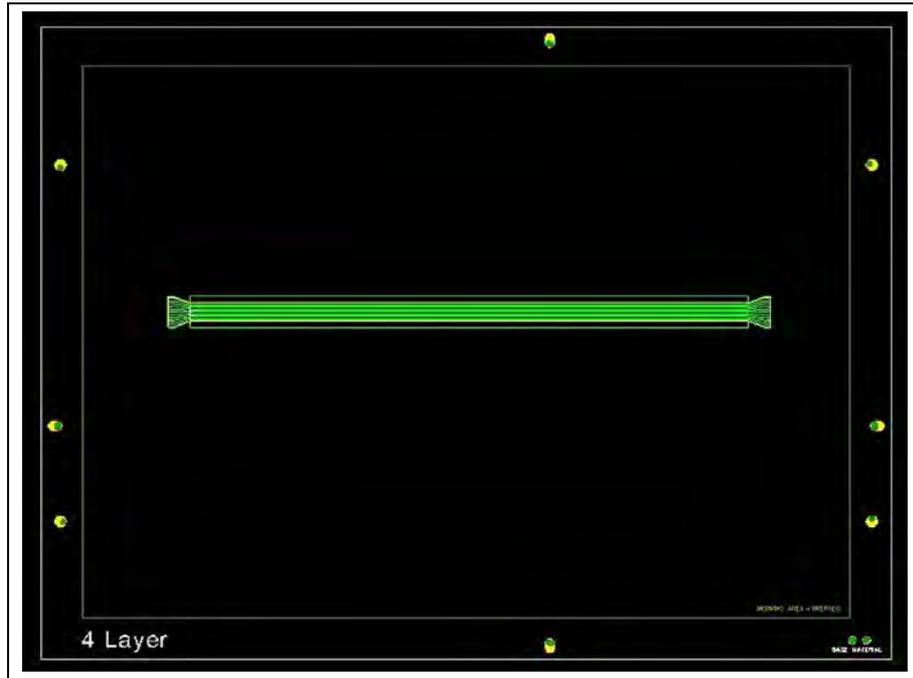
- Importing data
1. Click on File > Import...
- ➔ The following dialog is displayed:

Fig. 192: Import



2. Select the file “Tutor_Starrflex.cam”.
 3. Click on [OK].
- ➔ The data are displayed in the CAM view:

Fig. 193: CAM view



The modified template with imported .cam file now contains the following data:

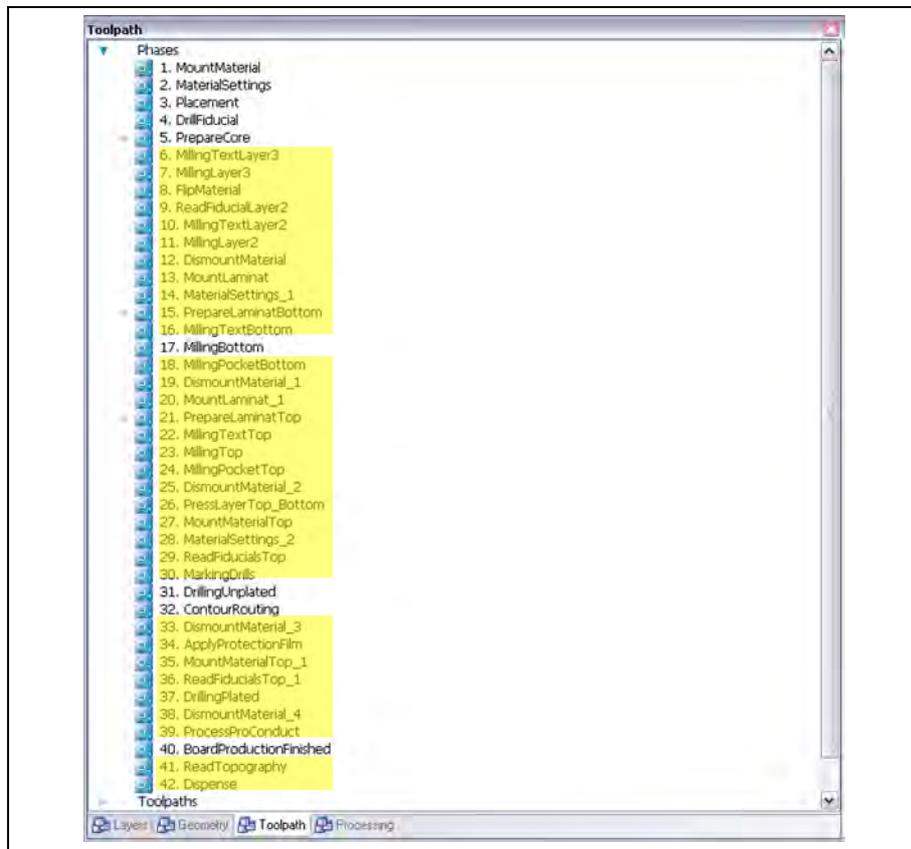
Material used	Layer	Function
FR4	CuttingInside	Cutout for flexible section
FR4	CuttingLaminat+Core	Slots for registration system of the LPKF MultiPress S
FR4	CuttingLaminat	Cutout for fiducials
Prepreg	CuttingInside	Cutout for flexible section
Prepreg	CuttingLaminat	Cutout for fiducials
Flexible material	BottomLayer	Insulation
Flexible material	Fiducial	Fiducials for aligning the individual layers for bonding
Flexible material	CuttingLaminat+Core	Slots for registration system of the LPKF MultiPress S
Bonded FR4 and flexible material	BoardOutline	Contour routing of the PCB

Due to the selected template "4Layer_ProConduct_MultiPressS.cbf", the job contains production phases that are not needed for processing this tutorial's PCB. These can be deleted in the "toolpath" pane before processing.

■ Deleting unnecessary production phases

1. Click on the “Toolpath” pane.

Fig. 194:
“Toolpath” pane



2. Delete the following phases in the “toolpath” pane:
 - MillingTextLayer3 through to MillingTextBottom
 - MillingPocketBottom through to MarkingDrills
 - DismountMaterial3 through to ProcessProConduct
 - ReadTopography through to Dispense
3. Select the desired phase.
4. Press the right mouse button in order to open the context menu.
5. Select “Delete” in the context menu.
 - ➔ The selected phase is deleted.
6. Repeat steps 3 to 5 for all other production phases to be deleted.
 - ◆ The unnecessary production phases are deleted.

■ Assigning imported data to production phases

Some layers of the imported data need to be assigned to a production phase. The generated toolpaths are thus assigned to a specific process step.

1. Click on the “Layer” pane.
2. Select the row of the desired layer:

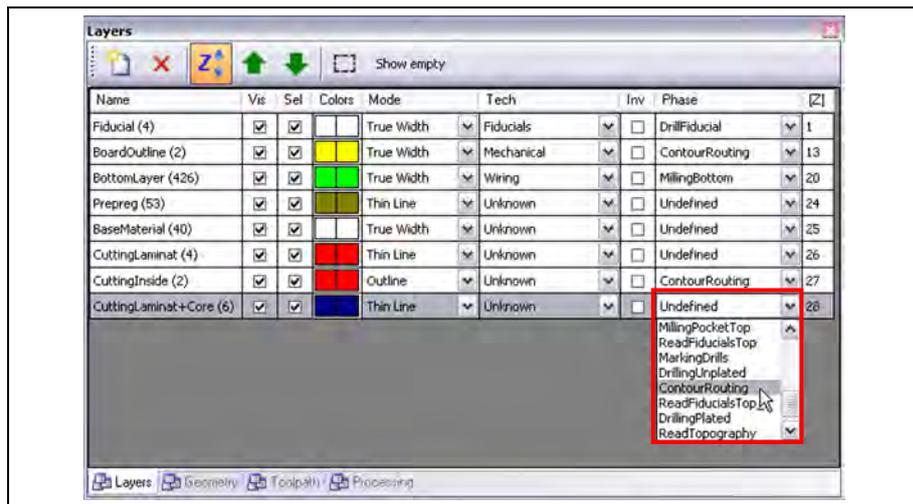
Fig. 195: “Layers” pane



3. In the layer row, click on the arrow in the “Phase” column.

➔ The phase selection list is displayed:

Fig. 196: Phase selection list



4. Assign the phase according to the following table:

Layer	Phase
CuttingLaminat	DrillingUnplated
CuttingInside	ContourRouting

◆ The data are assigned to production phases.

In this tutorial, the various physical layers are processed only on one side and are not turned over during processing (no “FlipMaterial” phases).



Note

As this tutorial uses a multi-layer template, some phases can contain Top-side data per default, other phases can contain Bottom-side data.

Ensure that the toolpaths of all production phases of this tutorial are on the same side, e.g. the Bottom side, so that the physical layers can be aligned correctly in the later process stages. You can check this as follows:

1. Click on the “Toolpath” pane.
 2. Click on the desired phase using the right mouse button.
 - → The context menu is displayed.
 3. Click on “Edit” in the context menu.
 - The “Edit” dialog is displayed. If the phase contains toolpaths, the check box “The data are mirrored” must be checked.
 4. Check the check box “The data are mirrored” if the phase contains toolpaths.
 5. Click on [Apply].
 6. Click on [Close].
-

6.4 Processing the individual physical layers

The imported .cam file contains the data for producing the whole flexi-rigid PCB but these are needed at different stages of the production process.

Creating the toolpaths should be done separately for each production stage. Thus, the data needed for each production stage can be selected correspondingly for generating the toolpaths.

Processing the individual physical layers is divided into three sections:

- i. Processing the flexible material
- ii. Processing the FR4 material
- iii. Processing the prepreg material

6.4.1 Drilling and milling the flexible material

In this section, the flexible material of the flexi-rigid PCB is drilled and milled. The following steps are necessary for drilling and milling the flexible material:

- i. Creating toolpaths
- ii. Loading the tool magazine and assigning tools to holder positions
- iii. Starting the processing



This tutorial uses a 0.15-mm-thick flexible polyimide material with an 18- μ m-thick copper layer.

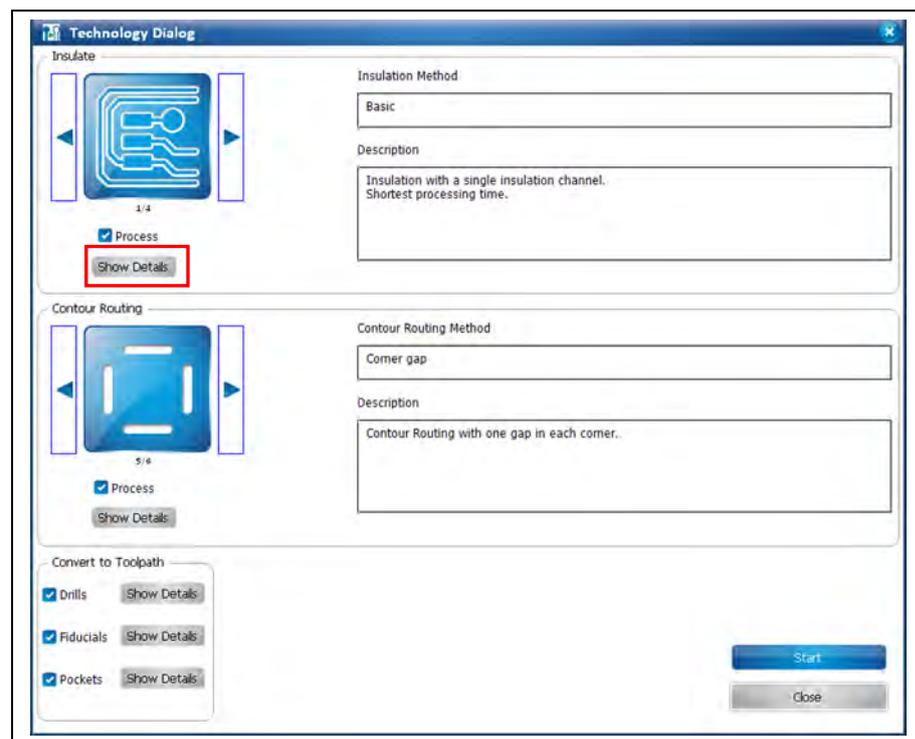
Note

Creating toolpaths

1. Click on Toolpath > Technology dialog...

➔ The following dialog is displayed:

Fig. 197:
Technology
Dialog



In the technology dialog, the settings for generating toolpaths can be modified by clicking on the [Show details] buttons.

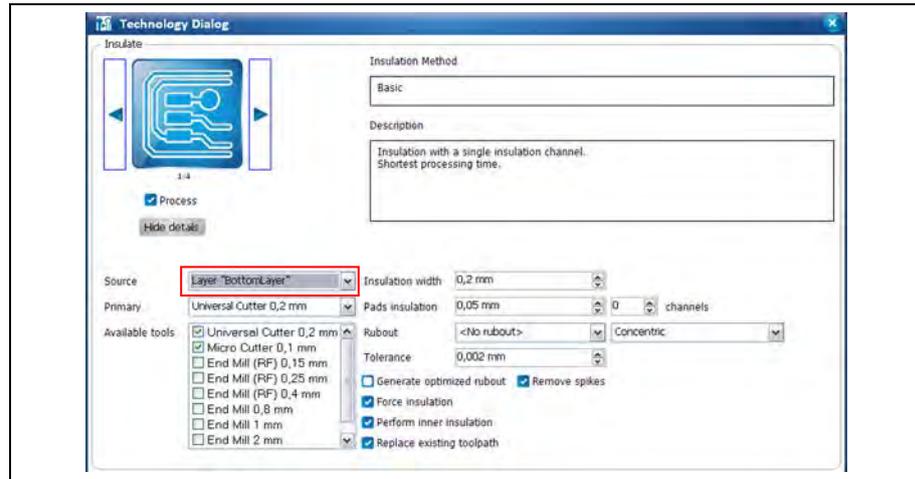
Note

For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

2. Click on [Show Details] in the “Insulate” section.

➔ The details for insulation are displayed.

Fig. 198:
Insulation details



3. Select the layer "BottomLayer" in the "Source" selection list.

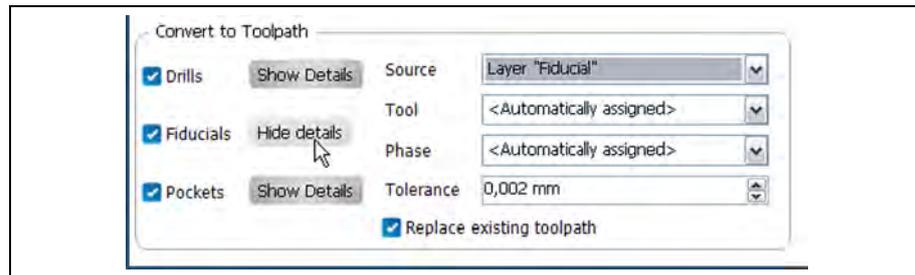
4. Click on [Hide details].

➔ The details are hidden.

5. Click on [Show Details] next to "Fiducials".

➔ The details for fiducials are displayed.

Fig. 199:
Fiducials - details



6. Select the "Fiducial" layer in the "Source" selection list.

7. Click on [Hide details].

➔ The details are hidden.

8. Deactivate the following functions by clicking on the corresponding check marks:

- Contour Routing
- Drilling
- Milling pockets

9. Click on [Start].

➔ The toolpaths are created and the computation results are displayed..

◆ The toolpaths are created.

Loading the tool magazine and assigning tools to holder positions



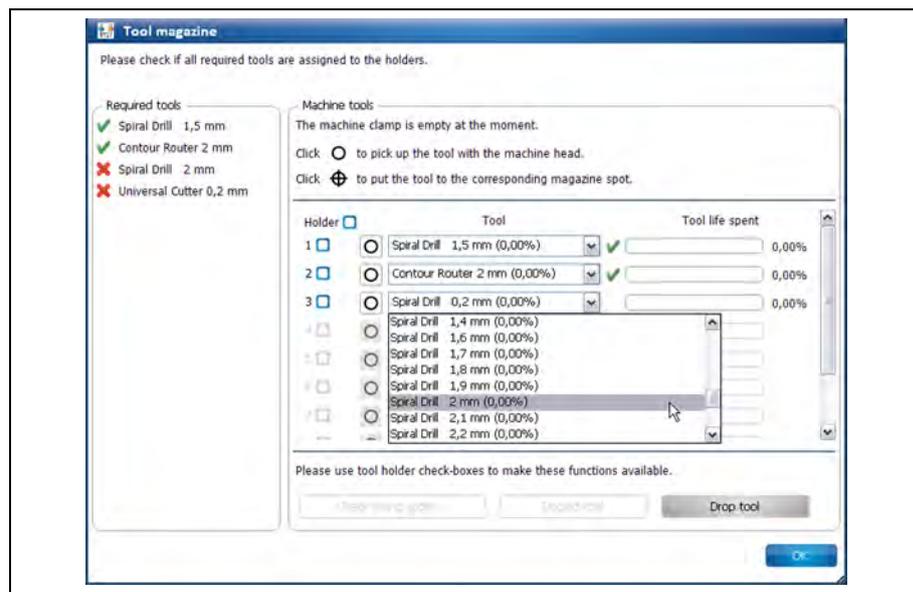
This section is only relevant, if you use a ProtoMat with automatic tool change (S63 or S103).

Note

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

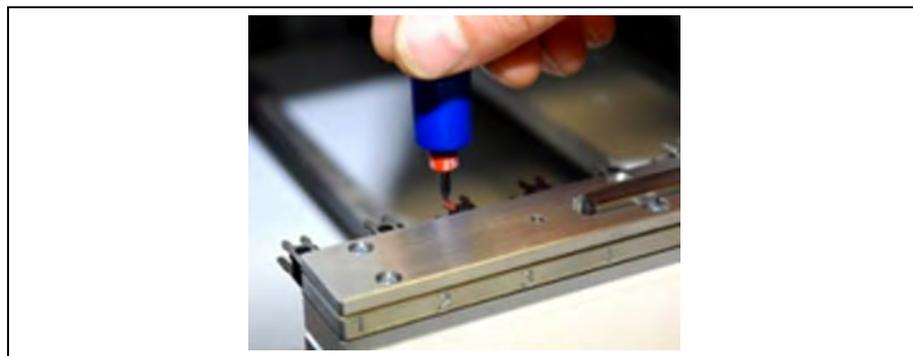
Fig. 200: Tool magazine



➔ The tools required for the job are displayed. Tools required for the job that are missing are marked by a red "X".

2. Insert the required tools into the tool holders of the machine:

Fig. 201: Inserting a tool



3. Assign the tools to the corresponding positions in the dialog.

➔ The tool holders of the machine are loaded:

Fig. 202: Loaded tool holder

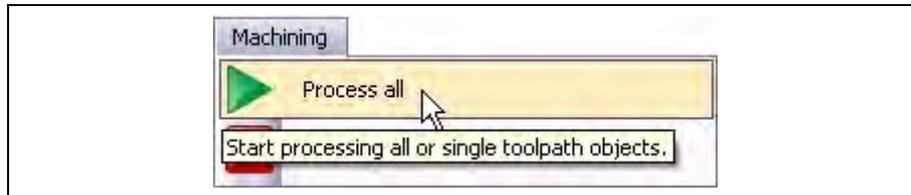


◆ The tools are inserted into the tool holders and assigned accordingly.

Starting the processing

1. Click on Machining > Process all.

Fig. 203:
Machining > Process all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

Once you have started the processing, the ProtoMat machine executes the job in individual phases. The phases are displayed in messages:

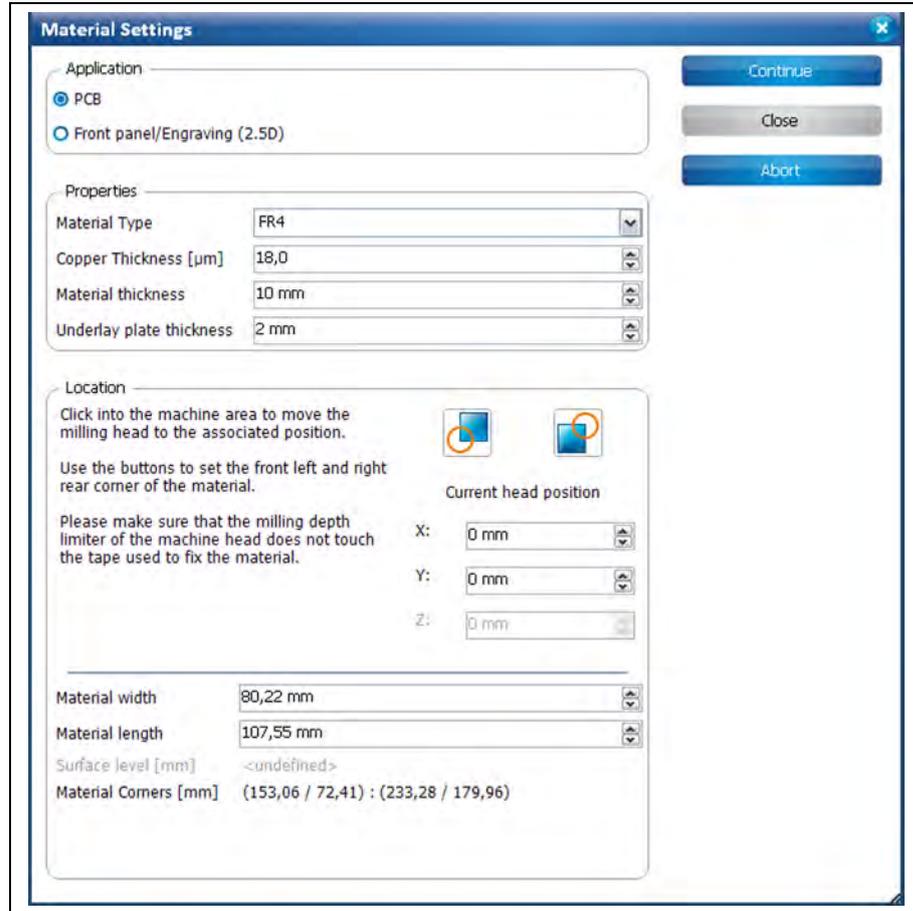
Phase "MountMaterial"

1. Mount the flexible material onto the processing area of the machine.
2. Fasten the base material onto the table top using the adhesive tape.
3. Click on [OK].

Phase “Material Settings”

➔ The following dialog is displayed:

Fig. 204: Material Settings



■ Entering the material settings

1. Enter the correct values for the material used.

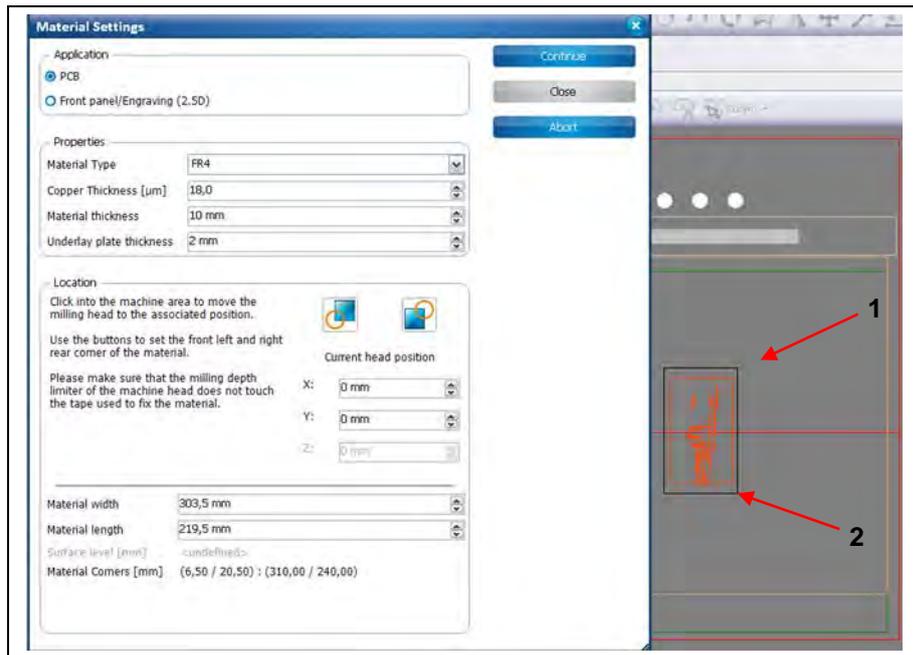


PCB is selected as default.

Note

2. Enter “18” µm into the field \Copper Thickness\.
3. Enter “0.15” mm into the field \Material Thickness\.
4. Define the processing area:
 - a) Move the “Material Settings” dialog off to the side.
 - b) Click on the position in the machining view that represents the right corner of your material:

Fig. 205: Defining the right corner



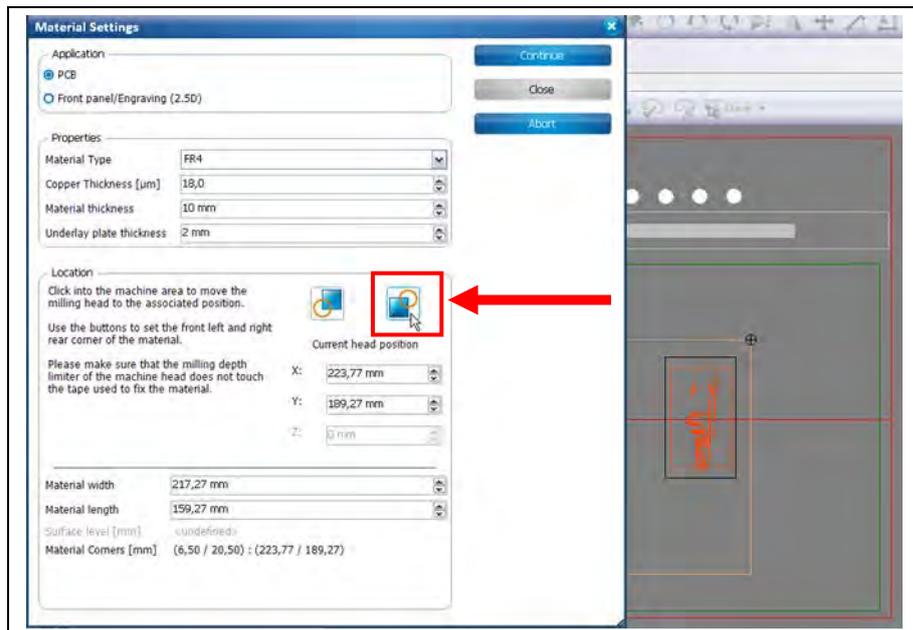
/1/ Click on machining area

/2/ Job

➔ The machine head moves to this position.

c) Click on the corresponding icon in the "Material Settings" dialog:

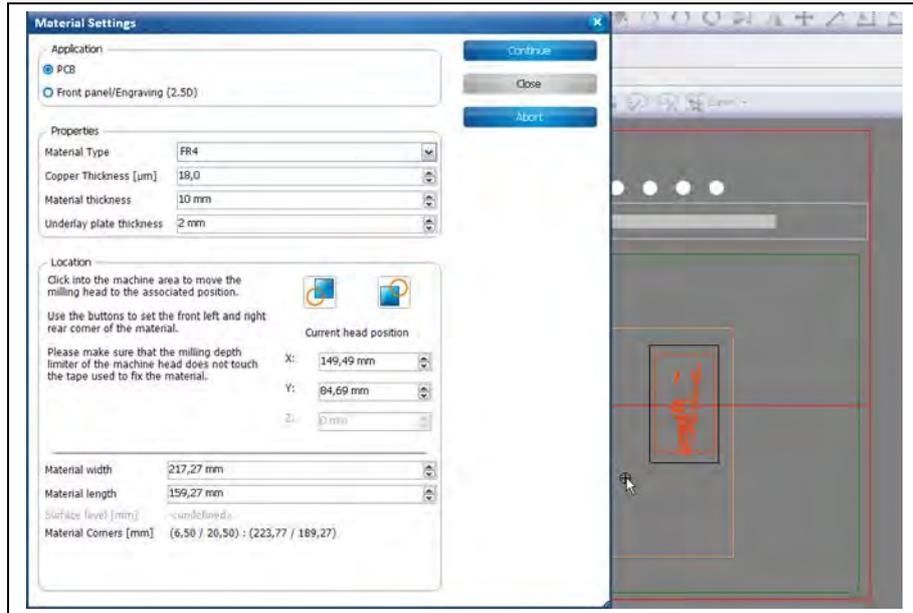
Fig. 206: Click on the icon



➔ The machining area is adjusted.

d) Click on the position in the machining view that represents the left corner of your material:

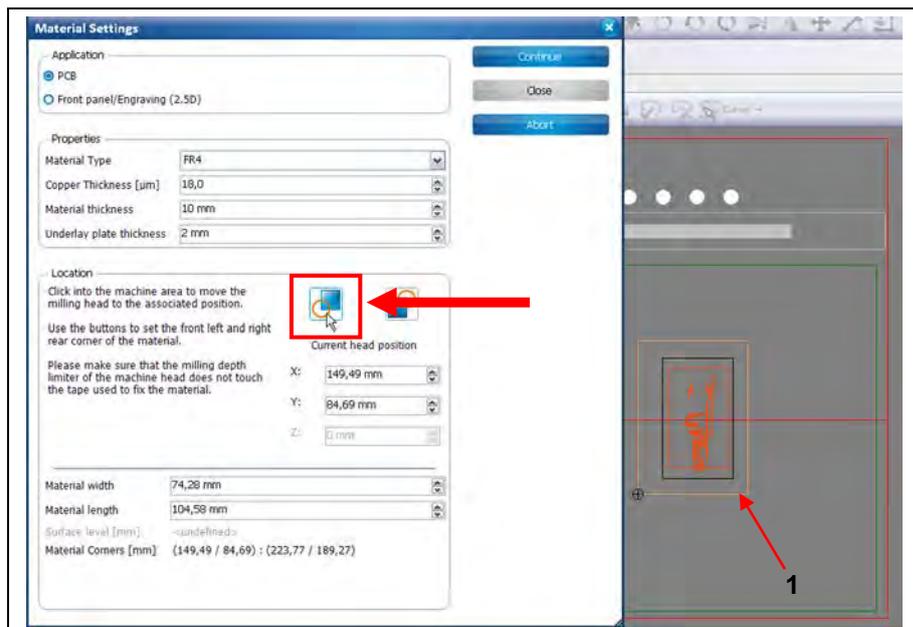
Fig. 207: Defining the left corner



➔ The machine head moves to this position.

e) Click on the corresponding icon in the “Material Settings” dialog:

Fig. 208: Defined processing area



/1/ Defined processing area

➔ The processing area has been fit to the material.

5. Click on [Continue].

◆ The material settings were entered.

Phase "Placement"

In this phase, the job can be placed arbitrarily on the base material, be rotated and/or be multiplied if necessary.



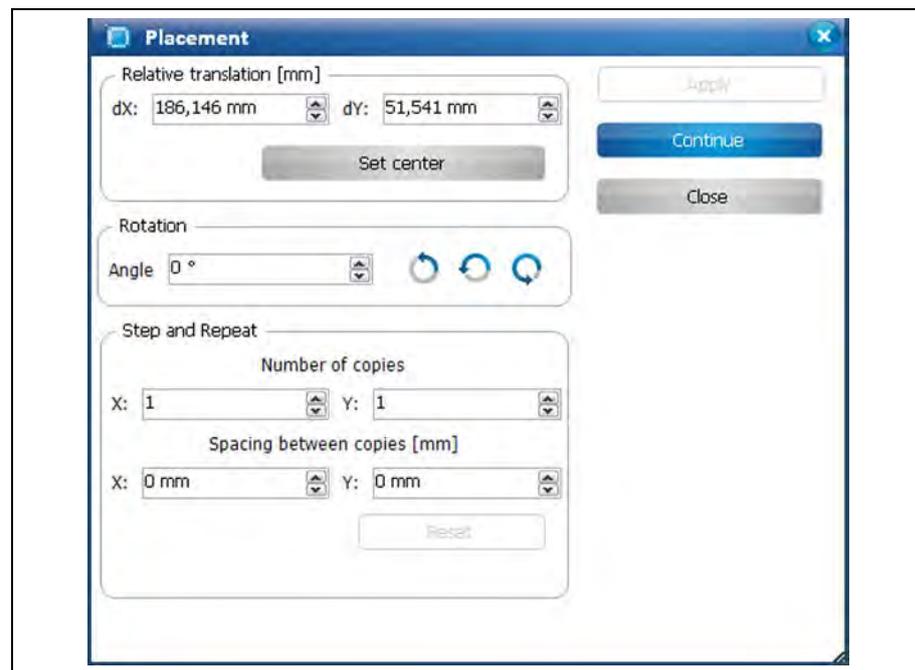
Note

At this point, the whole circuit board design including the template and the toolpaths possibly contained in the template are moved.

If only the design is to be moved within the template, you have to use the menu item Modify > Transform....

➔ Following dialog is displayed:

Fig. 209:
Placement



1. Click on the job and drag it to the desired position using the mouse.
- Or
1. Enter the new position in the dialog.
 2. If desired, rotate the job data by entering a rotation angle.
 3. If desired, multiply the job data by entering the number of copies and spacing values in X and Y direction in the corresponding fields (Step and Repeat section).

Phase "DrillFiducial"

➔ The Spiral Drill 1.5 mm is picked up to drill the fiducials.



Note

If the spindle motor has not run before, a 2-minute warm-up phase is started.

Phase “PrepareCore”

- ➔ The slots for registration system of the LPKF MultiPress S are drilled.

Phase “MillingBottom”

- ➔ The flexible material is milled.

Phase “Board Production Finished”

- ➔ A message informs you that the processing is finished.
- ◆ The milling and drilling of the flexible material is finished.

6.4.2 Processing the FR4 material

In this section, the FR4 material of the flexi-rigid PCB is processed.

The following steps are necessary in this section:

- i. Deleting toolpaths
- ii. Creating toolpaths
- iii. Loading the tool magazine and assigning tools to holder positions
- iv. Starting the processing



Note

This tutorial uses FR4 material with an 1 mm-thick copper layer.

Deleting toolpaths

The toolpaths that were generated in the previous section of this tutorial for insulation and drilling the fiducials have to be deleted. They are not needed for processing the FR4 material.

1. Click on the "Toolpath" pane.
 2. Click on the plus symbol next to phases "DrillFiducial" and "MillingBottom".
- ➔ The toolpaths of the phases are displayed.

The following toolpaths have to be deleted:

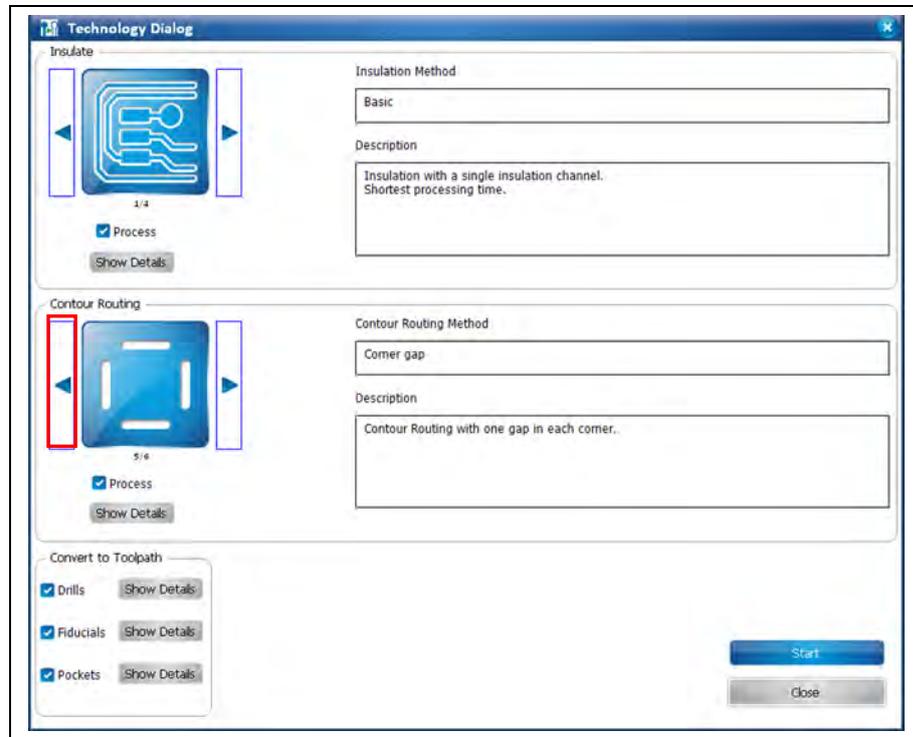
Phase	Toolpath
DrillFiducial	Fiducials_Fiducial_Spiral Drill 1.5 mm
MillingBottom	Insulate_BottomLayer_Universal Cutter 0.2 mm
	Insulate_BottomLayer_Universal Cutter 0.2 mm_1

3. Select the desired toolpath.
 4. Press the right mouse button in order to open the context menu.
 5. Select "Delete" in the context menu.
- ➔ The selected toolpaths are deleted.
6. Repeat steps 3 to 5 for all other toolpaths to be deleted.

Creating toolpaths

1. Click on Toolpath > Technology dialog...
- ➔ The following dialog is displayed:

Fig. 210:
Technology
Dialog



Note

In the technology dialog, the settings for generating toolpaths can be modified by clicking on the [Show details] buttons.

For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

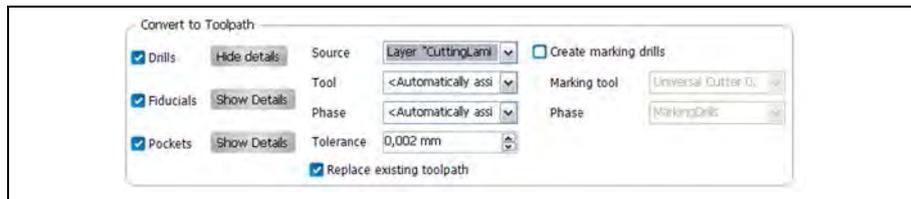
2. Click on the left arrow icon in the “Contour Routing” section until the “Basic” contour routing method is displayed.
 3. Click on [Show details].
- ➔ The details for contour routing are displayed:

Fig. 211: Contour
routing details



4. Select the layer "CuttingInside" in the "Source" selection list.
5. Activate the {Inside} radio button.
6. Click on [Hide details].
- ➔ The details are hidden.
7. Click on [Show Details] next to "Drills".
- ➔ The details are displayed:

Fig. 212: Drills details



8. Select the layer "CuttingLaminat" in the "Source" selection list.
9. Deactivate the option "Create marking drills".
10. Click on [Hide details].
- ➔ The details are hidden.
11. Deactivate the following functions by clicking on the corresponding check marks:
 - Insulate
 - Drilling fiducials
 - Pockets
12. Click on [Start].
- ➔ The toolpaths are created and the computation results are displayed..
- ◆ The toolpaths are created.

Loading the tool magazine and assigning tools to holder positions



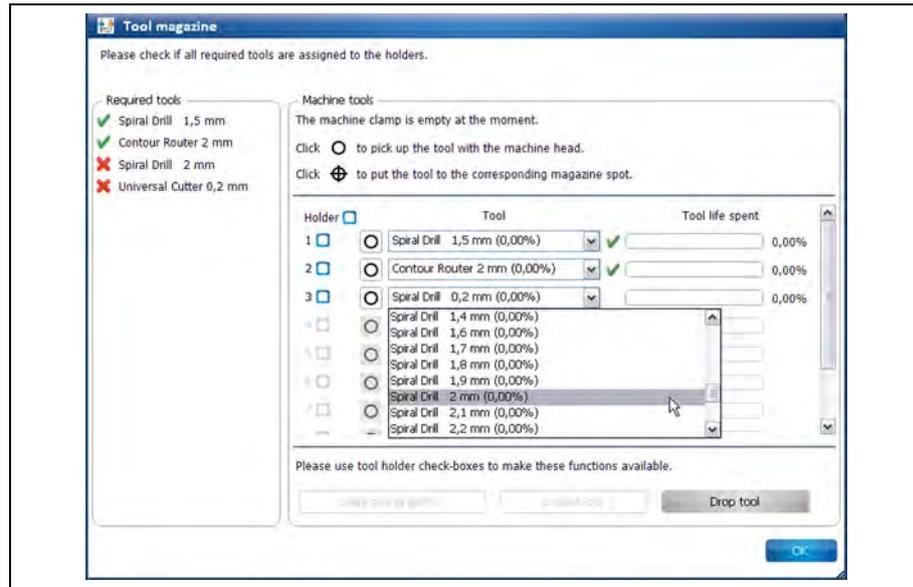
This section is only relevant, if you use a ProtoMat with automatic tool change (S63 or S103).

Note

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

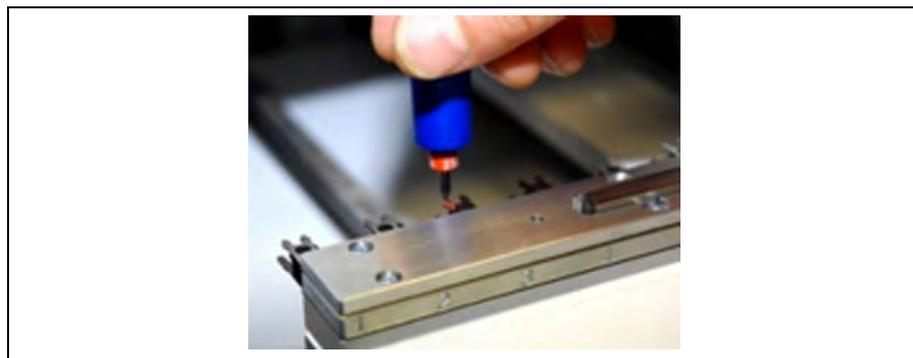
Fig. 213: Tool magazine



➔ The tools required for the job are displayed. Tools required for the job that are missing are marked by a red "X".

2. Insert the required tools into the tool holders of the machine:

Fig. 214: Inserting a tool



3. Assign the tools to the corresponding positions in the dialog.

➔ The tool holders of the machine are loaded:

Fig. 215: Loaded tool holder

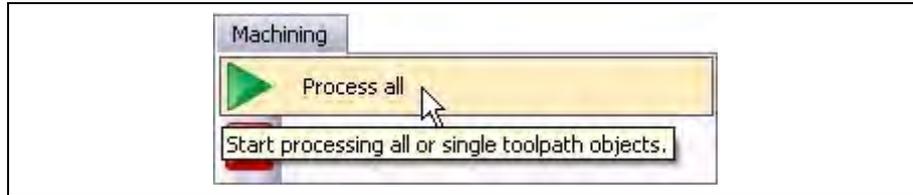


◆ The tools are inserted into the tool holders and assigned accordingly.

Starting the processing

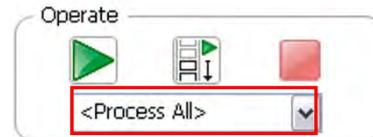
1. Click on Machining > Process all.

Fig. 216:
Machining > Pro-
cess all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

Once you have started the processing, the ProtoMat machine executes the job in individual phases. The phases are displayed in messages:

Phase "MountMaterial"

1. Mount the base material onto the processing area of the machine.
2. Fasten the material to the processing area using adhesive tape.
3. Click on [OK].

Phase “Material Settings”

➔ The following dialog is displayed:

Fig. 217: Material Settings

■ Entering the material settings

1. Enter the correct values for the material used.

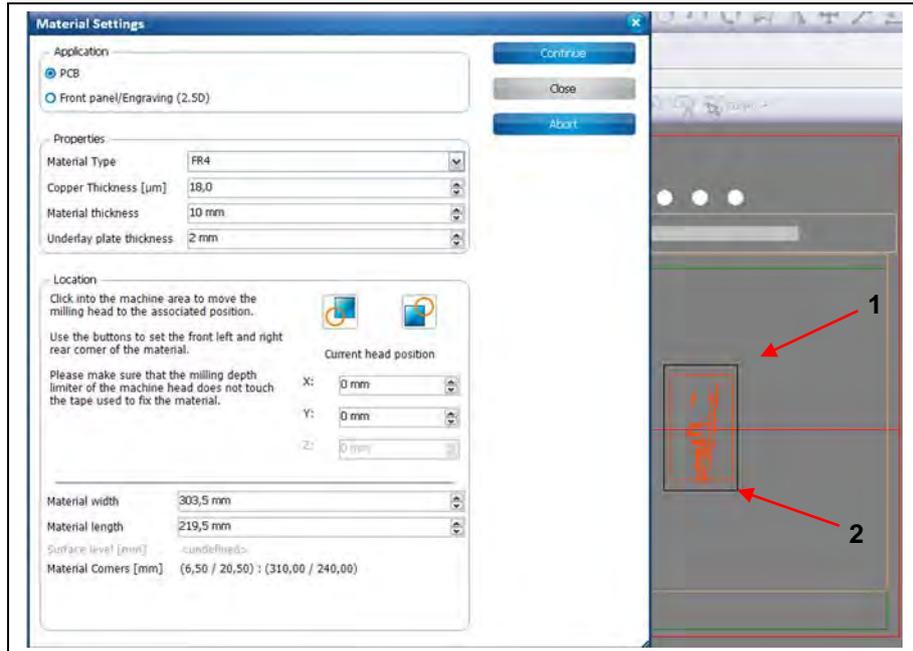


PCB is selected as default.

Note

2. Enter “18” μm into the field \Copper Thickness\.
3. Enter “0.15” mm into the field \Material Thickness\.
4. Define the processing area:
 - a) Move the “Material Settings” dialog off to the side.
 - b) Click on the position in the machining view that represents the right corner of your material:

Fig. 218: Defining the right corner



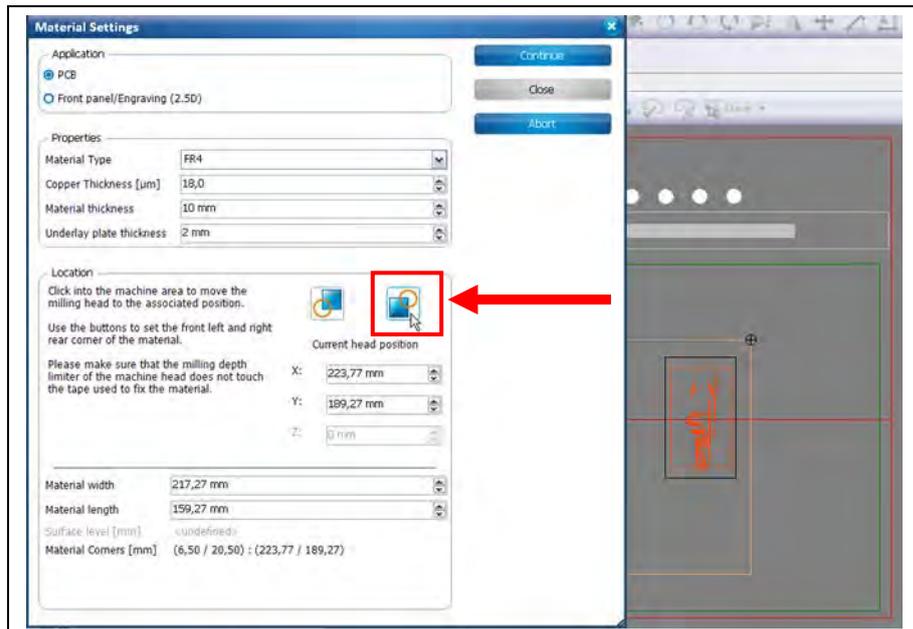
/1/ Click on machining area

/2/ Job

➔ The machine head moves to this position.

c) Click on the corresponding icon in the "Material Settings" dialog:

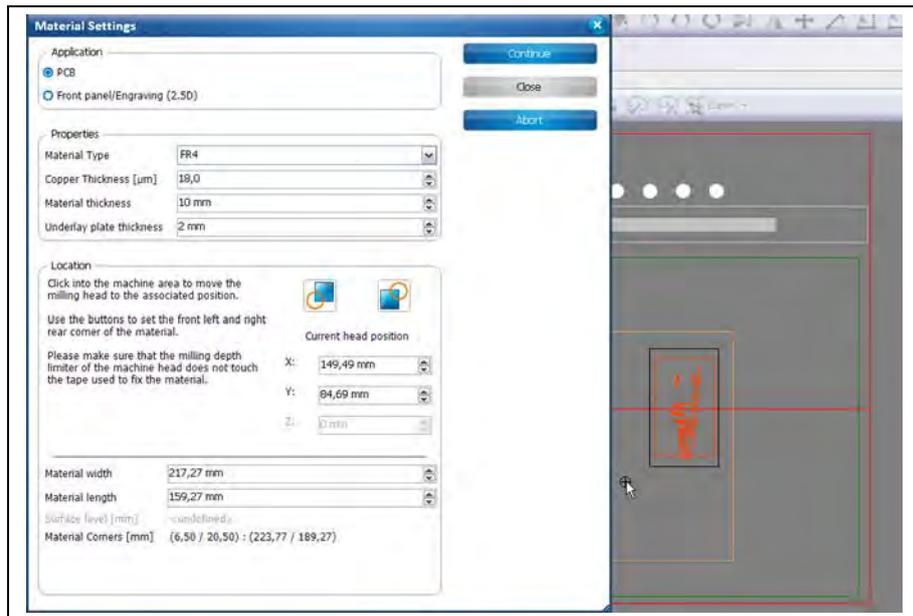
Fig. 219: Click on the icon



➔ The machining area is adjusted.

d) Click on the position in the machining view that represents the left corner of your material:

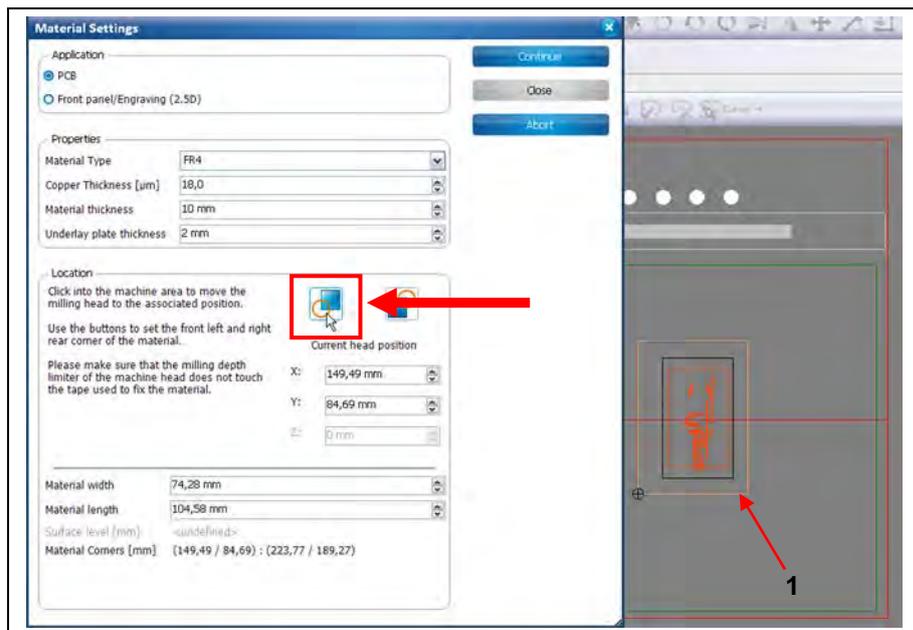
Fig. 220: Defining the left corner



➔ The machine head moves to this position.

e) Click on the corresponding icon in the "Material Settings" dialog:

Fig. 221: Defined processing area



/1/ Defined processing area

➔ The processing area has been fit to the material.

5. Click on [Continue].

◆ The material settings were entered.

Phase “Placement”

The job cannot be placed again at this stage as the position has already been defined by the flexible material in the previous steps.

1. Confirm the current placement by clicking on [OK].

Phase “PrepareCore”

- ➔ The slots for the registration system of the LPKF MultiPress S are drilled.

Phase “DrillingUnplated”

- ➔ The cutouts for aligning the prepreg for bonding are drilled.

Phase “ContourRouting”

- ➔ The cutout for the flexible section is cut out.

Phase “Board Production finished”

- ➔ A message informs you that the processing is finished.
- ◆ Processing of the FR4 material is finished.

6.4.3 Processing the prepreg material

This section of the tutorial is on processing the prepreg material of the flexi-rigid PCB.

The following steps are necessary for processing the prepreg material:

- i. Deleting toolpaths
- ii. Enlarging the cutouts around the fiducials
- iii. Creating toolpaths
- iv. Loading the tool magazine and assigning tools to holder positions
- v. Starting the processing



This tutorial uses a 100 mm thick prepreg material

Note

Deleting toolpaths

The toolpaths for processing the FR4 material have to be deleted as they are not necessary for processing the prepreg material or have to be re-computed.

■ Deleting toolpaths

1. Click on the "Toolpath" pane.
2. Click on the plus symbol next to phases "PrepareCore", "DrillingUnplated", and "ContourRouting".

➔ The toolpaths of the phases are displayed.

The following toolpaths have to be deleted:

Phase	Toolpath
PrepareCore	PrepareCore_Contour router 2 mm
	PrepareCore_Spiral Drill 2 mm
DrillingUnplated	Drills_CuttingLaminat_Contour router 2 mm
	Drills_CuttingLaminat_Spiral Drill 2 mm
ContourRouting	ContourRouting_CuttingInside_Contour router 2 mm
	ContourRouting_CuttingInside_Spiral Drill 2 mm

3. Select the desired toolpath.
4. Press the right mouse button in order to open the context menu.
5. Select "Delete" in the context menu.

➔ The selected toolpaths are deleted.

6. Repeat steps 3 to 5 for all other toolpaths to be deleted.

◆ The toolpaths are deleted.

Enlarging the cutouts around the fiducials

To enable recognising the holes in the flexible material for contour routing the PCB, the cutouts around the fiducials have to be enlarged in the prepreg material. This prevents the prepreg material from creeping into the fiducials during the bonding process.

■ Enlarging the cutouts around the fiducials

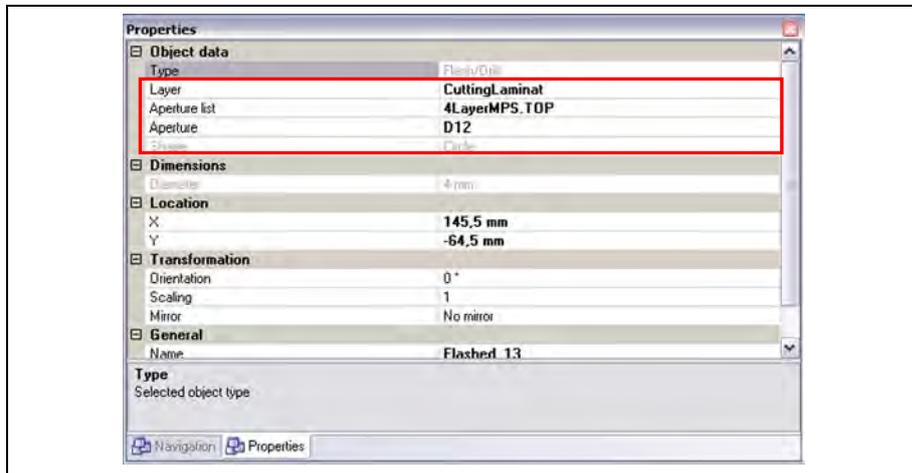
1. Select the cutouts around the fiducials:

Fig. 222:
Selecting a
cutout around a
fiducial



2. Click on the “Properties” pane.
3. Check the properties of the circular cutout around the fiducial:

Fig. 223:
Properties of the
cutout



The “Properties” pane displays the following information on the cutout:

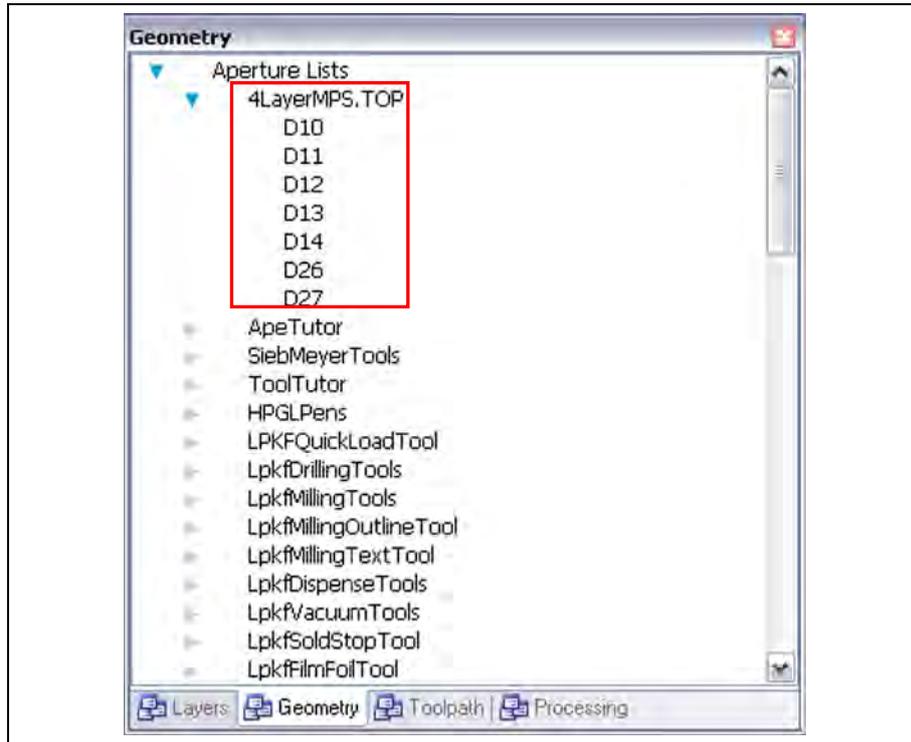
- Aperture D12 is used to create the cutout.
- Aperture D12 is in aperture list “4LayersMPS.TOP”.

As the cutout around the fiducials is to be enlarged, check the diameter of the aperture next.

4. Click on the “Geometry” pane.
5. Click on the plus symbol next to the aperture lists.

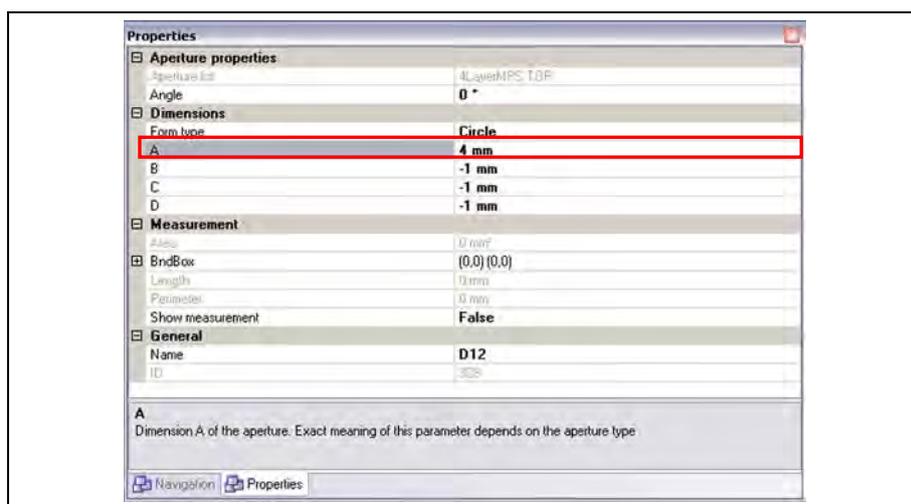
- ➔ The aperture lists are displayed:

Fig. 224: Aperture lists



6. Click on the plus symbol next to aperture list “4LayerMPS.TOP”.
- ➔ The apertures contained are listed. Among them, you find aperture D12 that is used for creating the cutout around the fiducial.
7. Select aperture D12.
8. Click on the “Properties” pane again.
- ➔ The properties of aperture D12 are displayed:

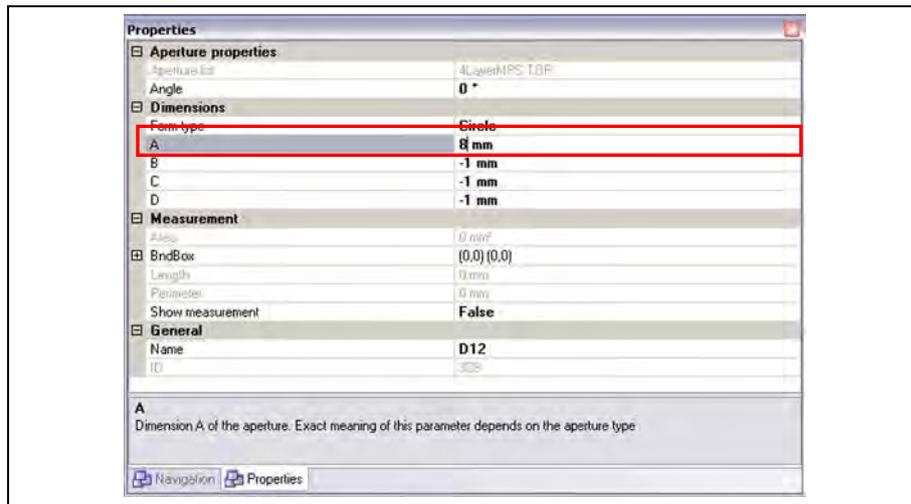
Fig. 225: Aperture size



- ➔ The name of the aperture is displayed in the “General” section.
- ➔ The diameter of the aperture is displayed in the “Dimensions” section. The diameter is 4 mm.

9. Click on the aperture size in order to modify it:

Fig. 226:
Modifying the
aperture size



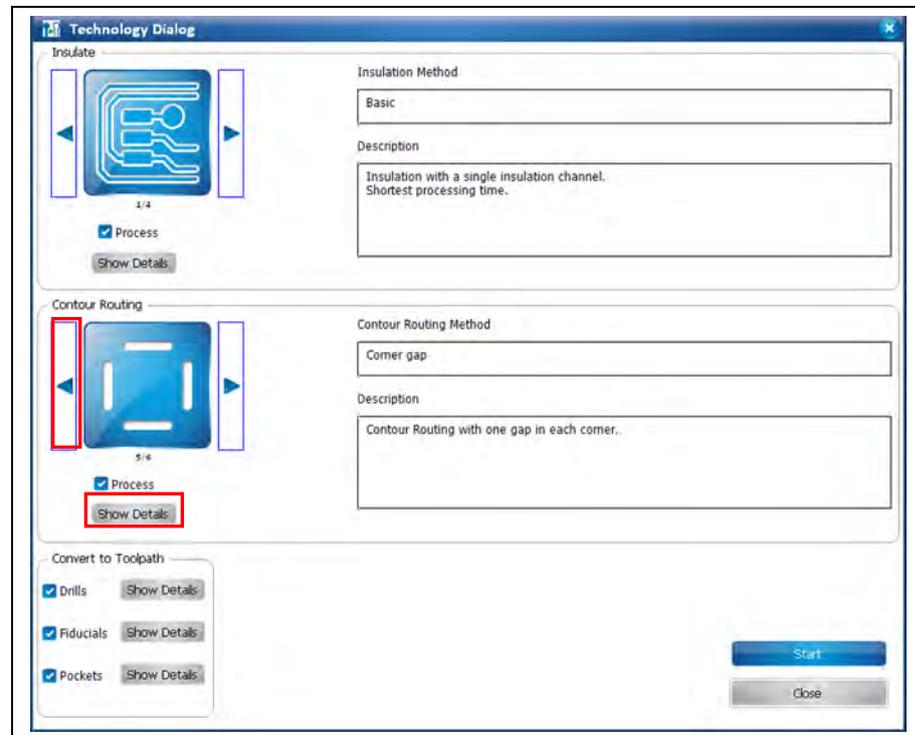
10. Overwrite the "4" with an "8".

- ➔ The aperture size is changed from 4 mm to 8 mm diameter.
- ◆ The cutouts around the fiducials are enlarged.

Creating toolpaths

- Creating toolpaths
- 1. Click on Toolpath > Technology dialog...
- ➔ The following dialog is displayed:

Fig. 227:
Technology
Dialog

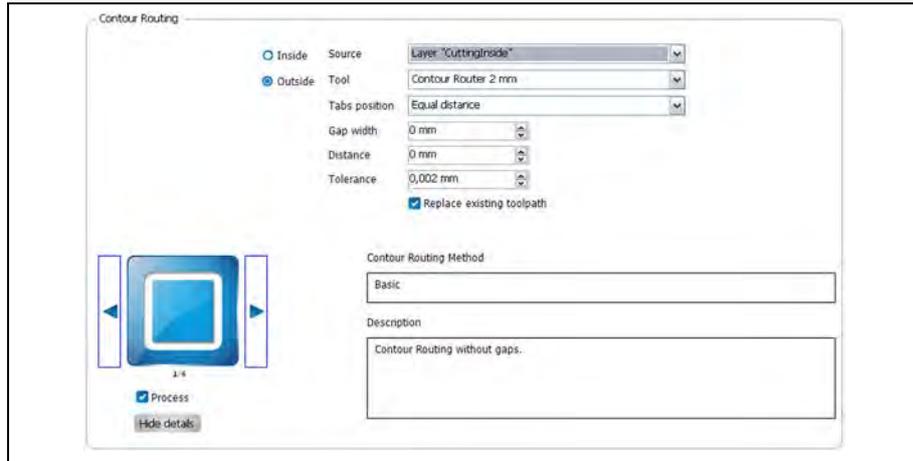


Note

In the technology dialog, the settings for generating toolpaths can be modified by clicking on the [Show details] buttons. For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

- 2. Click on the left arrow icon in the “Contour Routing” section until the “Basic” contour routing method is displayed.
- 3. Click on [Show details].
- ➔ The details for contour routing are displayed:

Fig. 228: Contour routing details



4. Select the layer "CuttingInside" in the "Source" selection list.
5. Activate the {Outside} radio button.
6. Select the "Contour router 2 mm" tool.

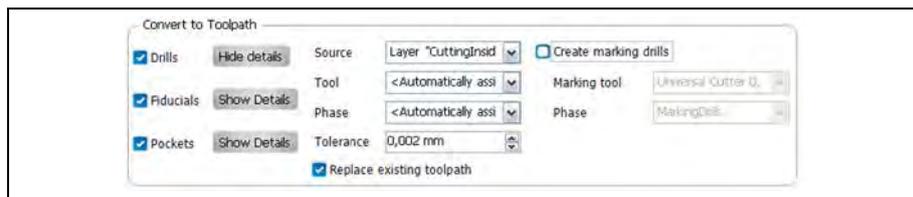


Note

Using the outside contour in combination with the 2-mm tool enlarges the cutout by 2 mm compared with the FR4 material. This is intentional as the prepreg material "creeps" during the bonding process.

7. Click on [Hide details].
- ➔ The details are hidden.
8. Click on [Show Details] next to "Drills".
- ➔ The details are displayed:

Fig. 229: Drills details



9. Select the layer "CuttingLaminat" in the "Source" selection list.
10. Deactivate the option "Create marking drills".
11. Click on [Hide details].
- ➔ The details are hidden.

12. Deactivate the following functions by clicking on the corresponding check marks:
 - Insulate
 - Fiducials
 - Pockets

13. Click on [Start].
 - ➔ The toolpaths are created and the computation results are displayed..
 - ◆ The toolpaths are created.

Loading the tool magazine and assigning tools to holder positions



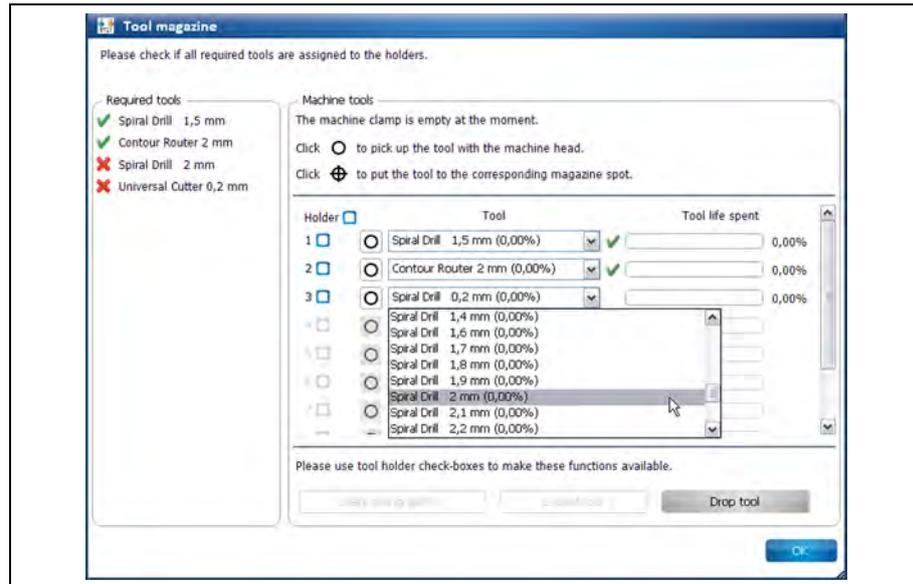
This section is only relevant, if you use a ProtoMat with automatic tool change (S63 or S103).

Note

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

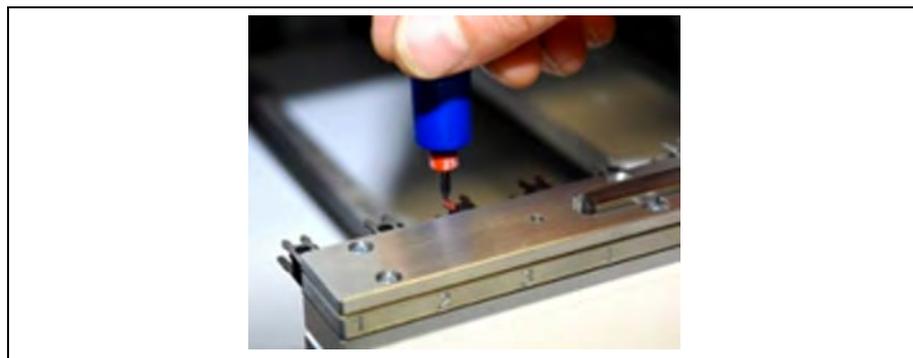
Fig. 230: Tool magazine



➔ The tools required for the job are displayed. Tools required for the job that are missing are marked by a red "X".

2. Insert the required tools into the tool holders of the machine:

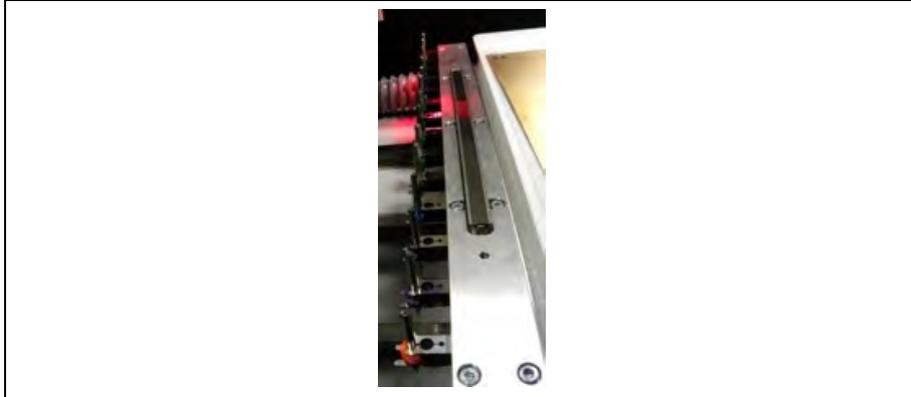
Fig. 231: Inserting a tool



3. Assign the tools to the corresponding positions in the dialog.

➔ The tool holders of the machine are loaded:

Fig. 232: Loaded tool holder

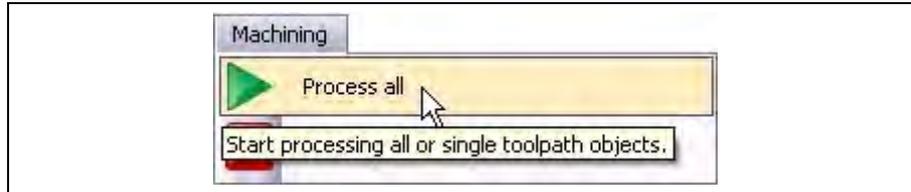


◆ The tools are inserted into the tool holders and assigned accordingly.

Starting the processing

1. Click on Machining > Process all.

Fig. 233:
Machining > Process all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

Once you have started the processing, the ProtoMat machine executes the job in individual phases. The phases are displayed in messages:

Phase "MountMaterial"

1. Mount the base material onto the processing area of the machine.
2. Fasten the material to the processing area using adhesive tape.
3. Click on [OK].

Phase “Material Settings”

➔ The following dialog is displayed:

Fig. 234: Material Settings

■ Entering the material settings

1. Enter the correct values for the material used.

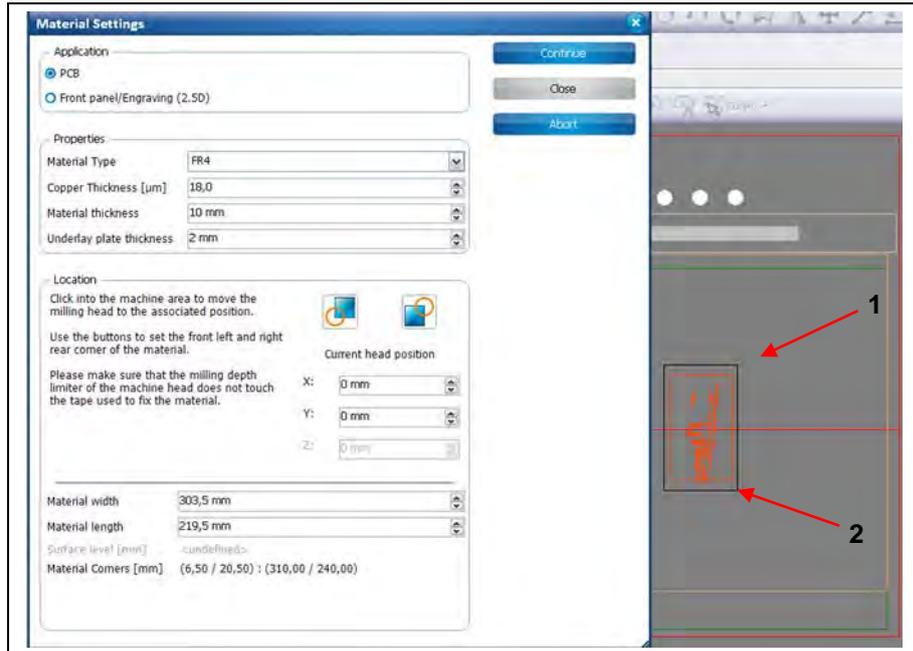


PCB is selected as default.

Note

2. Enter “18” µm into the field \Copper Thickness\.
3. Enter “0.15” mm into the field \Material Thickness\.
4. Define the processing area:
 - a) Move the “Material Settings” dialog off to the side.
 - b) Click on the position in the machining view that represents the right corner of your material:

Fig. 235: Defining the right corner



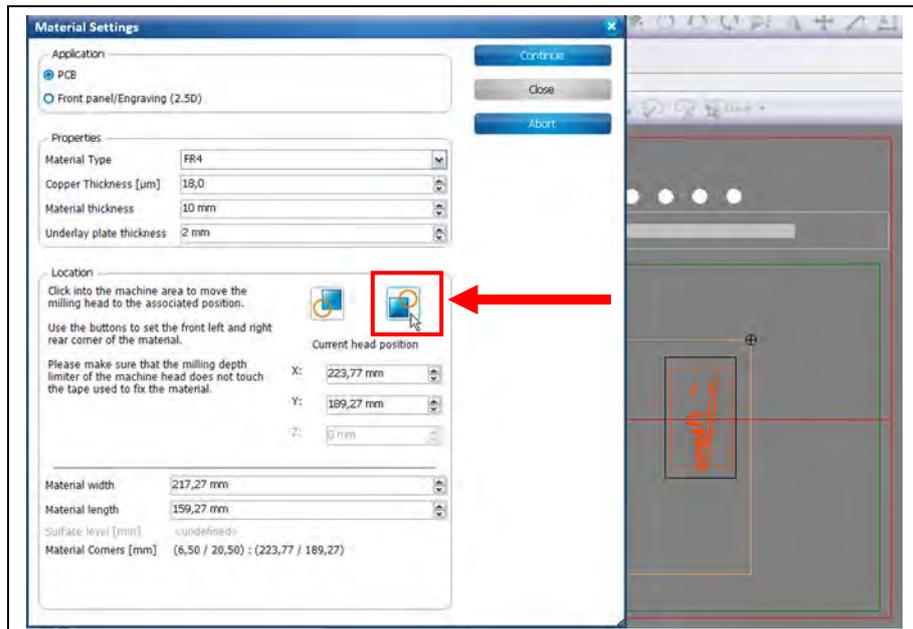
/1/ Click on machining area

/2/ Job

➔ The machine head moves to this position.

c) Click on the corresponding icon in the "Material Settings" dialog:

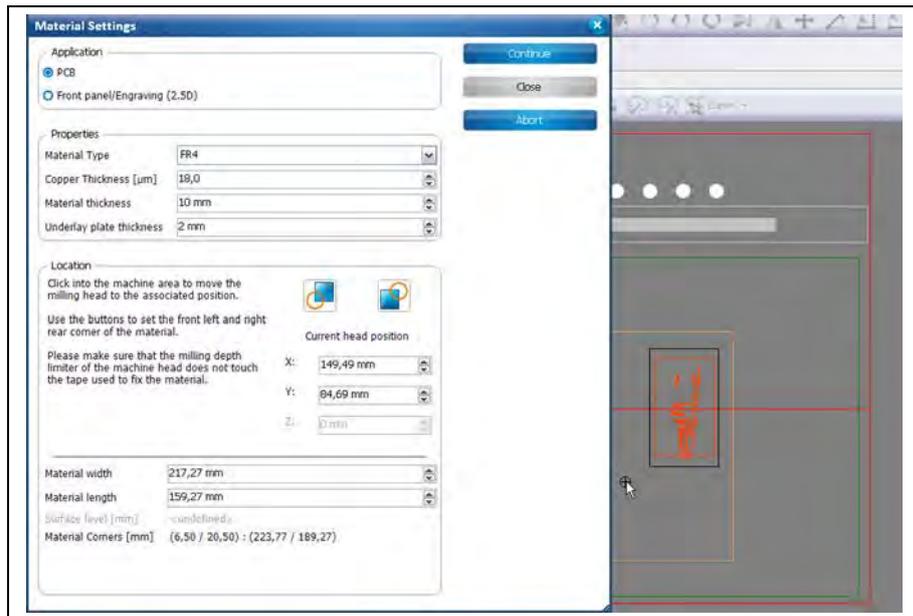
Fig. 236: Click on the icon



➔ The machining area is adjusted.

d) Click on the position in the machining view that represents the left corner of your material:

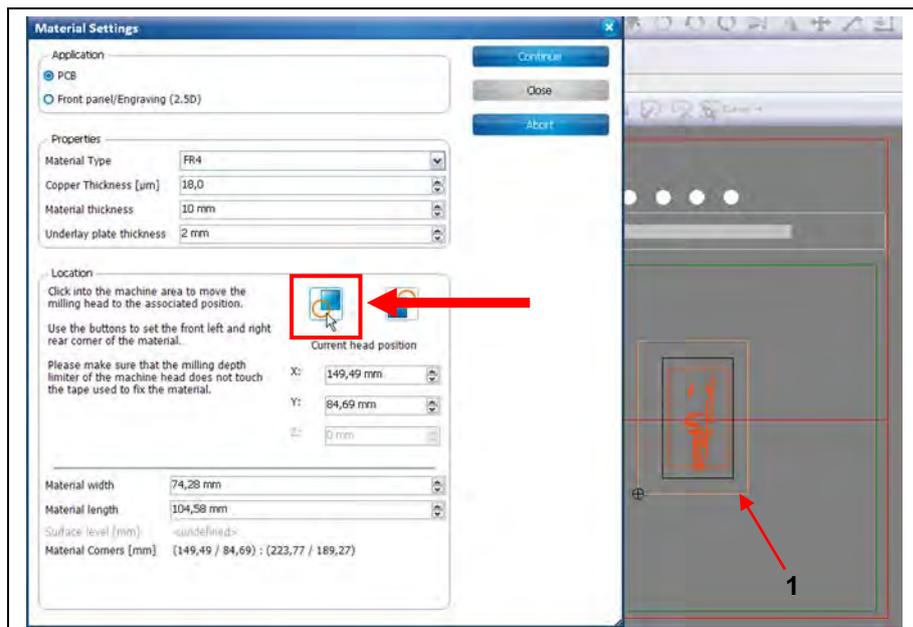
Fig. 237: Defining the left corner



➔ The machine head moves to this position.

e) Click on the corresponding icon in the "Material Settings" dialog:

Fig. 238: Defined processing area



/1/ Defined processing area

➔ The processing area has been fit to the material.

5. Click on [Continue].

◆ The material settings were entered.

Phase “Placement”

The job cannot be placed again at this stage as the position has already been defined by the flexible material in the previous steps.

1. Confirm the current placement by clicking on [OK].

Phase “DrillingUnplated”

- ➔ The cutouts for aligning the prepreg for bonding are drilled.

Phase “ContourRouting”

- ➔ The cutout for the flexible section is cut out.

Phase “Board Production Finished”

- ➔ A message informs you that the processing is finished.
- ◆ Processing of the prepreg material is finished.

6.5 Bonding the individual physical layers

The flexible material is bonded to the rigid FR4 material using the prepreg material.



We recommend the LPKF MultiPress S system for this process step.

Note

- Bonding the physical layers
- 1. Stack the physical layers produced in the previous steps in the following order:
 - I. **Flexible material**
→ Ensure that the milled side faces downwards!
 - II. **Prepreg material**
→ Align the prepreg material to the flexible material. Use the fiducials in the flexible material as reference.
 - III. **FR4 material**
→ Align the FR4 material to the flexible material. Use the reference hole system as reference.



Insert a strip of corresponding size into the cutout of the FR4 material (e.g. the material previously cut out). This supports the flexible material while bonding and achieves better results.

Note

2. Bond the materials with constant temperature and constant pressure.
- ◆ The physical layers are bonded.

6.6 Contour routing the bonded PCB

The following steps are necessary for contour routing the bonded PCB:

- i. Inserting a production phase
- ii. Creating toolpaths
- iii. Loading the tool magazine and assigning tools to holder positions
- iv. Starting the processing

Inserting a production phase

An additional production phase for reading the fiducials has to be inserted before the toolpaths for contour routing the PCB are generated.



Note

The cutouts in the prepreg and FR4 material enable to read the fiducials in the flexible material from the Top side (the flexible material facing downwards).

■ Inserting a production phase



Note

Ensure that no existing production phase is selected in the "Toolpath" pane, otherwise no new production phase can be created!

1. Click on Machining > Production phase...

➔ The following dialog is displayed:

Fig. 239: Creating a new phase



2. Enter the name "ReadFiducials" for the new phase.
3. Select the processing step "Read fiducials top".



Note

You can enter this processing step manually if it does not exist.

4. Enter "8" for processing order.
5. Activate the check box "The phase contains toolpath data".
6. Click on [Apply].

- ➔ The new phase ReadFiducials is inserted before the ContourRouting phase in order to read the fiducials from the Top side of the PCB.



You can check in the “Phases” section of the “Toolpath” pane whether the new phase has been inserted.

Note

- ◆ The new production phase has been inserted.

Deleting toolpaths

All toolpaths previously generated for processing the individual physical layers have to be deleted. They are not needed for routing the the PCB.

■ Deleting toolpaths

1. Click on the “Toolpath” pane.
2. Click on the plus symbol next to “Phases”.

- ➔ The phases are displayed.



The individual phases are marked with a plus symbol if they contain toolpaths.

Note

3. Click on the plus symbols of all phases in order to display all toolpaths.

- ➔ The toolpaths are displayed.

4. Select a toolpath.
5. Press the right mouse button in order to open the context menu.
6. Select “Delete” in the context menu.

- ➔ The selected toolpaths are deleted.

7. Repeat steps 4 to 6 for all other toolpaths.

- ◆ The toolpaths are deleted.

Creating toolpaths

The PCB has to be put on the processing area of the ProtoMat machine with the flexible material facing downwards to enable contour routing. The PCB is thus processed from the Top side.



Note

Check in the “Toolpath” pane whether the “ContourRouting” phase is processed on the Top side.

Follow these steps:

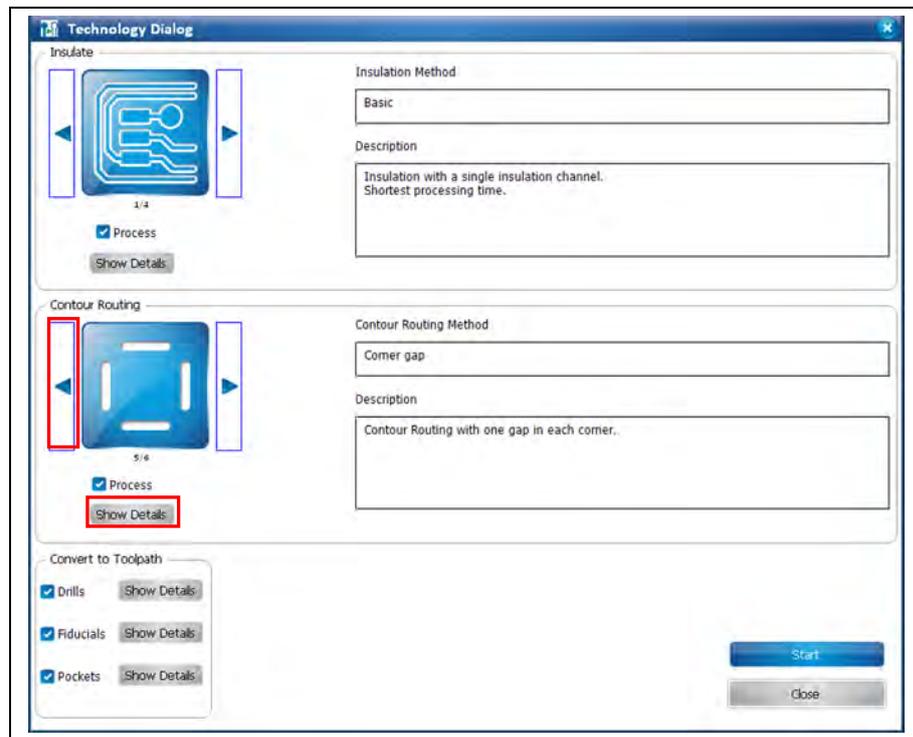
1. Click on the desired phase using the right mouse button.
2. Click on “Edit” in the context menu.
3. Make sure that the check box “The data are mirrored” is unchecked. If not, uncheck the mark.

■ Creating toolpaths

1. Click on Toolpath > Technology dialog...

➔ The following dialog is displayed:

Fig. 240:
Technology
Dialog



Note

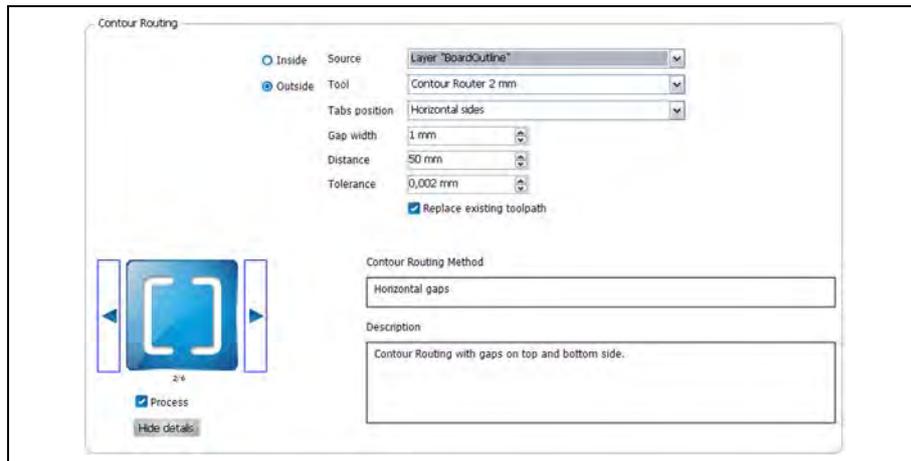
In the technology dialog, the settings for generating toolpaths can be modified by clicking on the [Show details] buttons.

For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.

2. Click on the left arrow icon in the “Contour Routing” section until the “Horizontal gaps” contour routing method is displayed.
3. Click on [Show details].

➔ The details for contour routing are displayed:

Fig. 241: Contour routing details



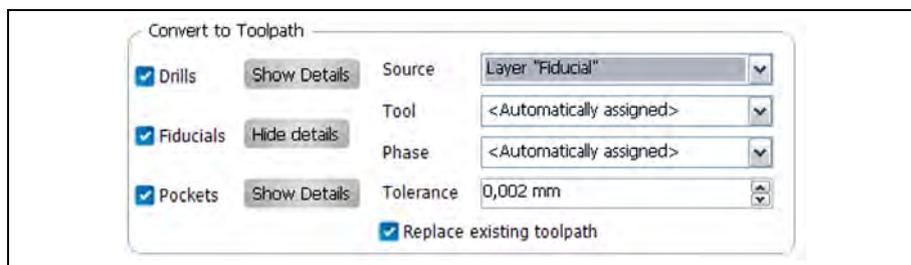
4. Select the "BoardOutline" layer in the "Source" selection list.
5. Activate the {Outside} radio button if it is not active.
6. Click on [Hide details].

➔ The details are hidden.

7. Click on [Show Details] next to "Fiducials".

➔ The details for fiducials are displayed.

Fig. 242: Fiducials - details



8. Select the "Fiducial" layer in the "Source" selection list.
 9. Click on [Hide details].
- ➔ The details are hidden.
10. Deactivate the following functions by clicking on the corresponding check marks:
 - Insulate
 - Drills
 - Pockets
 11. Click on [Start].
- ➔ The toolpaths are created and the computation results are displayed.
- ◆ The toolpaths are created.

Loading the tool magazine and assigning tools to holder positions



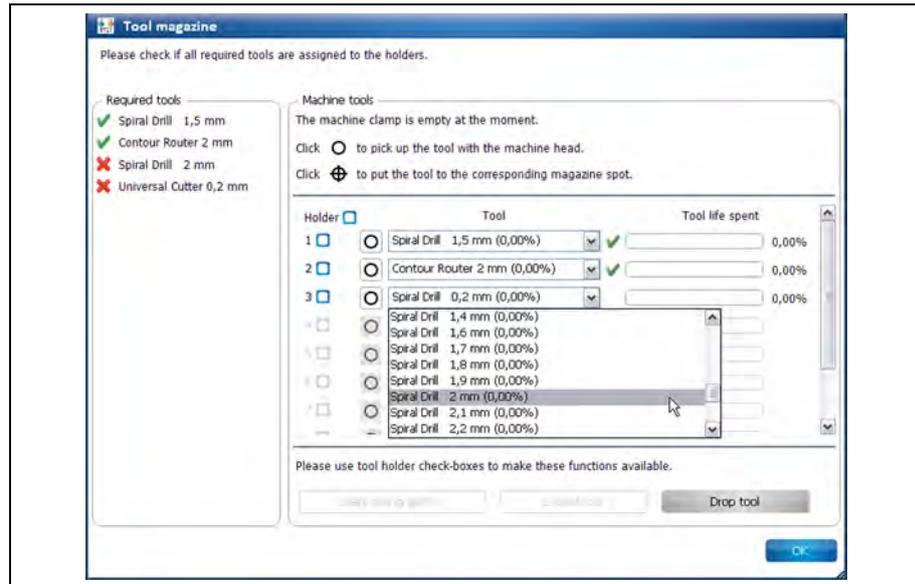
This section is only relevant, if you use a ProtoMat with automatic tool change (S63 or S103).

Note

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

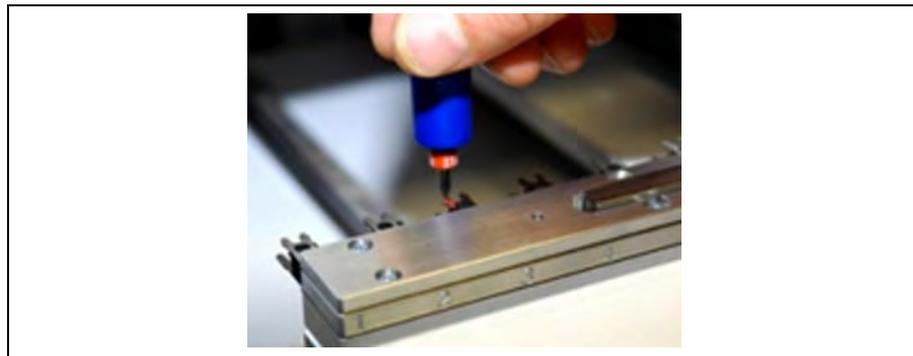
Fig. 243: Tool magazine



➔ The tools required for the job are displayed. Tools required for the job that are missing are marked by a red "X".

2. Insert the required tools into the tool holders of the machine:

Fig. 244: Inserting a tool



3. Assign the tools to the corresponding positions in the dialog.

➔ The tool holders of the machine are loaded:

Fig. 245: Loaded tool holder

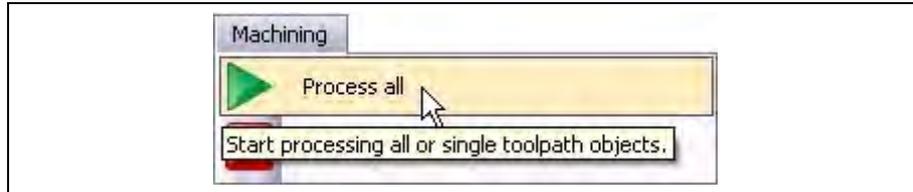


◆ The tools are inserted into the tool holders and assigned accordingly.

Starting the processing

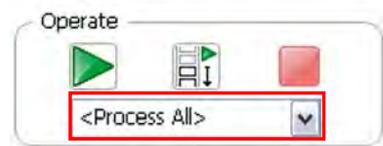
1. Click on Machining > Process all.

Fig. 246:
Machining > Process all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

Once you have started the processing, the ProtoMat machine executes the job in individual phases. The phases are displayed in messages:

➔ Phase "MountMaterial"

1. Mount the PCB onto the machine's processing area with the flexible material facing down (Top side up).



Note

Ensure that the positioning holes are at the right font corner.

2. Fasten the material to the processing area using adhesive tape.
3. Click on [OK].

Phase “Material Settings”

➔ The following dialog is displayed:

Fig. 247: Material Settings

■ Entering the material settings

1. Enter the correct values for the material used.

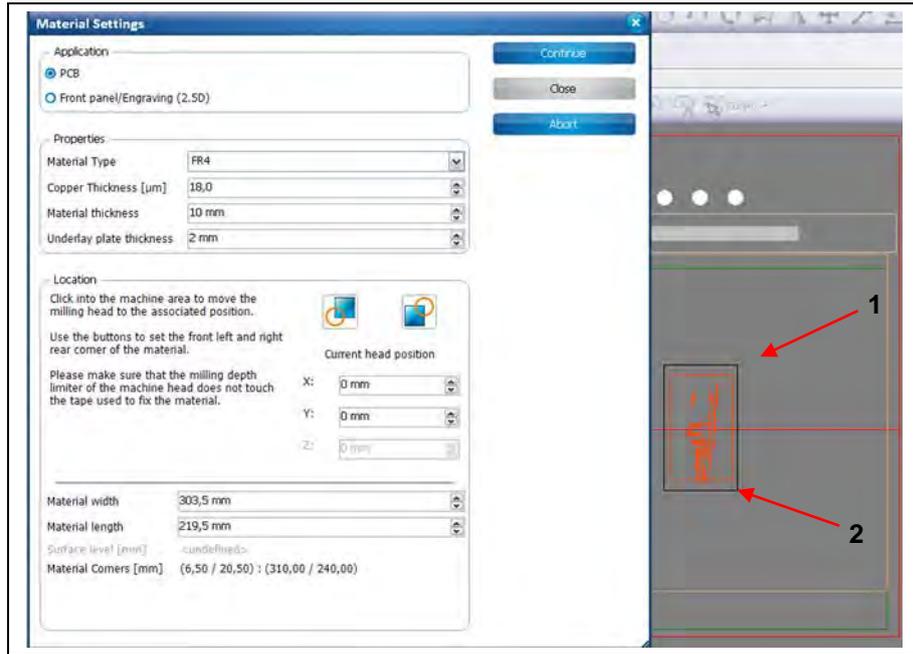


PCB is selected as default.

Note

2. Enter “18” µm into the field \Copper Thickness\.
3. Enter ”0.15” mm into the field \Material Thickness\.
4. Define the processing area:
 - a) Move the “Material Settings” dialog off to the side.
 - b) Click on the position in the machining view that represents the right corner of your material:

Fig. 248: Defining the right corner



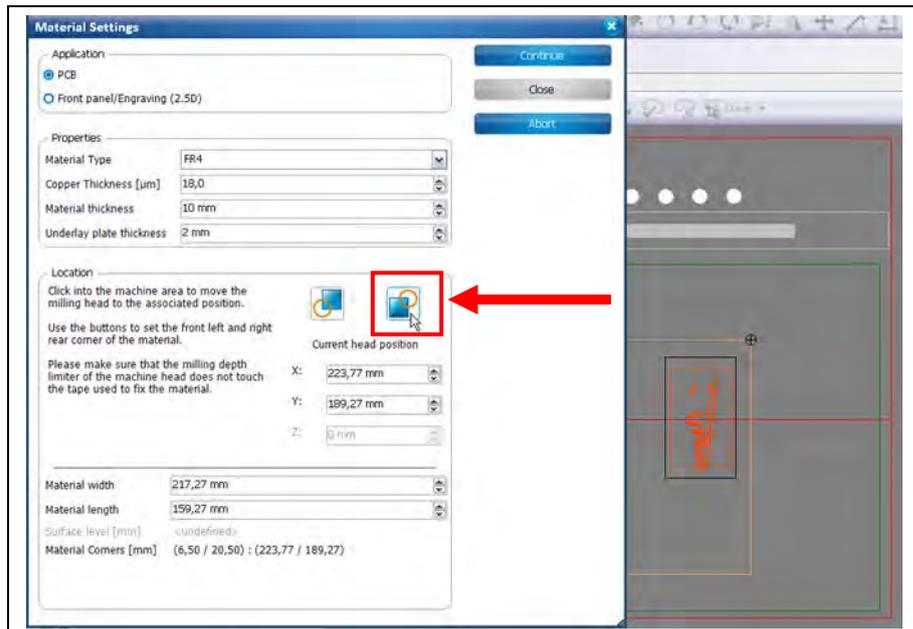
/1/ Click on machining area

/2/ Job

➔ The machine head moves to this position.

c) Click on the corresponding icon in the "Material Settings" dialog:

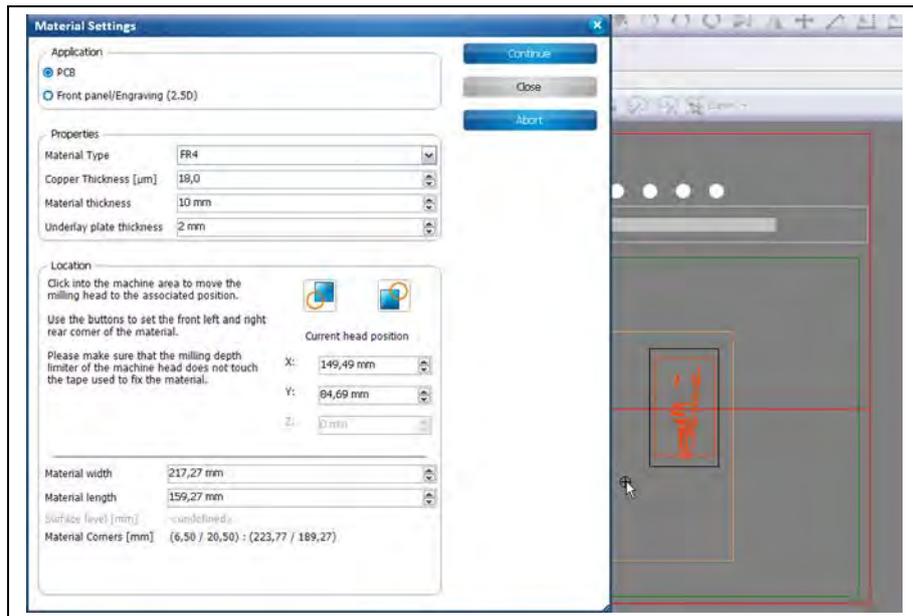
Fig. 249: Click on the icon



➔ The machining area is adjusted.

d) Click on the position in the machining view that represents the left corner of your material:

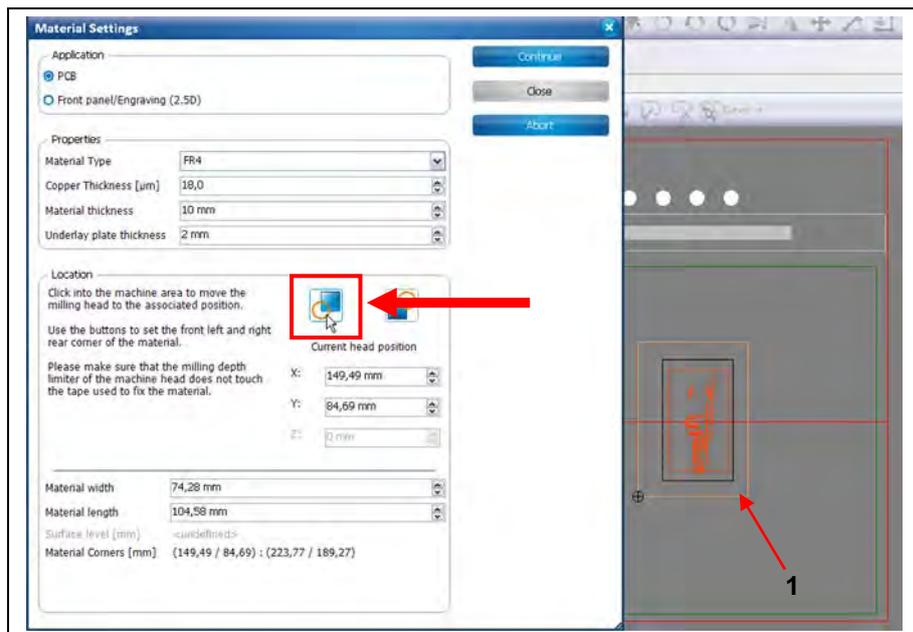
Fig. 250: Defining the left corner



➔ The machine head moves to this position.

e) Click on the corresponding icon in the “Material Settings” dialog:

Fig. 251: Defined processing area



/1/ Defined processing area

➔ The processing area has been fit to the material.

5. Click on [Continue].

◆ The material settings were entered.

Phase “Placement”

The job cannot be placed again at this stage as the position has already been defined by the flexible material in the previous steps.

1. Confirm the current placement by clicking on [OK].

Phase “Read Fiducial”

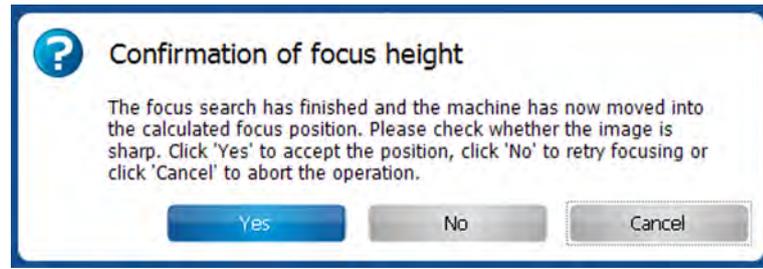
➔ The camera searches for the fiducials in the flexible material.



Note

If the fiducial search is performed for the first time (after having started CircuitPro) the camera is performing an autofocus five times.

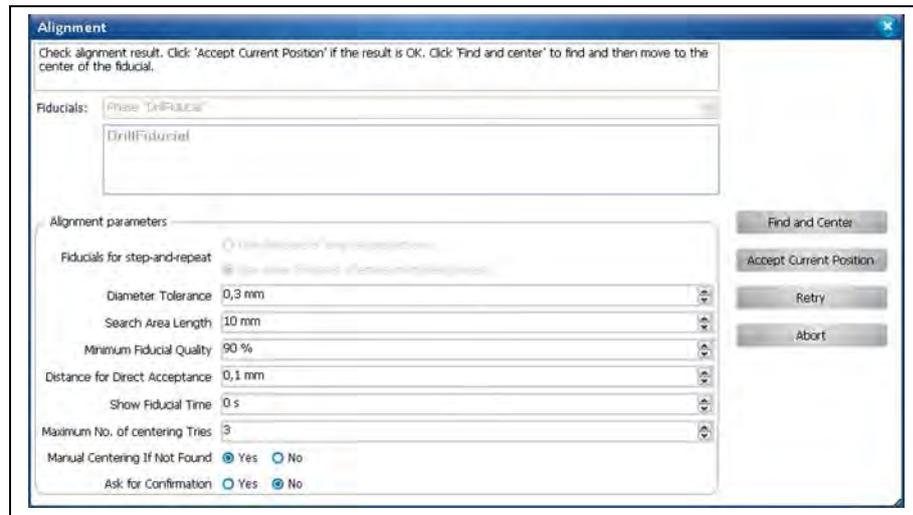
Afterwards the following message is displayed which prompts you to confirm the focus height:



If the material is placed at nearly the same position as before, the positions of the fiducials are recognised automatically.

The following dialog is displayed if the fiducials have not been recognized automatically:

Fig. 252:
Alignment



1. Enlarge the search area by increasing the value of the field \Search Area Length\.
2. Restart the search by clicking on [Start].
3. Repeat steps one and two if necessary.



Note

Enlarging the search spiral increases the time required for searching the fiducials. Try to put the material at the same position as before when turning the material over (if this is not predetermined by reference pins).

Phase “ContourRouting”

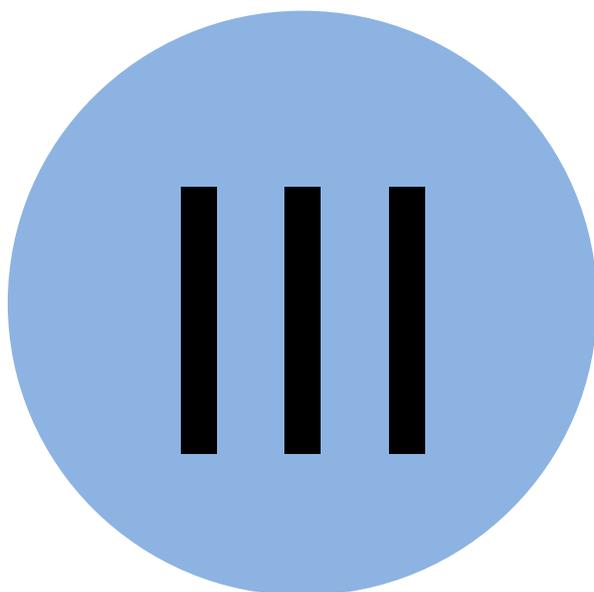
- ➔ The bonded PCB is cut out.

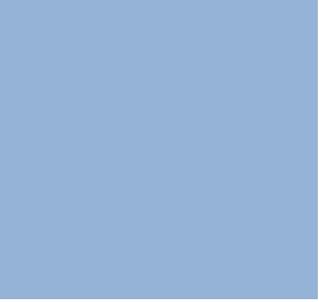
Phase “Board Production Finished”

- ➔ A message informs you that the processing is finished.
- ◆ The bonded PCB has been cut out.

Creating the flexi-rigid PCB is finished successfully.

Basics





CircuitPro: Basic CAM operations



1 CircuitPro: Basic CAM operations

This tutorial shows you how to process basic CAM operations in CircuitPro.

The following steps are necessary to complete the tutorial successfully:

- i. Executing the Process Planning Wizard
- ii. Importing the Gerber files
- iii. Importing the drill file
- iv. Establishing rubout boundaries
- v. Inserting fiducials
- vi. Creating toolpaths



Tip

You are able to perform these steps by using the virtual machine. This enables you to work without a real machine. Click on Machining > Connect > Virtual.

1.1 Executing Process planning wizard



CircuitPro must be running.

Note

■ Executing the Process planning wizard

1. Click the icon "Process planning Wizard" on the toolbar:

Fig. 253: Icon
Process Planning
Wizard

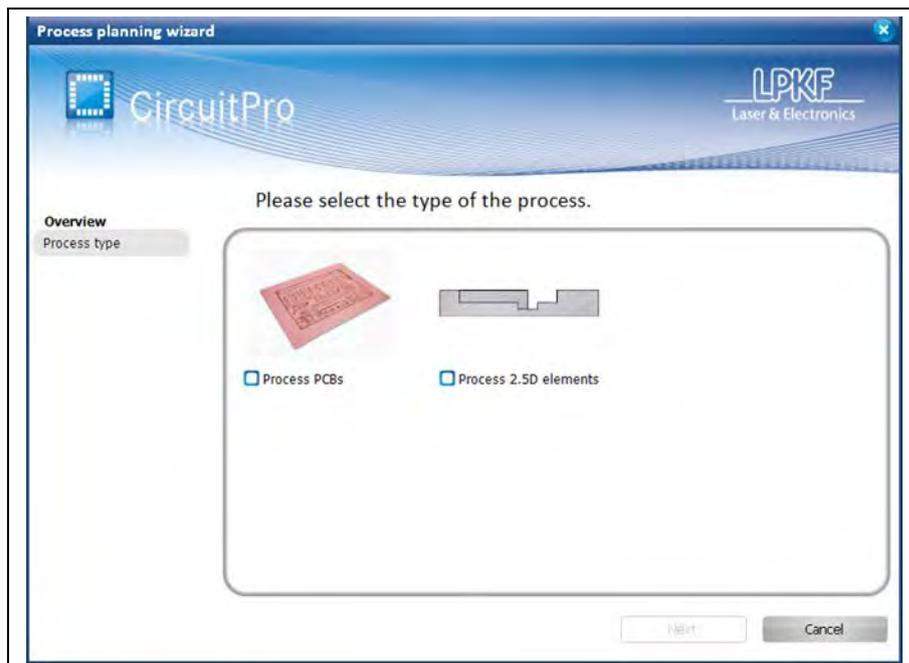


Or

1. Click on Wizards > Process planning wizard...

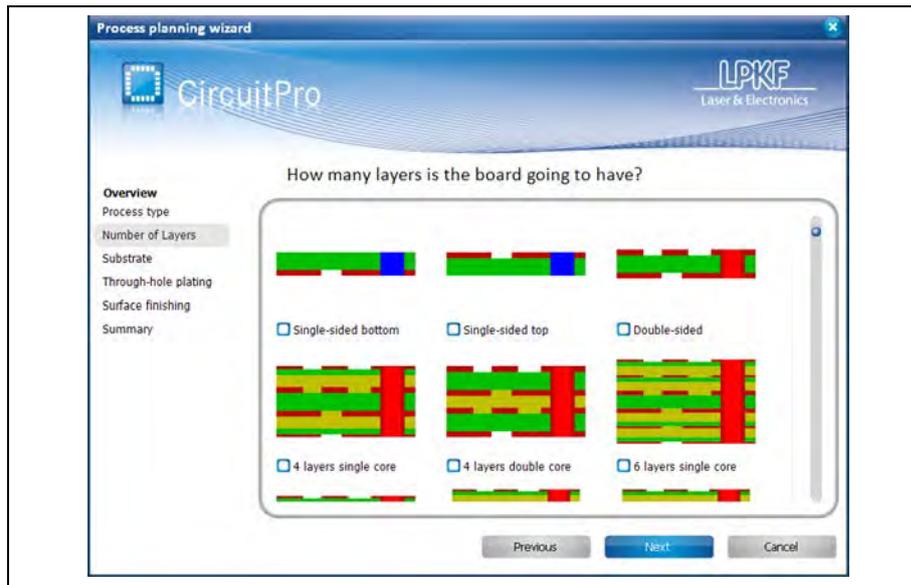
➔ The process planning wizard starts:

Fig. 254: Process
planning wizard



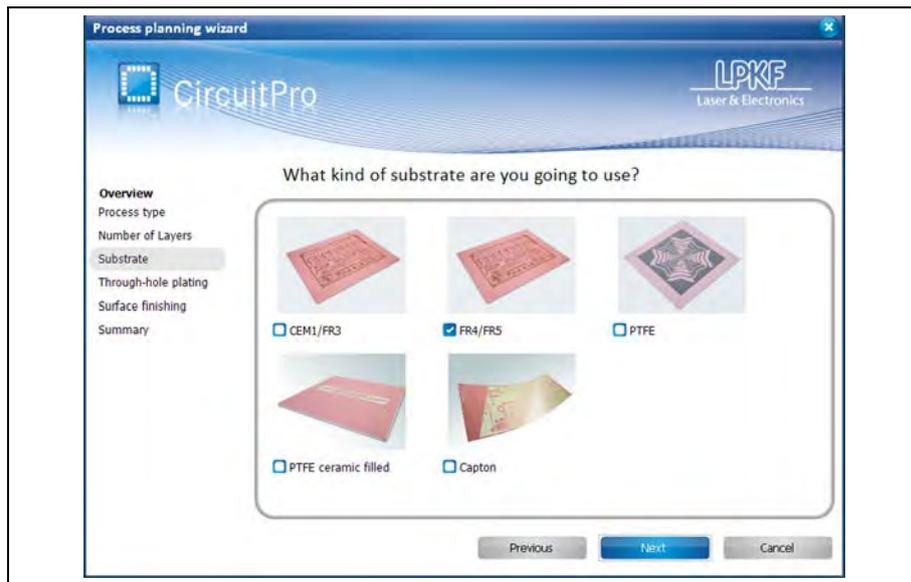
2. Select the type of process.
3. Click on [Next].

Fig. 255: Process planning wizard



4. Select the number of copper layers you are going to use.
5. Click on [Next].

Fig. 256: Selecting the substrate



6. Select which substrate you are going to use.



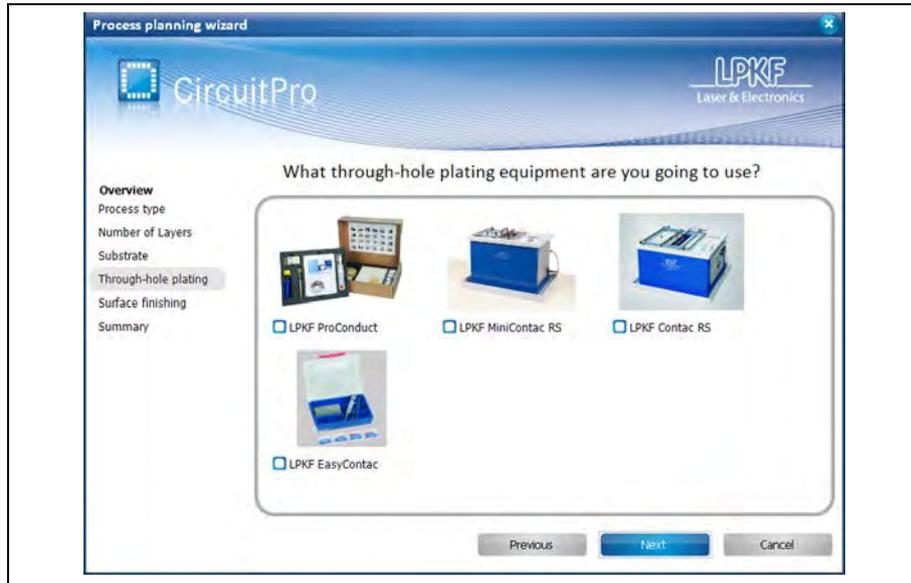
Note

The options will be presented based on what you told the system in the Equipment configuration wizard during in the initial installation.

You can start the Equipment configuration wizard any time to customize the available systems.

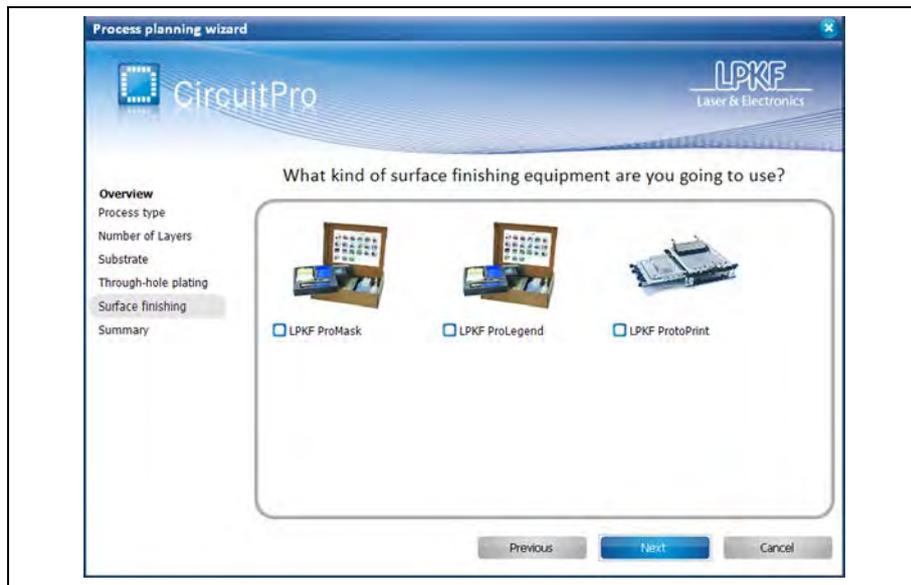
7. Click on [Next].

Fig. 257: Through hole plating equipment



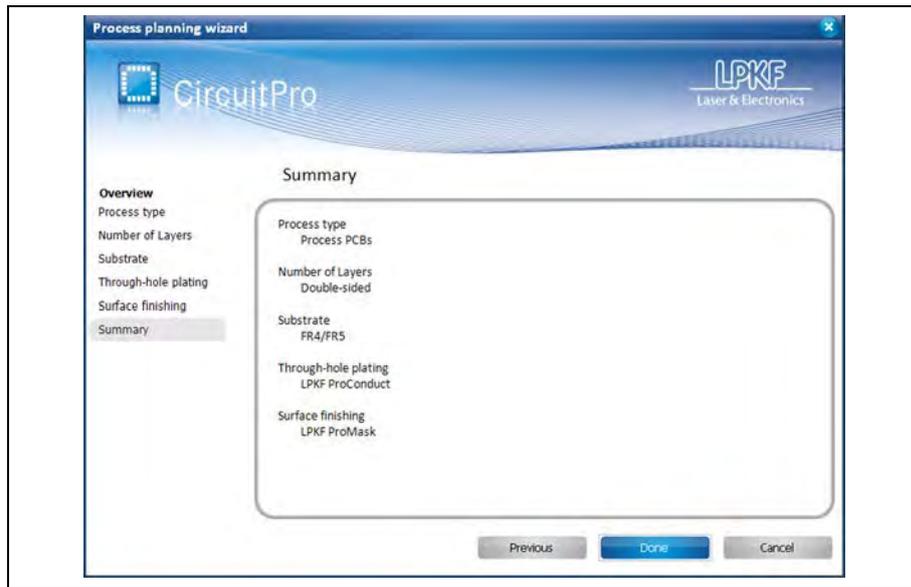
8. Select which through-hole plating method will be used for PCB production.
9. Click on [Next].
10. Select if you are using any surface finishing (silkscreen or solder mask):

Fig. 258: Surface finishing



11. Click on [Next].

Fig. 259:
Summary



12. Verify the configuration and click on [Done].

- ◆ The Process planning wizard is finished.

1.2 Importing the Gerber files



Tip

The LPKF tutor data are stored in “My Document\LPKF Laser & Electronics\ LPKF CircuitPro 1.5\Example Data”.

■ Importing the Gerber files

1. Click on the icon “Import” on the toolbar:

Fig. 260: Icon import

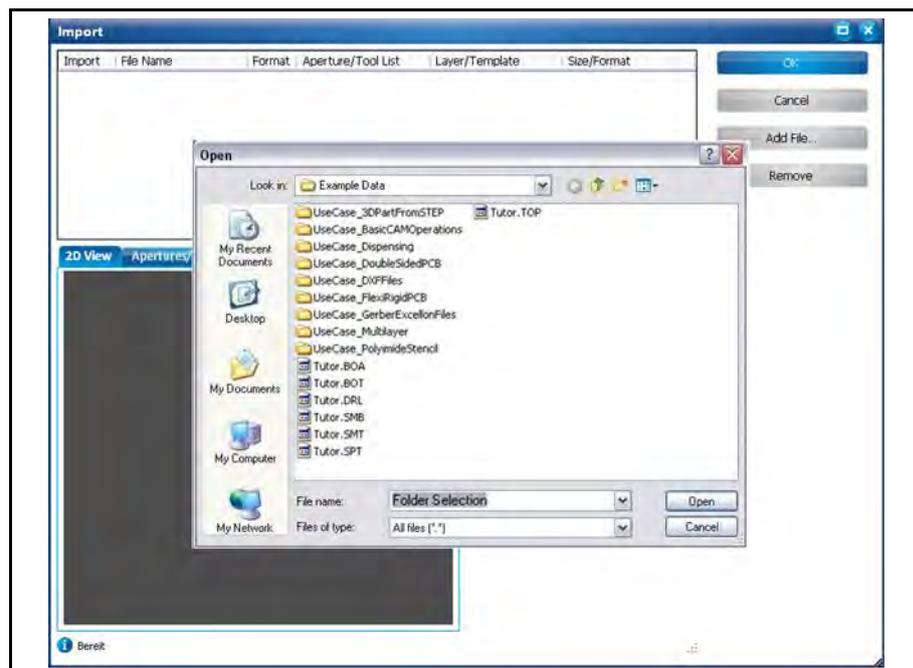


Or

1. Click on File > Import...

➔ The following dialog is displayed:

Fig. 261: Import dialog

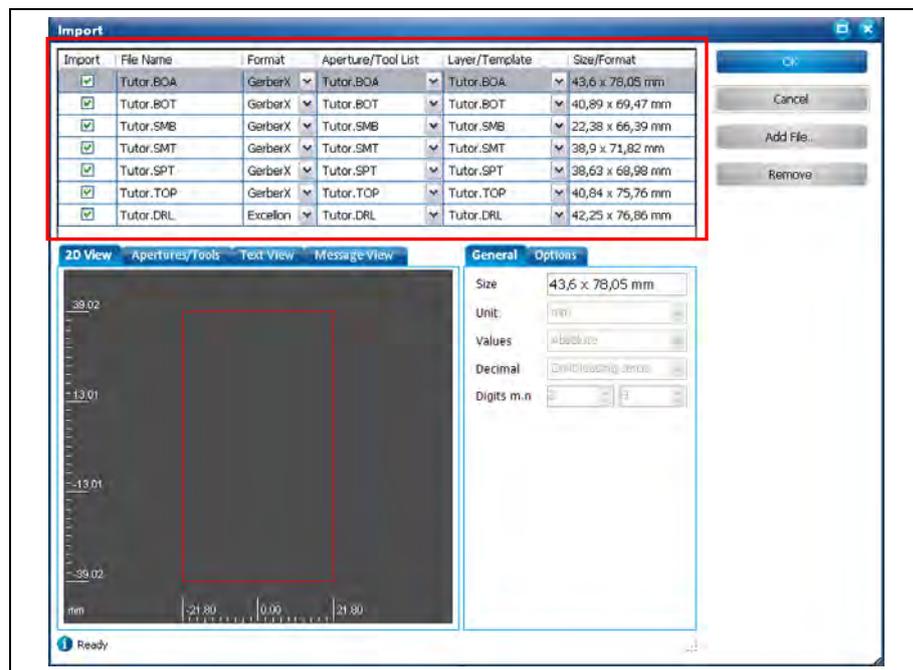


2. Select all required data according to the table below.
3. Click on [Open].
4. Assign the imported files to their corresponding layers according to the following table:

File	Layer
.BOA	Board Outline
.BOT	Bottom Layer
.SMB	Solder Mask Bottom
.SMT	Solder Mask Top
.SPT	Solder Paste Top
.TOP	Top Layer

- Therefore, click on the column "Layer/Template" and choose the corresponding layer in the drop down list:

Fig. 262:
Assigned layers



Note

Instead of manually assigning the individual files to the layers, you can activate the options "Use layer name" and "Apply to all Gerber files".

If a file contains layer names these are automatically assigned. Please note that this is only available for Gerber files. All other files require assigning the layers manually via the drop-down menu.

- Click on [OK].

- ➔ The data are displayed in the CAM view.
- ◆ The Gerber files were imported.



Note

For more information about importing Gerber files, please refer to the tutorial "Processing Gerber and Excellon files".

1.3 Importing the drill file



This step can be completed at the same time as importing the Gerber files.

Note

■ Importing the drill file

1. Click the icon "Import" on the toolbar:

Fig. 263: Icon import

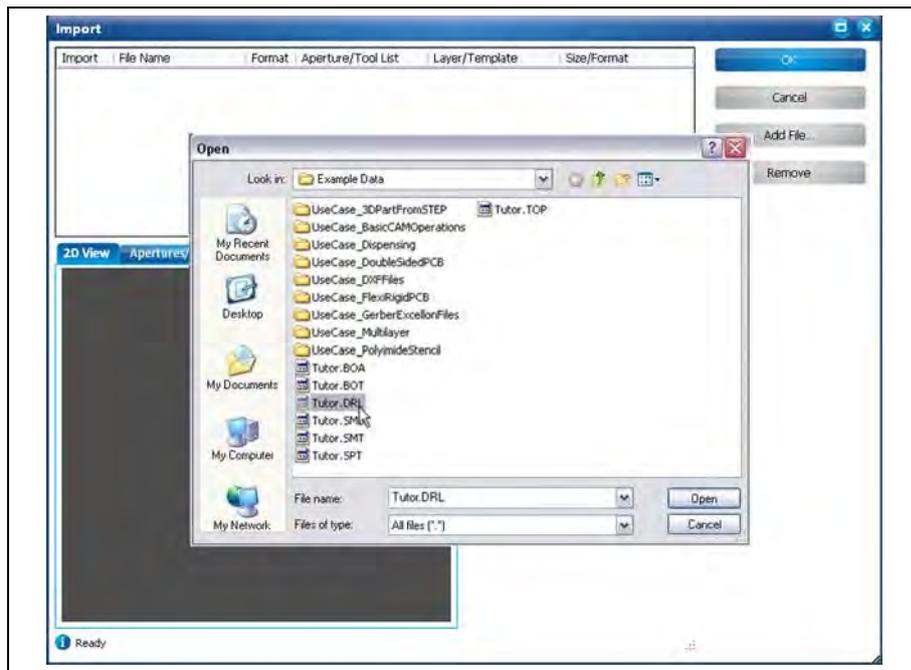


Or

1. Click on File > Import....

➔ The following dialog is displayed:

Fig. 264: Import dialog



2. Select the drill file "Tutor.DRL".
3. Click on [Open].

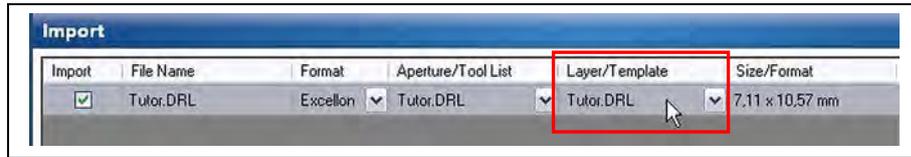


In this document, the DRL file is the Excellon drill file. This may also be a text file, (.txt. extension) depending on your layout package.

Note

4. Change the layer/template column to DrillPlated or DrillUnplated:

Fig. 265:
Changing the
layer column



Note

This depends on whether you are using a through-hole plating system:

- If so, use the layer DrillPlated.
- If not, use the layer DrillUnplated.

This will also affect when the holes are drilled:

- DrillPlated is drilled on the bottom side of the board.
- DrillUnplated is drilled from the top side of the board.

If this is a single sided board, use DrillUnplated.

In the graphic display, the correct view of the drills, you should see the different sized apertures and the correct location of your drills.

- If the display is correct, continue with step 6.
- If the display is not correct, continue with step 5.

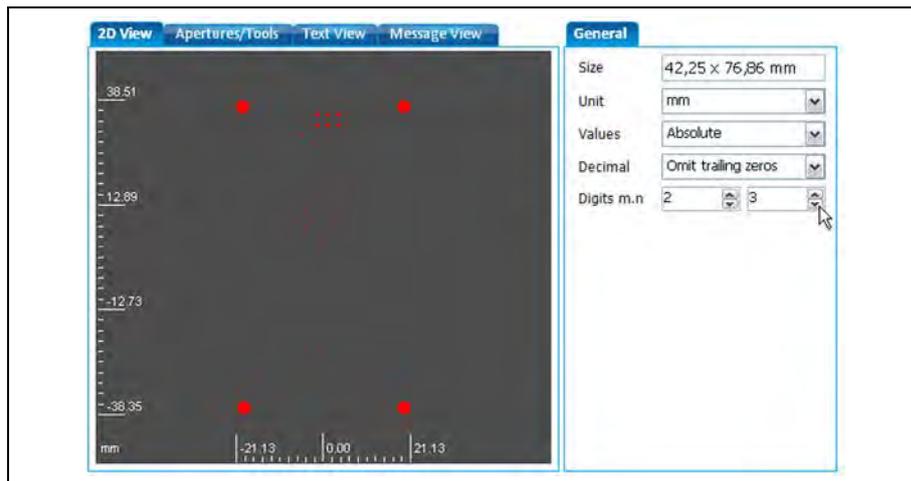
5. Check the settings in the tab "General". If necessary change the settings until your design is displayed correctly.



Note

For details about the settings, please refer to the tutorial "Processing Gerber and Excellon files".

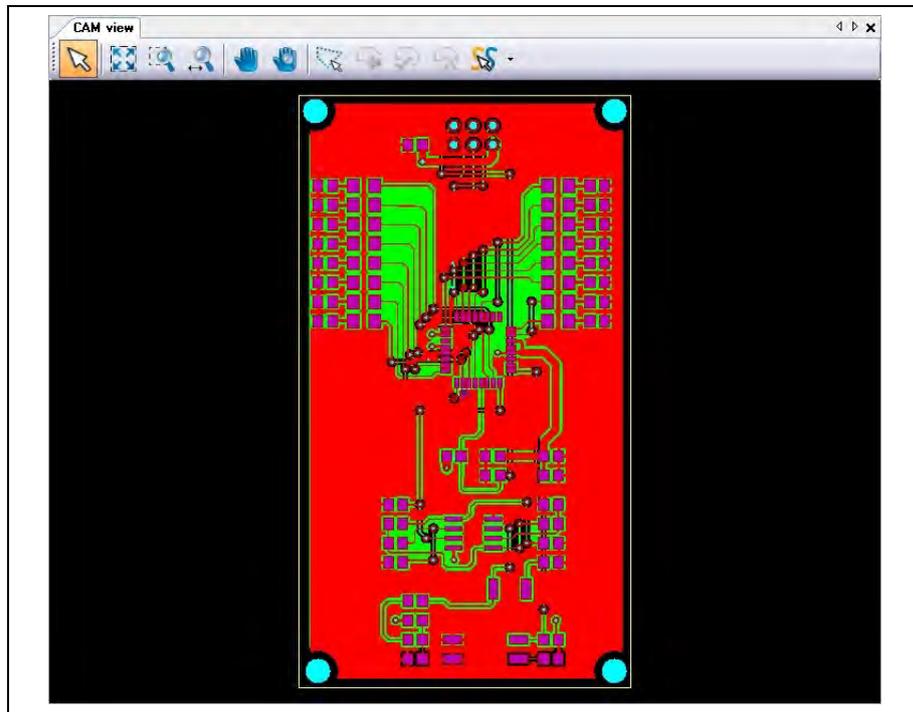
Fig. 266: Display
correct



6. Click on [OK].

➔ The design is now displayed on screen:

Fig. 267: Design
displayed in CAM
view



◆ The drill file was imported.

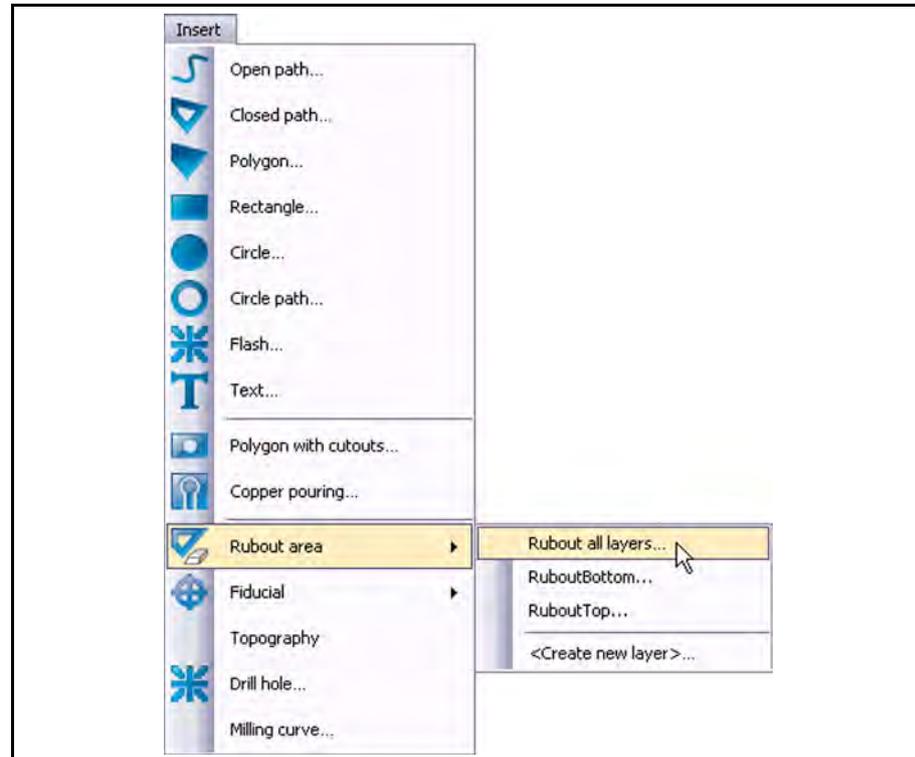
1.4 Inserting a rubout area (optional)

You can remove excess copper if desired. Therefore you have to insert rubout areas.

■ Inserting a rubout area

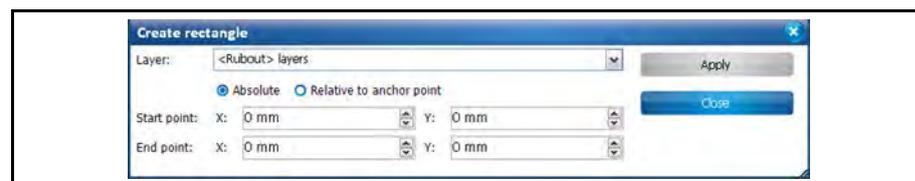
1. Click on Insert > Rubout area > Rubout all layers...:

Fig. 268: Rubout all layers



➔ The following dialog is displayed:

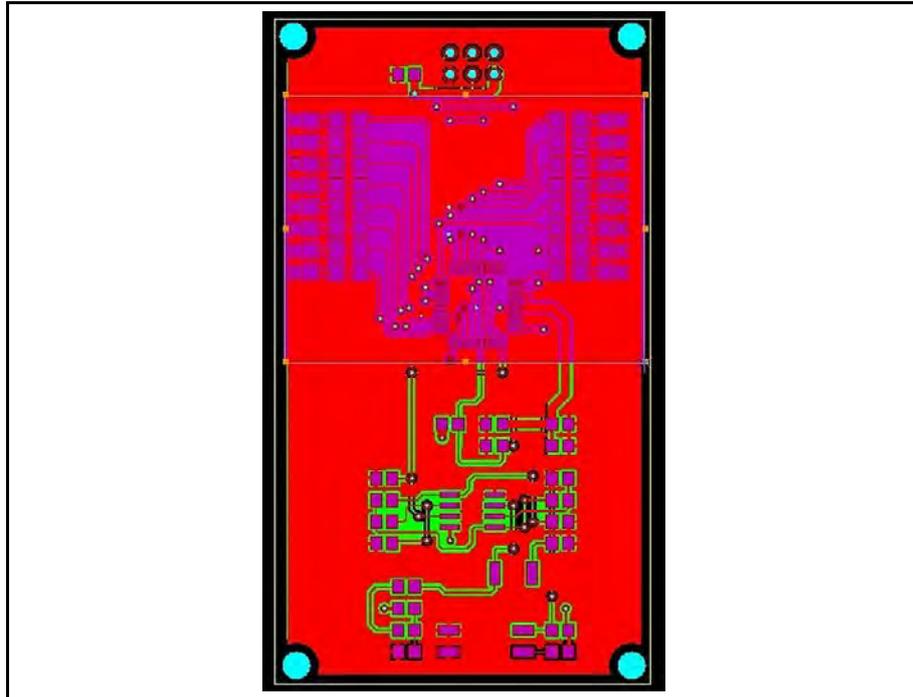
Fig. 269: Create rectangle



2. Move the "Create rectangle" dialog off to the side.
3. Single click on one corner of the board, or on the desired location.
4. Single click on the opposite corner of the board, or on the desired location.
5. Click on [Close].

➔ The rubout area is created on your board:

Fig. 270: Rubout
area



If you wish to create a rectangle by a specific size, you may use the coordinate system in the rectangle window.

Note

◆ The rubout area was inserted.

1.5 Inserting fiducials (optional)

For aligning the top and bottom sides of the circuit board you need fiducials. Fiducials are optical marks on the surface of the circuit board with a defined diameter of 1.5 mm.

The fiducials are drilled into the board and are recognised by the cameras of the ProtoMat systems.



Note

For working with fiducials you need the camera system for fiducial recognition.



Tip

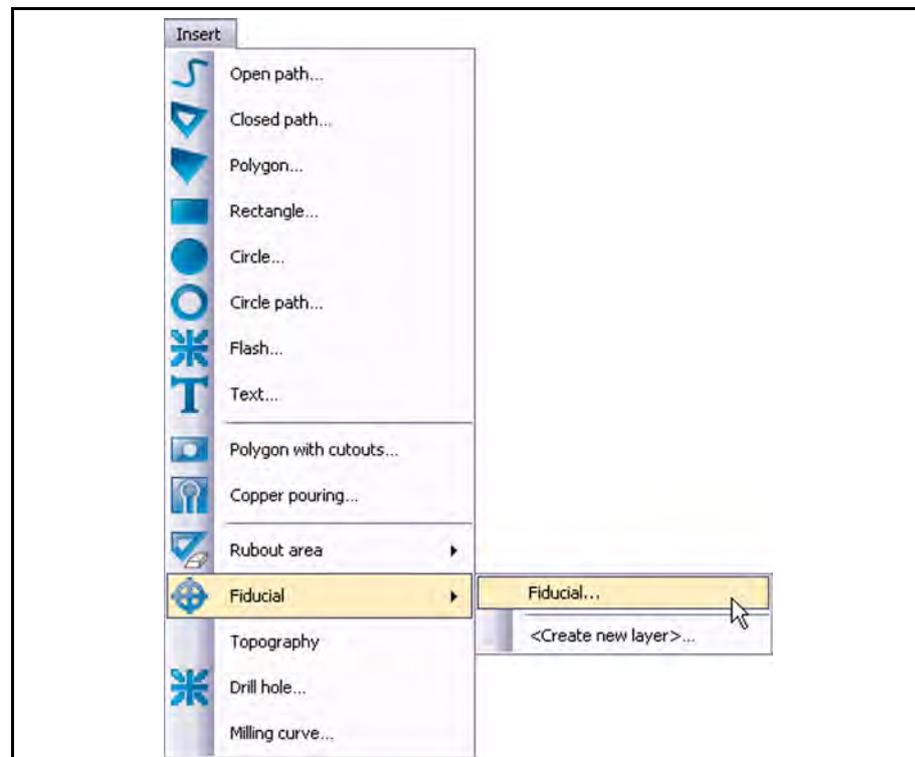
Ideally you insert four fiducials for aligning the top and the bottom sides.

You are also able to work with two fiducials. In this case you have to insert them diagonally into the layout.

■ Inserting fiducials

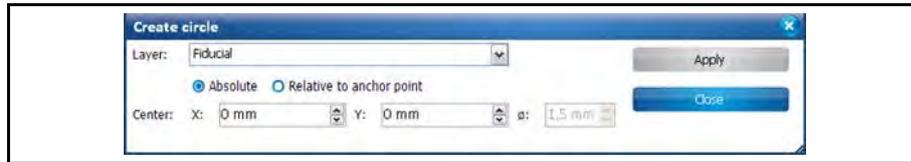
1. Click on Insert > Fiducial > Fiducial:

Fig. 271: Insert > Fiducial



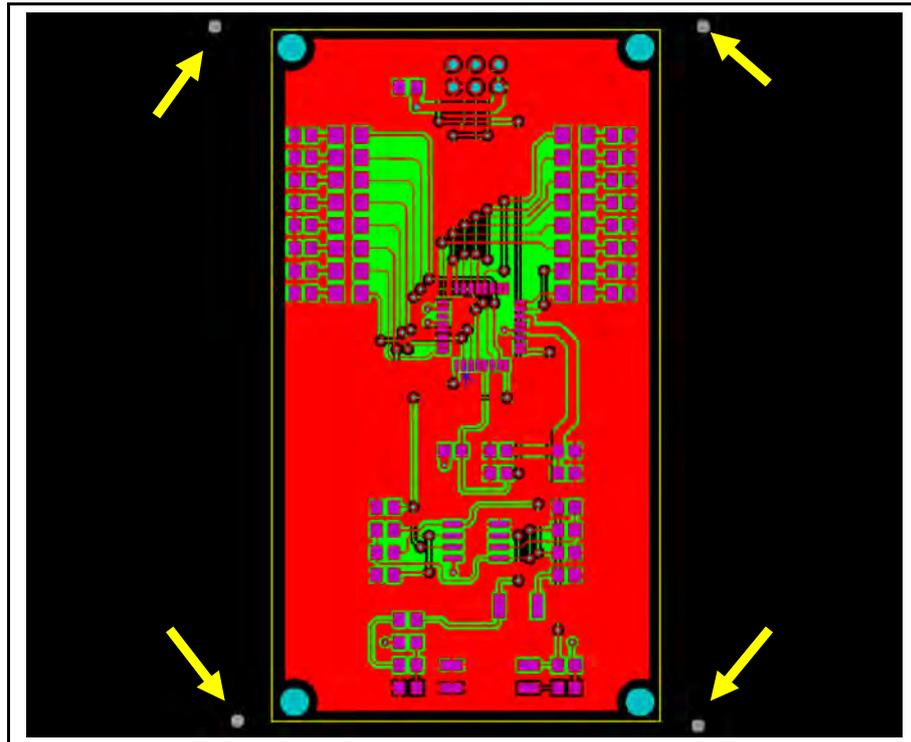
➔ The following dialog is displayed:

Fig. 272: Create circle



2. Move the “Create circle” dialog off to the side.
3. Left click where you want to place each fiducial hole:

Fig. 273: Example for placing fiducials



Note

Best results are just off each corner of the board.

4. Click on [Close].
- ◆ The fiducials were inserted.

1.6 Creating toolpaths

The Technology Dialog is used for creating toolpaths in CircuitPro.



Note

There are various options how to create toolpaths etc. For more detailed information please refer to the chapter "Technology Dialog" in the compendium.

You will pass through following steps:

- i. Selecting the insulation type
- ii. Selecting the contour routing type
- iii. Creating the toolpaths

1. Click the icon "Technology Dialog" on the toolbar:

Fig. 274: Icon Technology Dialog

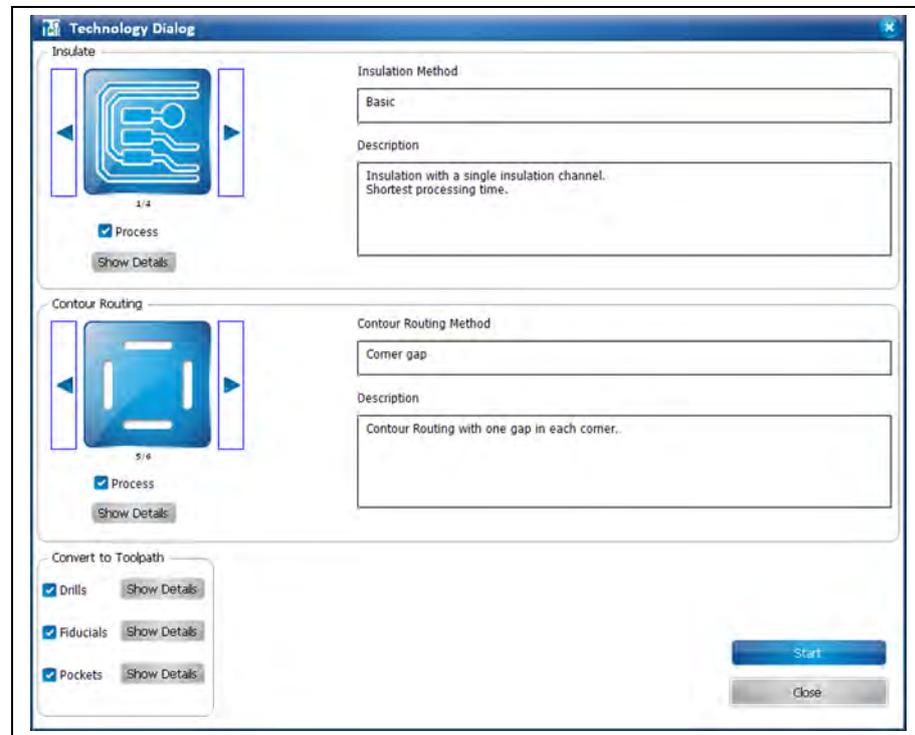


Or

1. Click on Toolpath > Technology Dialog...

➔ The following dialog is displayed:

Fig. 275: Technology Dialog



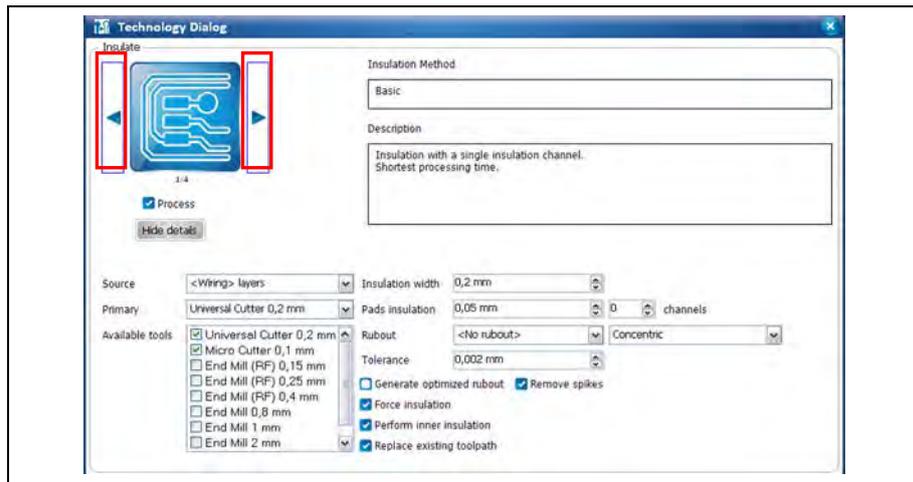
- Selecting the insulation type.
 1. Click on the right/left arrow keys in the section “Insulation” to choose the insulation type you desire.
 2. Click on [Show Details] for more information about the insulation type.

The following insulation types are available:

a) Basic insulation

→ The method “Basic insulation” is isolating the traced pads from the copper only.

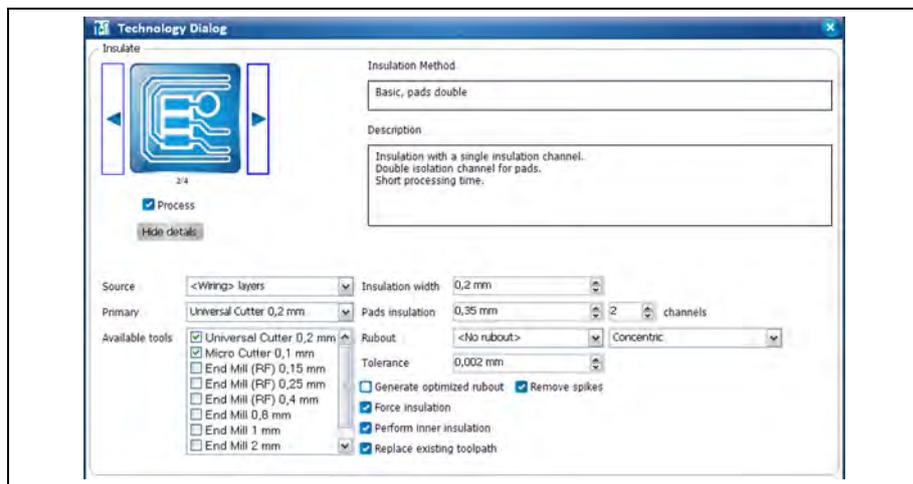
Fig. 276: Basic insulation



b) Basic insulation, pads double

→ The method “Basic insulation, double pads” isolates the traces and double isolates the pads.

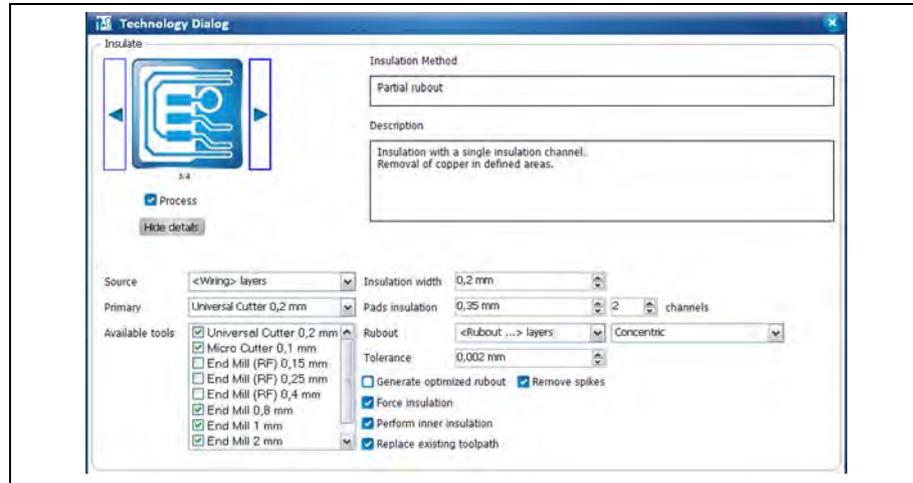
Fig. 277: Basic insulation, pads double



c) Partial rubout

→ The method “Partial rubout” isolates the traces and pads and removes any excess copper within the rubout boundary inside of your board.

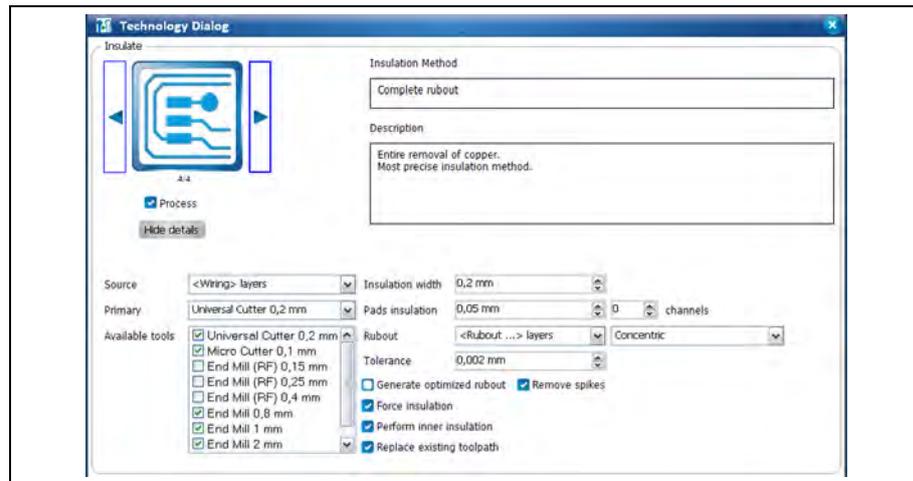
Fig. 278: Partial rubout



d) Complete rubout

→ The method “Complete rubout” isolates the traces and pads along with removing the excess copper from the entire board.

Fig. 279: Complete rubout



After you have chosen the insulation method, continue with step 3.

3. Select the milling tools that you wish to use in the available tools drop-down menu by checking/unchecking the tool name.



Note

A recommended combination of tools is: Universal Cutter 0.2 mm, the End Mill 0.4 mm and the End Mill 1.0 mm.

For **RF boards** use the 0.25 mm End Mill instead of the Universal Cutter.

If your designs have smaller spacing requirements than 8 mils, use the End Mill 0.15 mm (6 mil), the End Mill 0.1 mm (4 mil) or the Micro Cutter 0.1 mm (4 mil).

If milling a **RF board**, change the primary tool!

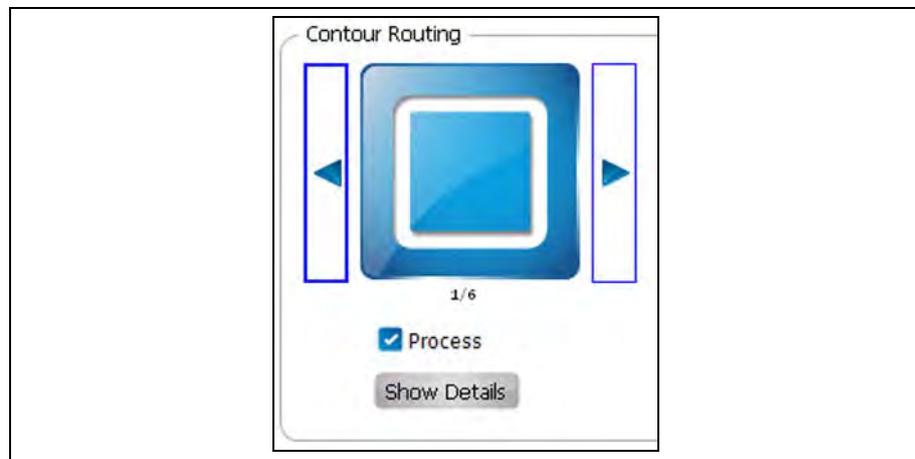
4. Click on [Hide details].
 - ◆ The insulation type has been selected.
- Selecting the contour routing type
1. Click on the right/left arrow keys in the section “Contour Routing” to choose the option you would like to cut the board out.
 2. Click on [Show Details] for more information about the contour routing types.

The following contour routing types are available:

a) Basic

→ The method “Basic” routes along the outside of your board without gaps.

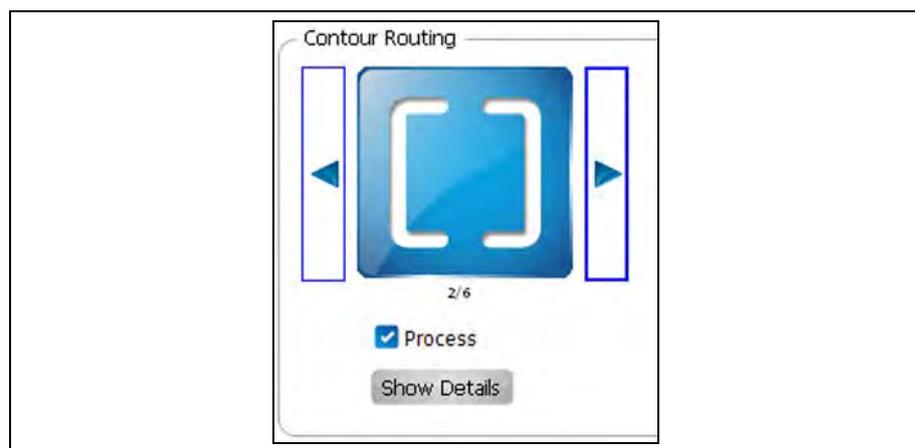
Fig. 280: Basic routing



b) Horizontal gaps

→ The method “Horizontal gaps” routes along the outside of your board with gaps on top and bottom side.

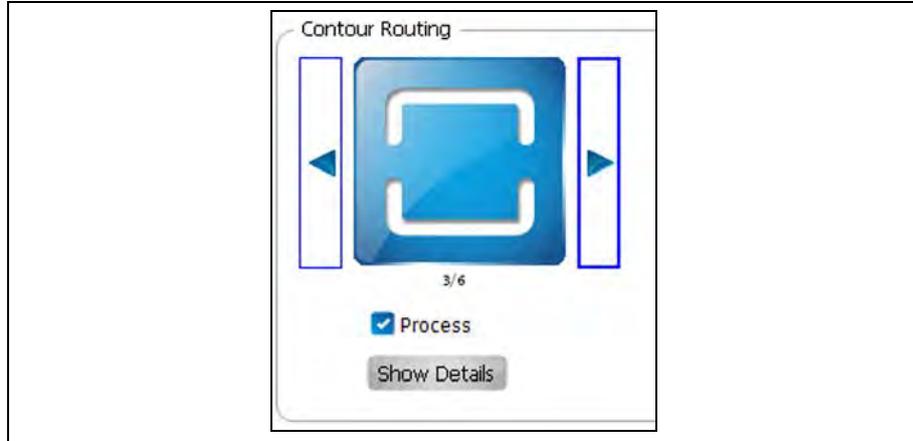
Fig. 281: Horizontal gaps routing



c) Vertical gaps

→ The method “Vertical gaps” routes along the outside of your board with gaps on the left and right side.

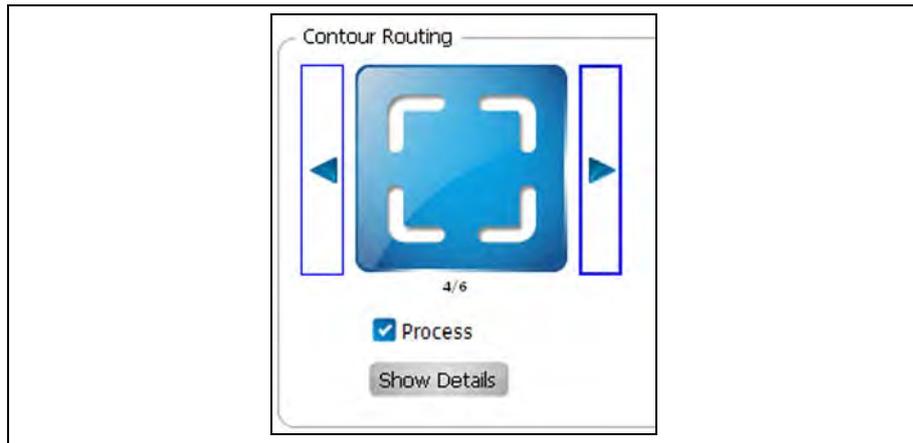
Fig. 282: Vertical gaps routing



d) Edge gaps

→ The method “Edge gaps” routes along the outside of your board leaving tabs in the center of each edge of the board.

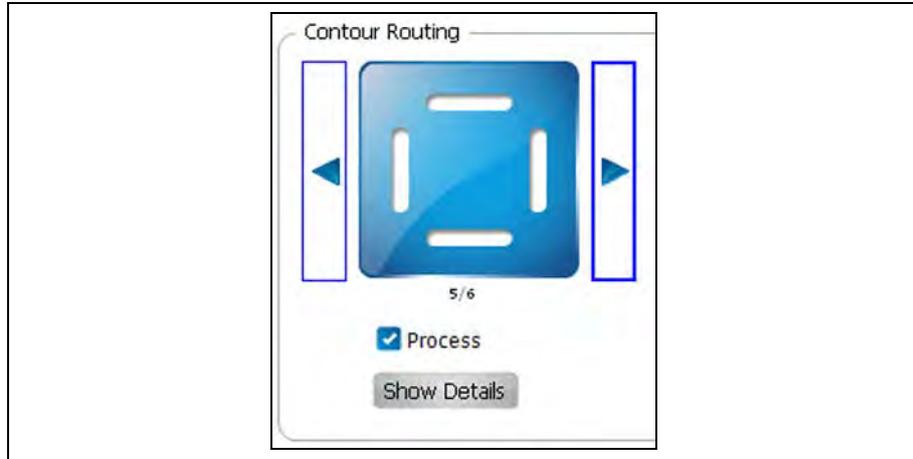
Fig. 283: Edge gaps routing



e) Corner gap

→ The method “Corner gap” routes along the outside of your board leaving tabs in the corners.

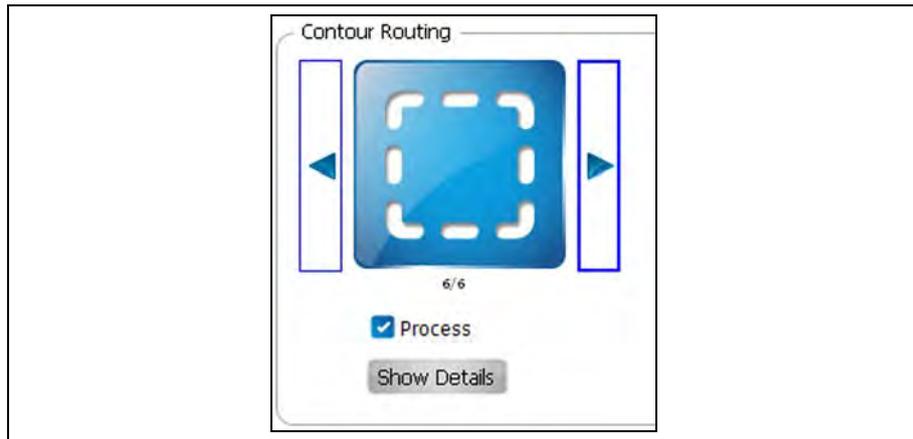
Fig. 284: Corner gap routing



f) Equidistant gaps

→ The method “Equidistant gaps” routes along the outside of your board leaving tabs at equal distance that you specify.

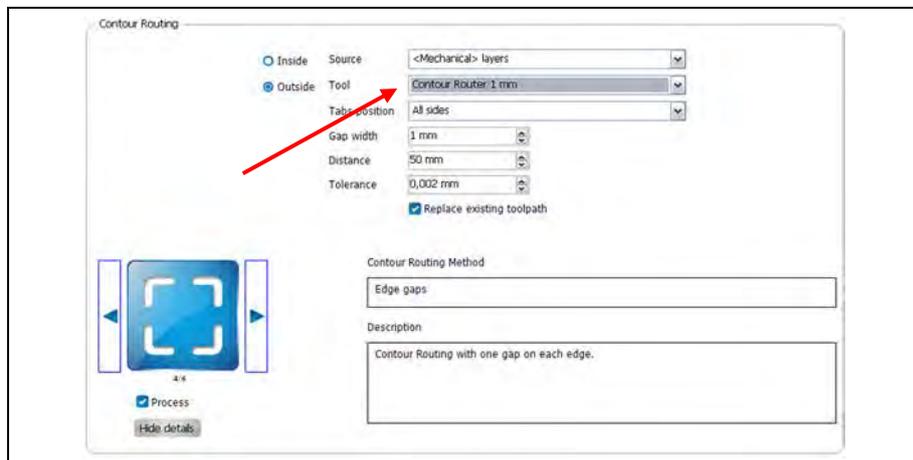
Fig. 285: Equidistant gaps routing



After you have chosen the contour routing method, continue with step 3.

3. If desired, change the tool used during the contour routing by clicking on the tool drop-down menu:

Fig. 286: Tool drop-down menu



4. Change the 2.0 mm contour router to the 1.0 mm contour router.



Note

If your design is larger than 2x3 inches, the 2.0 mm Contour Router is recommended.

If your design is smaller than 2x3 inches, the 1.0 mm Contour Router is recommended.



Note

The 1.0 mm Contour Router is also recommended if the board outline is not a traditional rectangle. This helps minimize the risk of a rounded corner.

- ◆ The contour routing type has been selected.

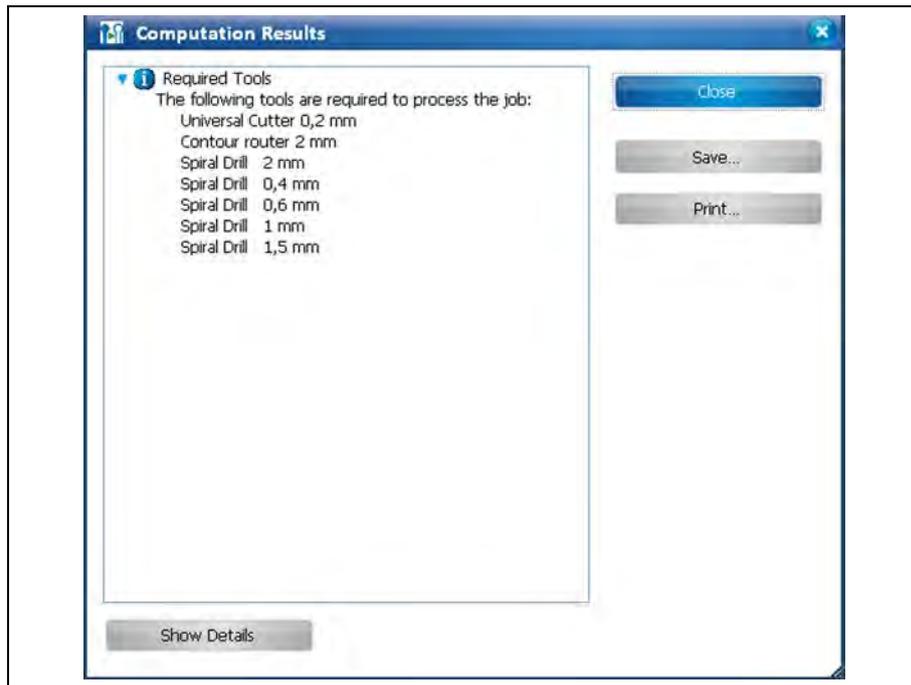
- Creating the toolpaths

After you have chosen your insulation method and your contour routing type, you have to start to create the toolpaths.

1. Click on [Start].

- ➔ The software will now create all toolpaths and identify which drills shall be used. When finished, you will get a report of which tools are used.

Fig. 287:
Computation
Results



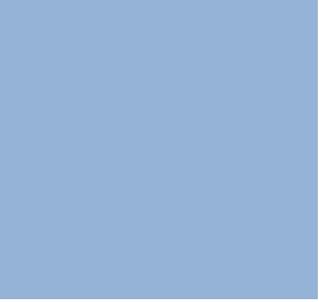
2. Click on [Close], to close the report window.

- ◆ The toolpaths have been created.



Note

Now your design is ready to be produced as a PCB. Continue with the steps described in the tutorial "CircuitPro - Basic machining operations".



CircuitPro: Basic machining operations



2 CircuitPro: Basic machining operations

This tutorial shows you how to process basic machining operations (ProtoMat S series) in CircuitPro and is based on using a 2-sided template without Through-Hole Plating. Although the document follows a certain set of steps, the steps could vary in order depending on the template chosen by the user.

Therefore you have to perform following steps:

- i. Switching to Machining view
- ii. Loading the tool magazine
- iii. Starting board production



Note

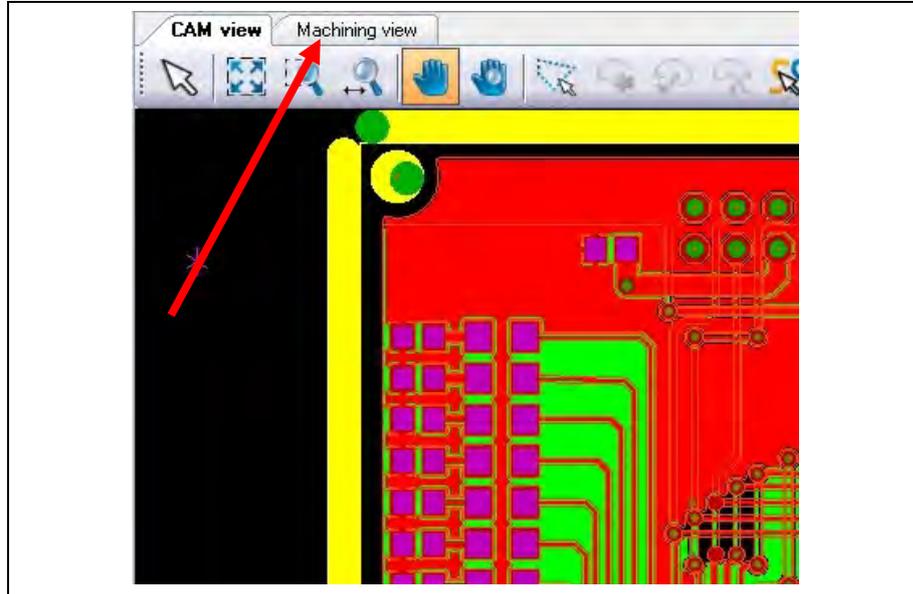
CircuitPro must be running and the machine must be connected.

2.1 Switching to Machining view

To start the board production, you have to switch to machining view first.

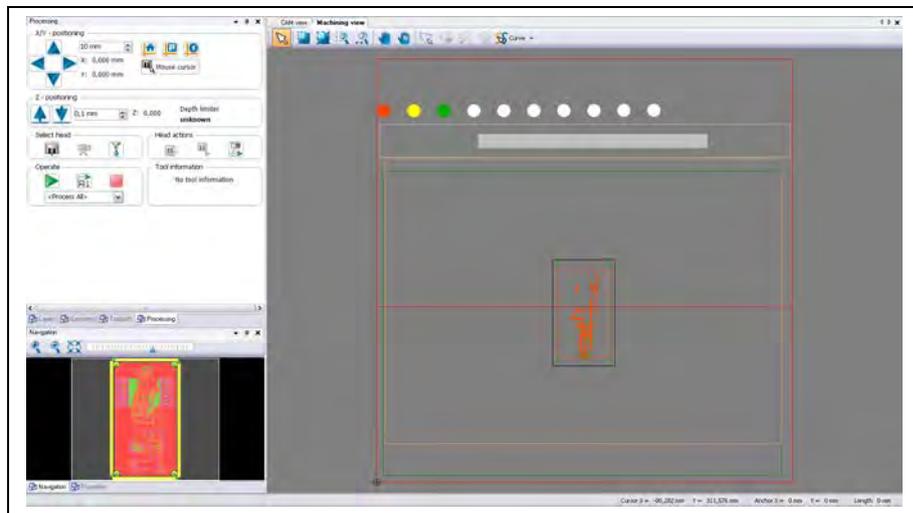
1. Click on the tab "Machining view":

Fig. 288: Tab
"Machining view"



- ➔ You will see your board on your screen:

Fig. 289:
Machining view



2.2 Loading the tool magazine

In this step you will learn, how to load the tool magazine and assign the tools to their tool holder positions.



This chapter is relevant, if you use a ProtoMat with automatic tool change (S63 or S103).

Note

■ Loading the tool magazine and assigning the tools to holder positions

1. Click on the icon "Tool magazine" on the toolbar:

Fig. 290: Icon Tool magazine

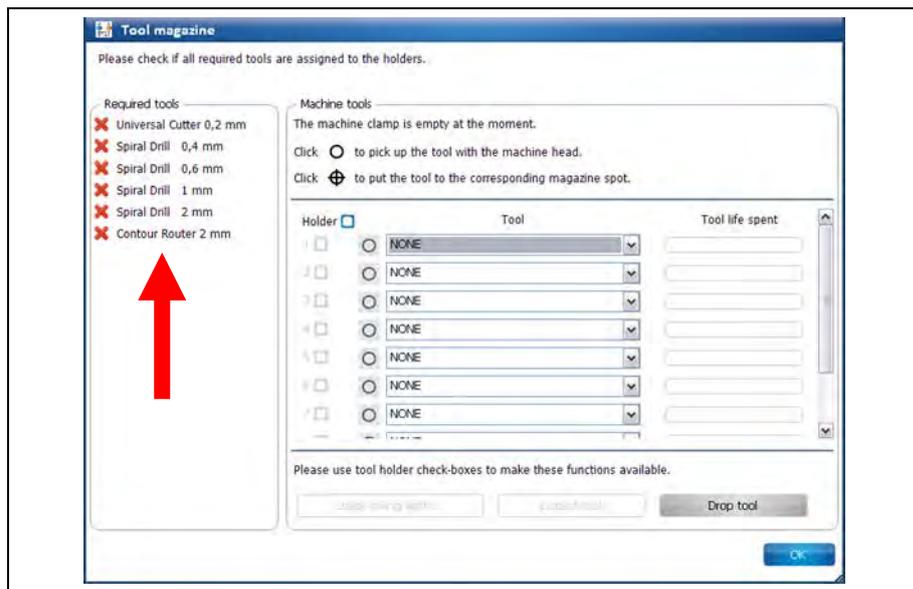


Or

1. Click on Edit > Tool magazine...

➔ The following dialog is displayed:

Fig. 291: Tool magazine

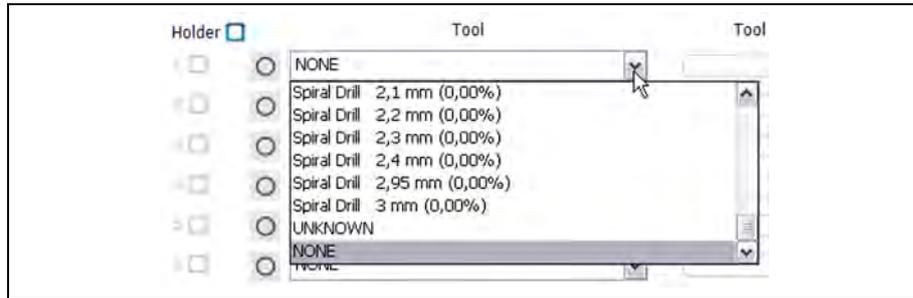


The required tools for producing the board are listed on the left of the screen (see arrow).

2. Click on the drop-down menu in each individual tool position.

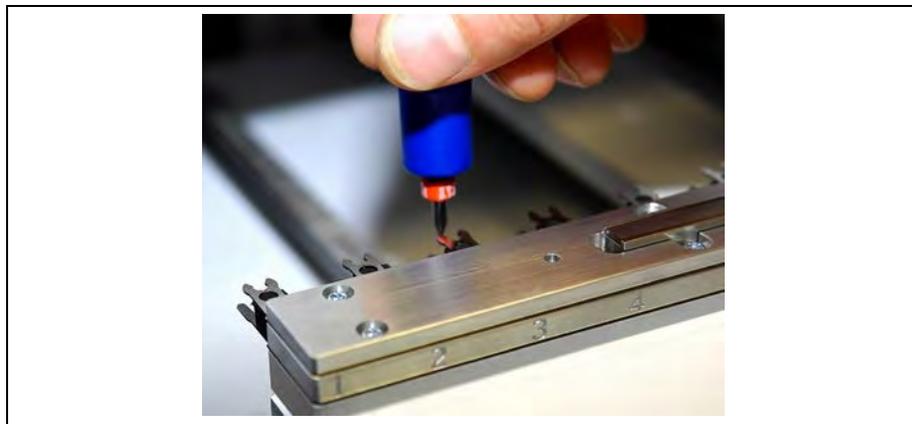
➔ The list of available tools is shown:

Fig. 292: Tool list



3. Select one of the tools that are listed on the left in section "Required tools".
4. Insert the chosen tool (in the dialog) physically into the corresponding tool holder:

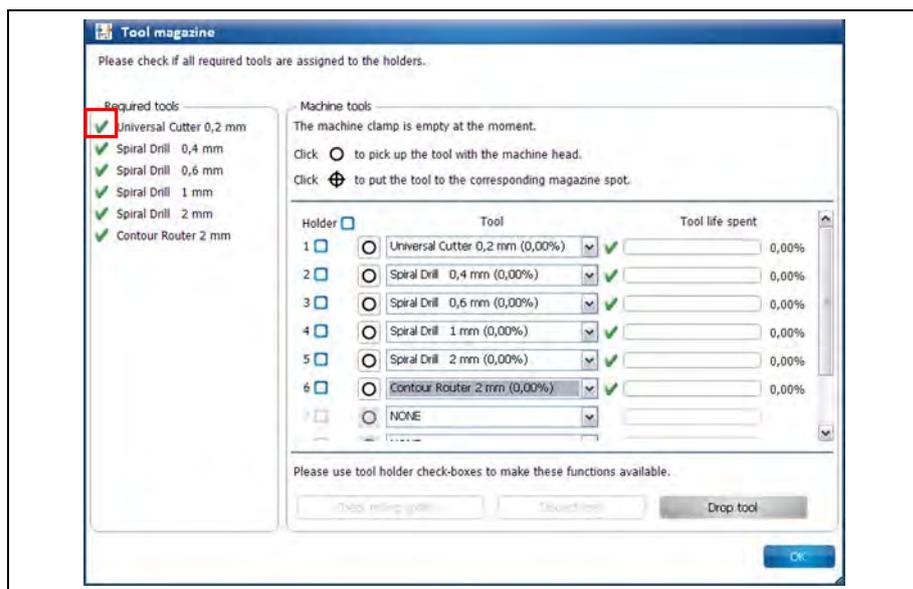
Fig. 293: Insert tool into holder



5. Repeat this procedure until you have placed all required tools into the tool holder of the machine.

➔ The tools you have placed are highlighted with a checkmark.

Fig. 294: Tool magazine





Note

As you assign each tool to a tool holder, the corresponding tool is highlighted with a checkmark. You will also notice that if you have a tool loaded into the tool magazine and it is not required for this design, there is no highlight like a checkmark or a cross.

6. Click on [OK].

➔ You will notice that the colors of the tool holders on the main layout screen match the colored rings on the physical tools, as shown below:

Fig. 295: Colors of the tool holders



◆ The tool magazine is loaded and the tools are assigned to holder positions.

2.3 Starting processing

This chapter describes the processing of the circuit board.

■ Starting processing

1. Click on the icon "Process Planning wizard" on the toolbar:

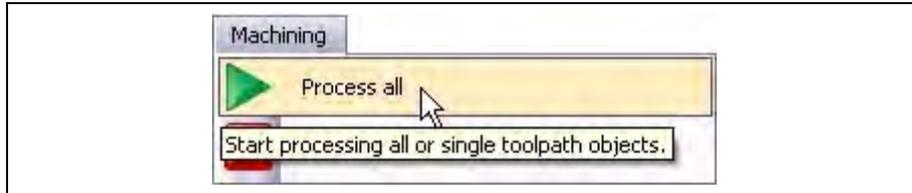
Fig. 296: Icon
"Process
Planning wizard"



Or

1. Click on Machining > Process all.

Fig. 297:
Machining >
Process all



Note

Make sure that <Process All> is selected in the combo box, so that all phases are executed.



Instead of processing all phases automatically, you can process the phases individually. In the combo box select the desired phase and click on the "Start processing" button.

You can also start processing beginning with a specific phase. Select the desired phase in the combo box and click on the "ladder" button. The selected phase and all following phases are processed in the correct order.

After the production started, the machine will process following phases in order. The phases are displayed via prompts.



Note

Depending on which ProtoMat you use the following phases could differ from the phases and messages displayed on your screen. Please follow the instructions on your screen.

For machines with manual tool exchange you are regularly asked to change the tool in the collet, for example.

Phase “MountMaterial”

1. Place the base material onto the machine’s table top.
2. Fasten the base material onto the table top using the adhesive tape.

Phase “MaterialSettings”

➔ Following dialog is displayed:

Fig. 298: Material settings

■ Entering the material settings

1. Enter the correct values for the material used.



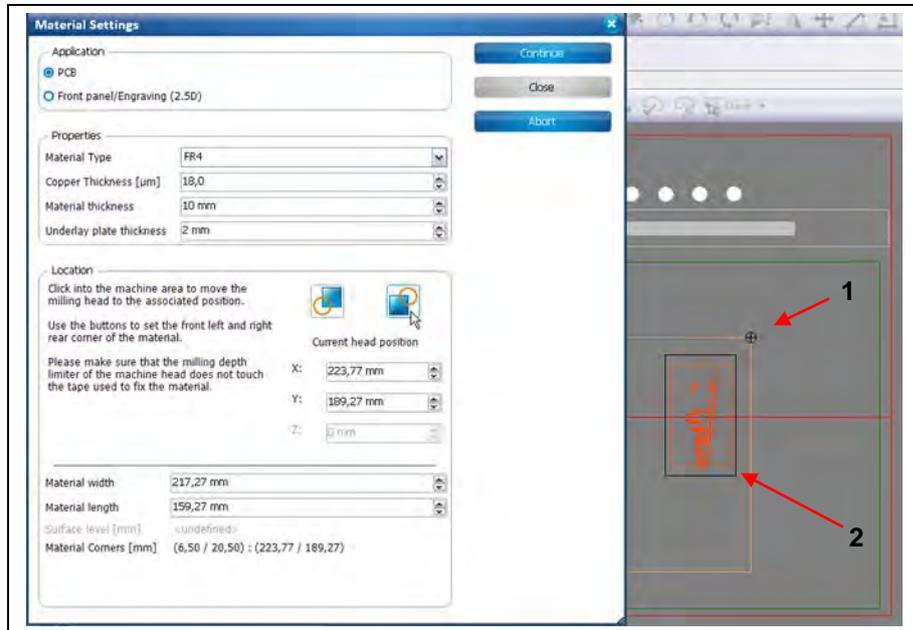
Note

PCB is selected as default.

2. Adapt the values of copper thickness and material thickness if necessary.

3. Define the processing area:
 - a) Move the dialog „Material Settings“ off to the side.
 - b) Using your mouse in the machining view, click on the right corner of your material:

Fig. 299: Right corner

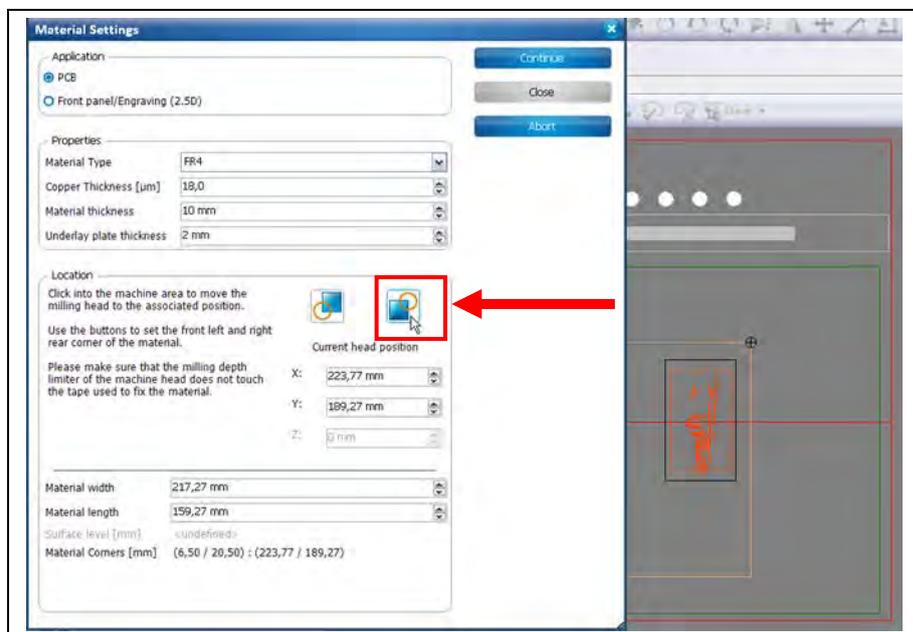


/1/ Click in the machining view

/2/ Material

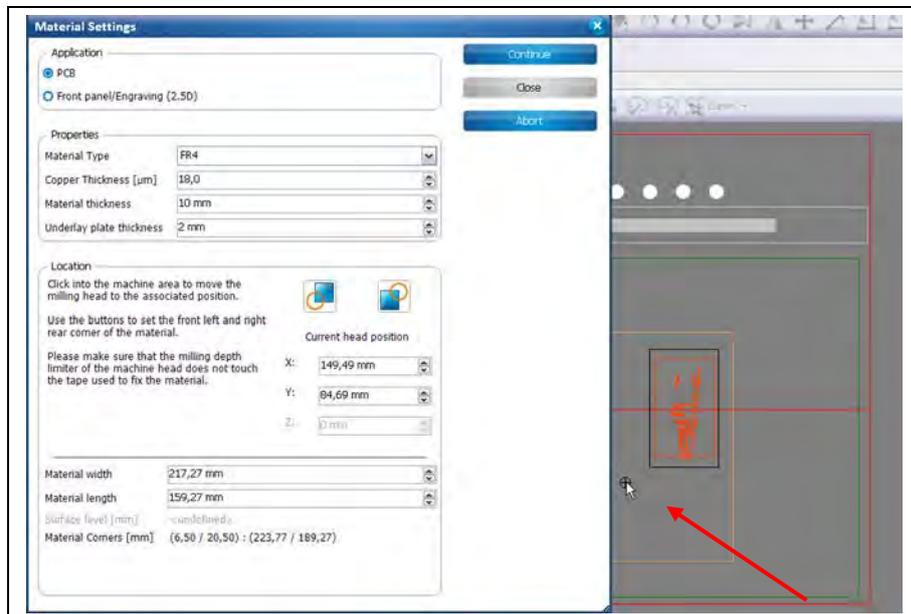
- ➔ The machine head moves to this position.
- c) Now click on the corresponding button in the dialog “Material Settings“:

Fig. 300: Click on button



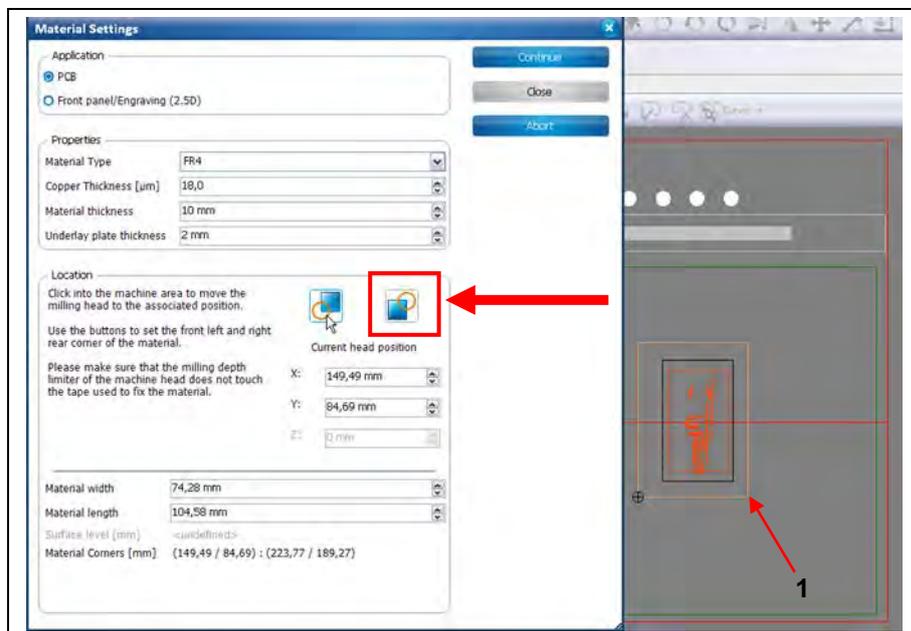
- ➔ The processing area is adapted.
- d) Using your mouse in the machining view, click on the left corner of your material:

Fig. 301: Left corner



- ➔ The machine head moves to this position.
- e) Now click on the corresponding button in the dialog "Material Settings":

Fig. 302: Defined processing area



/1/ Defined processing area

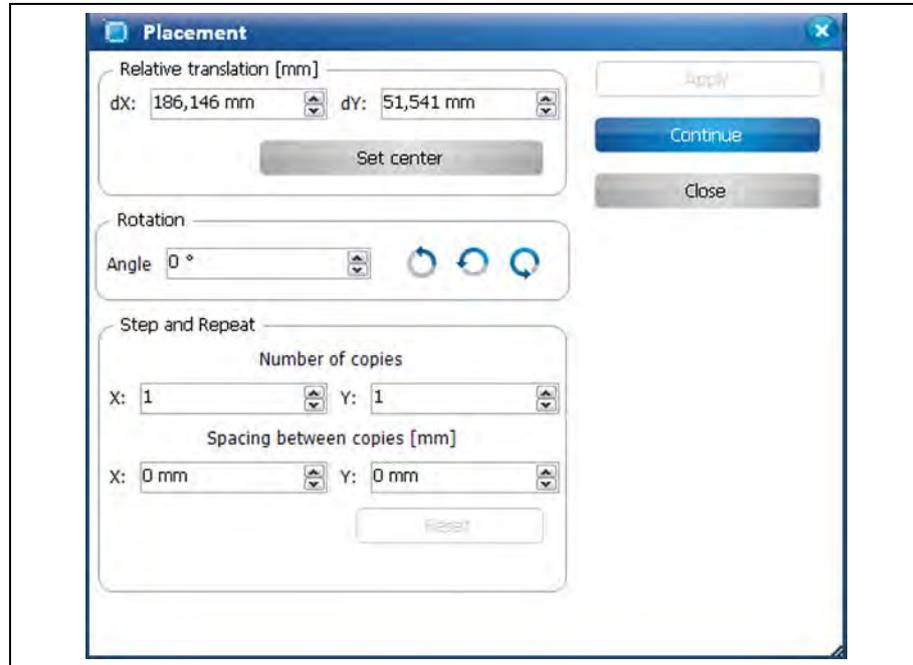
- ➔ The processing area was adapted to the material.
- 4. Click on [Continue].
- ◆ The material settings were entered.

Phase “Placement”

In this phase, the job can be placed arbitrarily on the base material and be multiplied if necessary.

➔ Following dialog is displayed:

Fig. 303:
Placement



1. Click on the job and drag it to the desired position using the mouse.
- Or
3. Enter the new position in the dialog.
2. If desired, multiply the job data by entering the number of copies and spacing values in X and Y direction in the corresponding fields (Step and Repeat section).



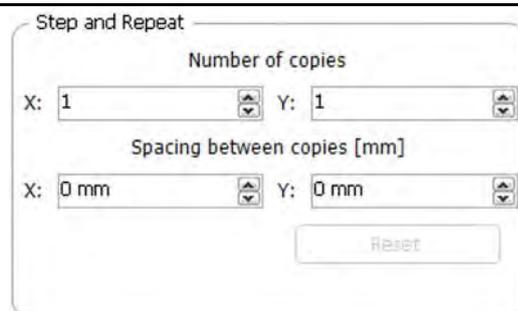
Tip

If you wish to create multiple copies of your layout, enter in section “Step and Repeat” the number of copies for each axis.

Then click on [OK].

The spacing between the copies is measured from the respective outer edge of the adjacent layout.

LPKF recommends increasing this distance slightly to produce a frame between the copies, as this makes it easier to cut out.



Phase “DrillFiducial”

- ➔ The machine picks up the tool “Spiral Drill 1.5 mm” and drills the fiducials.



If the spindle motor has not been used before, the motor is warmed up for 2 minutes.

Note

Phase “Marking Drills”

- ➔ The machine picks up the tool “Universal Cutter” and marks the positions for the drill holes.

Phase “Drilling Unplated”

- ➔ The machine picks up the required tool and drills the holes.



This phase may use more than one tool.

Note

Phase “Milling Bottom”

- ➔ The machine picks up the required tool and mills the isolation tracks.

Phase “Flip Material”

1. Flip the material.



If you are using a ProtoMat S43, S63 or S103 flip the material along the machine’s X-axis.

Note

If you are using a ProtoMat E33 flip the material along the machine’s Y-axis.

2. Confirm by clicking [OK].



The display in the machining view changes. The position of the design is adjusted to the circuit board. The side of the circuit board to be processed is now the “Top” side.

Note

Phase "Read Fiducials_Top"

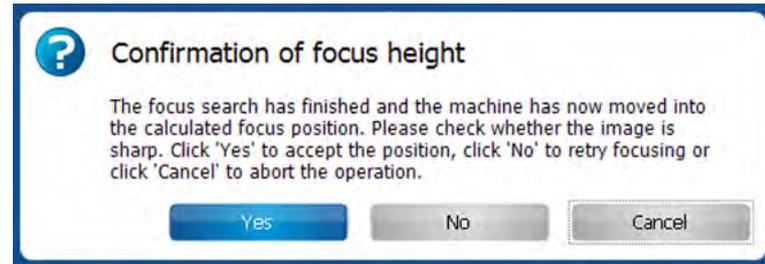
S43, S63 and S103



Note

If the fiducial search is performed for the first time (after having started CircuitPro) the camera is performing an autofocus five times.

Afterwards the following message is displayed which prompts you to confirm the focus height:

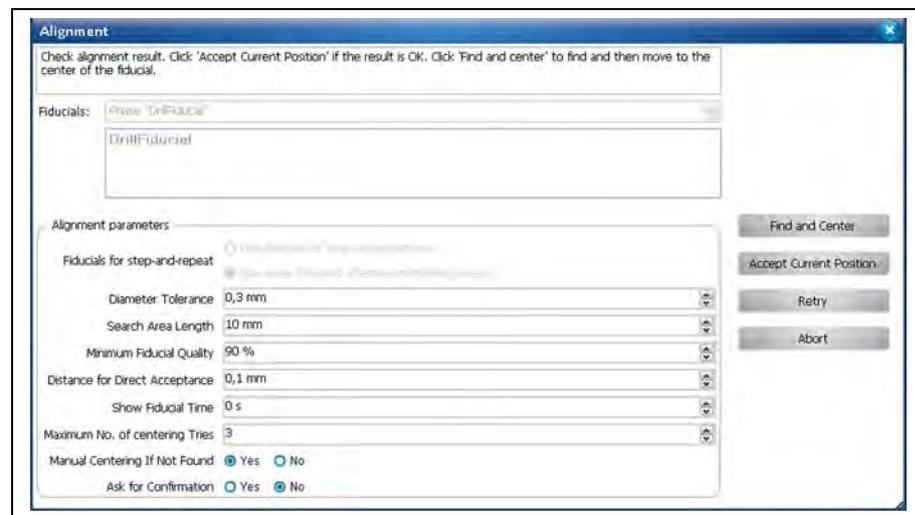


- ➔ The camera moves to the positions of the fiducials and determines the exact position.

The Top side is thus aligned to the Bottom side.

The following dialog is displayed if the fiducials have not been recognized automatically:

Fig. 304:
Alignment



1. Enlarge the search area by increasing the value of the field \Search Area Length\.
2. Start the search again.
3. Repeat above steps if necessary.



Note

Enlarging the search spiral increases the time required for searching the fiducials.

E33, S43 without camera

If no camera is available for processing the “Read Fiducials_Top” phase, the Top side is aligned to the Bottom side using the reference pins. The “Read Fiducials_Top” phase is not processed in this case.

Phase “Milling Top”

- ➔ The machine picks up the required tools and mills the isolation tracks on the Top side.

Phase “Contour Routing”

- ➔ The machine picks up the required tools and drills and mills the outline of the circuit board.

Phase “Board Production Finished”

- ➔ A message informs you that the production is finished.
- ◆ The production of the circuit board is finished.



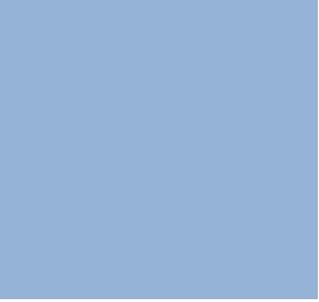
Note

If desired, continue with dispensing solder paste on your PCB. Therefore please refer to the tutorial “Dispensing solder paste using the ProtoMat S63 or S103”.



Tip

Subsequent to the board production you are able to label your circuit board or apply solder resist. LPKF offers you various systems such as the ProMask or ProLegend for these processes. Furthermore LPKF systems for applying solder paste, for mounting components and for soldering circuit boards are available. Do not hesitate to ask us.



Processing DXF files in CircuitPro



3 Processing DXF files in CircuitPro

This tutorial shows you how to process DXF files in CircuitPro.

The following steps are necessary to complete the tutorial successfully:

- i. Importing the DXF file
- ii. Converting the DXF file



Tip

You are able to perform these steps by using the virtual machine. This enables you to work without a real machine. Click on Machining > Connect > Virtual.

3.1 Importing the DXF file



Note

CircuitPro must be running.



Tip

The LPKF tutor data are stored in "My Document\LPKF Laser & Electronics\ LPKF CircuitPro 1.5\Example Data\ UseCase_DXFFiles".

■ Importing the file

1. Click the icon "Import" on the toolbar:

Fig. 305: Icon import

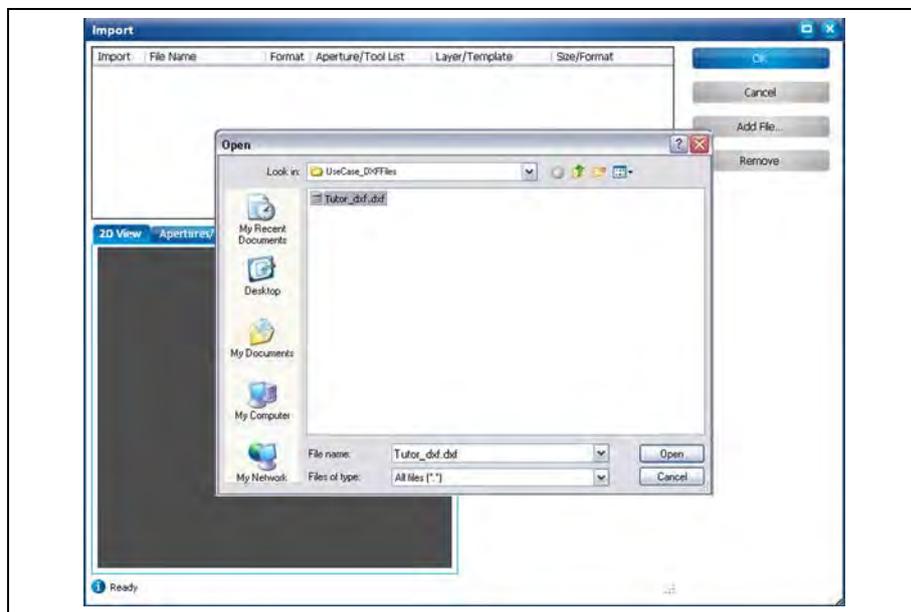


Or

1. Click on File > Import...

➔ The following dialog is displayed:

Fig. 306: Import dialog



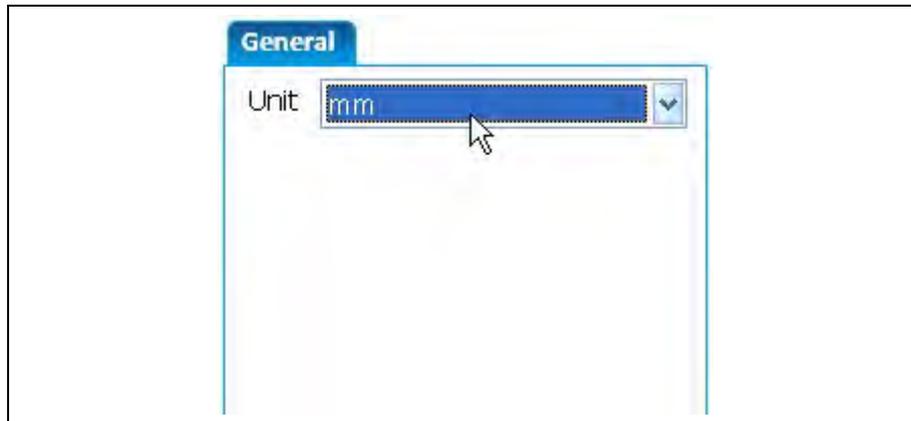
2. Select the file "Tutor_dxf.dxf".
 3. Click on [Open].
- ➔ The document's information is shown in the table.
4. Change the layer/template column to "TopLayer":

Fig. 307: Change column



5. Select in the tab "General" the increment that the DXF file was originally drawn in:

Fig. 308: Selecting the increment



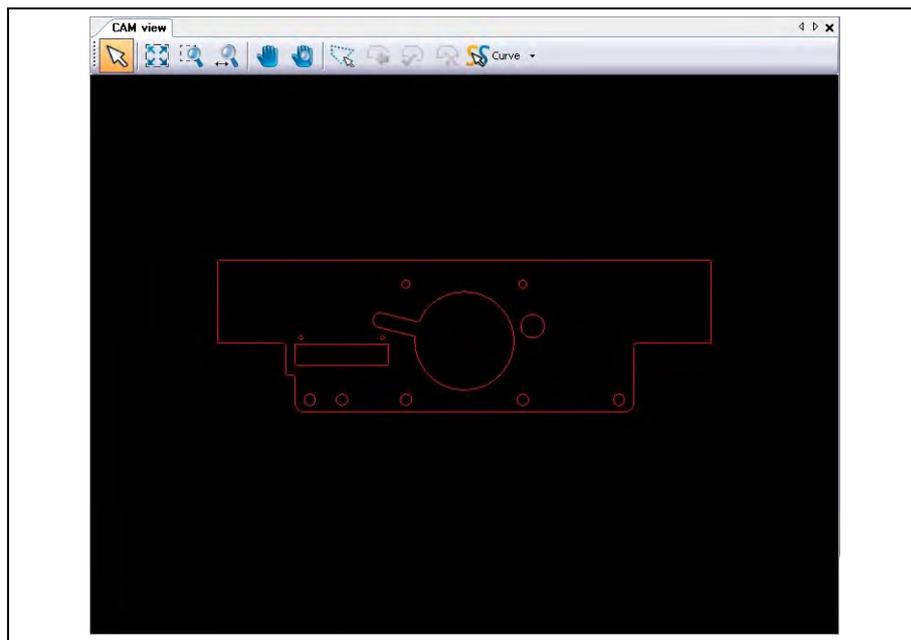
Tip

The information given in the column "Size/Format" enables you to check, if the actual size corresponds to the real size of your design.

If the size does not correspond to the real size, change the unit in the tab "General".

6. Click on [OK].
- ➔ The data is shown in the CAM view:

Fig. 309: CAM view



- ◆ The file was imported.



Note

Please note that certain data of the DXF files could be on non-visible layers.

When importing these files the corresponding layer is also non-visible in CircuitPro.

3.2 Converting the DXF file

Converting the DXF file includes the following steps:

- Defining the Board Outline
- Assigning the objects to the corresponding layers
- Converting the objects to polygons
- Converting the drilling holes

■ Defining the Board Outline



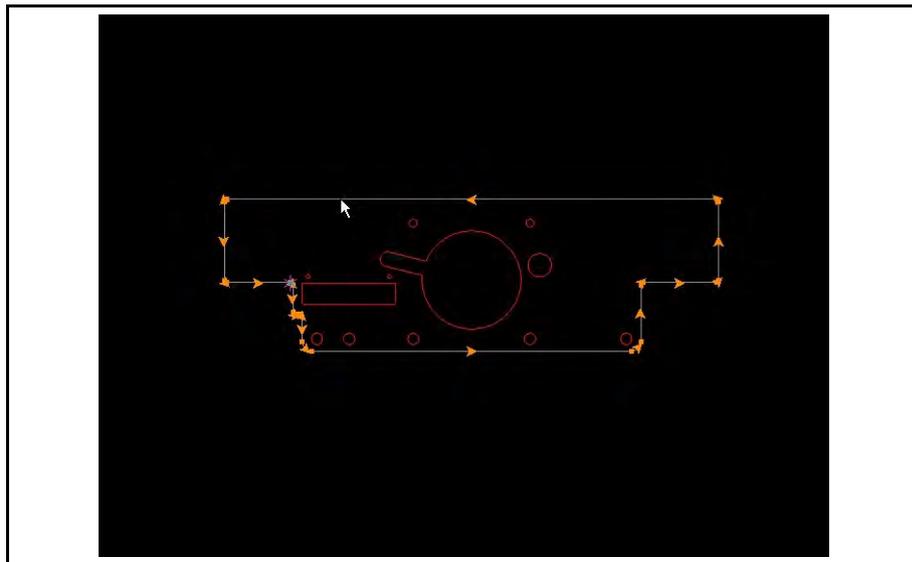
Tip

You are able to highlight every single element of the board outline by clicking and holding down the Shift-key.

After having highlighted all elements you can combine them to a closed path by using the functions "Convert to closed path".

1. Click on the board outline:

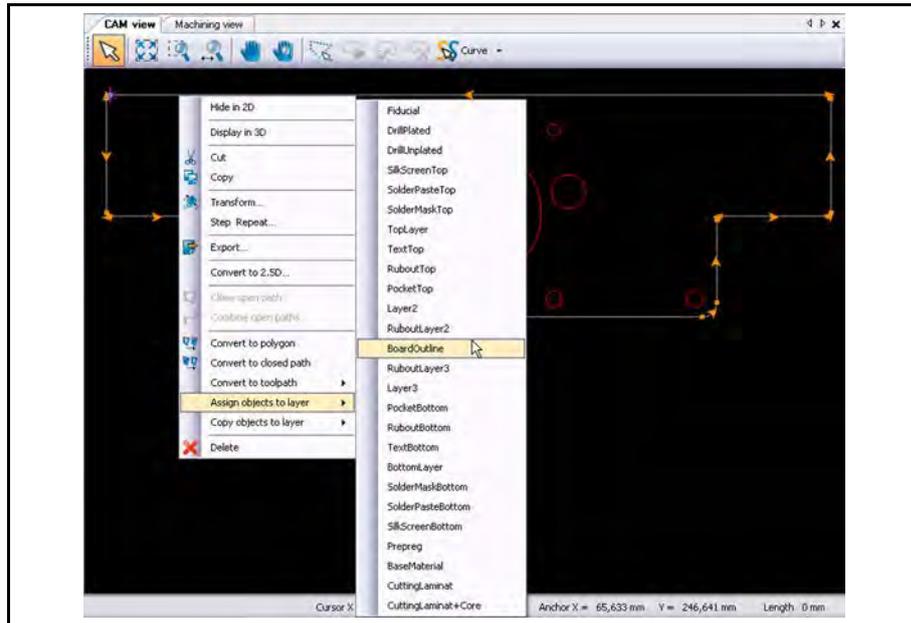
Fig. 310: Board outline selected



2. Press the right mouse button.
3. Select "Assign objects to layer".

4. Click on "BoardOutline":

Fig. 311: Assign objects to layer



- ➔ The color of the board outline changes.
- ◆ The board outline is defined.

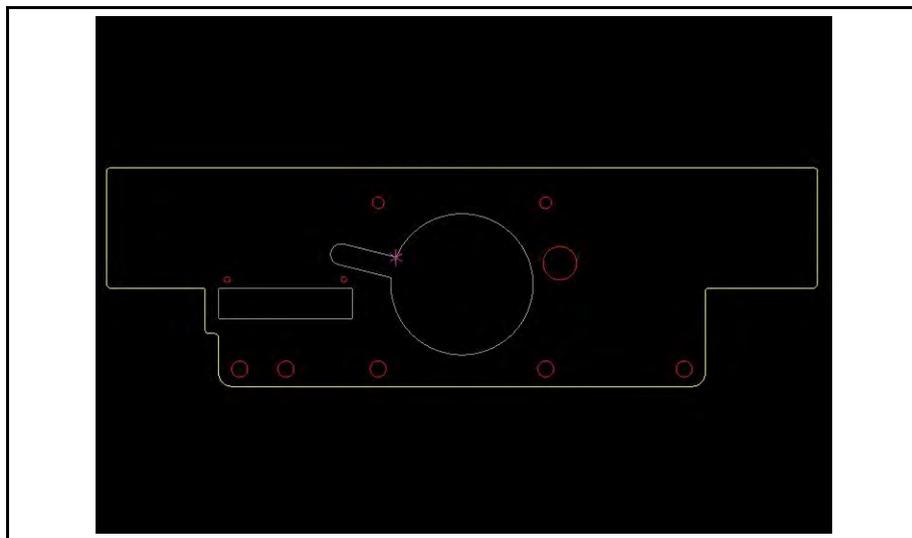
- Assigning the objects to the corresponding layers

1. Highlight all traces that will go onto the top layer.



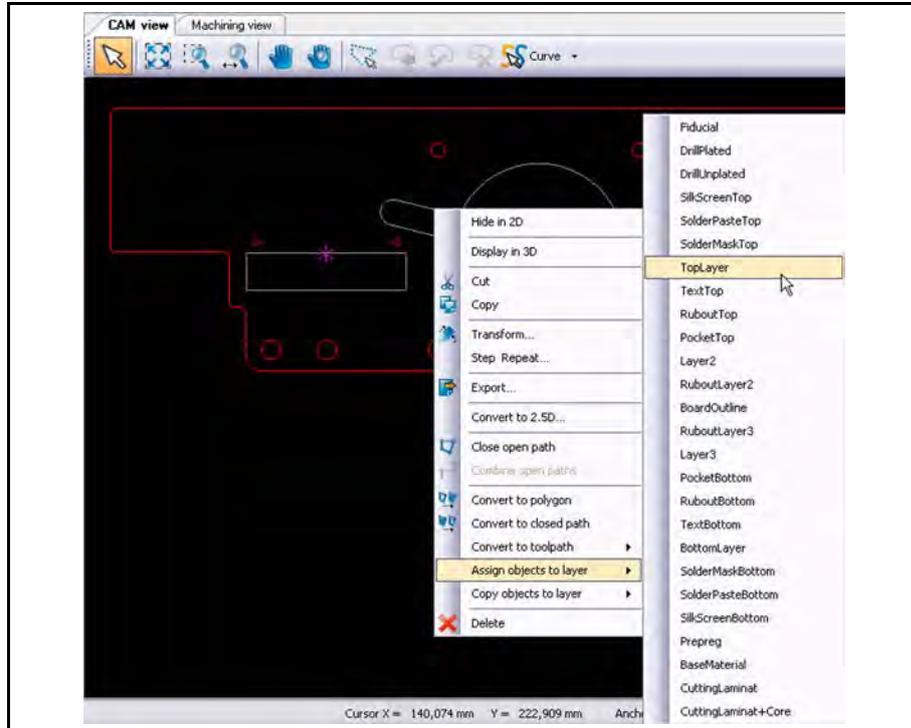
Tip

You can also single click on the traces. Hold the control key down to select multiple items.

Fig. 312:
Highlighting the traces

2. Press the right mouse button.
3. Select "Assign objects to layer".
4. Click on "TopLayer":

Fig. 313: Assign objects to layer

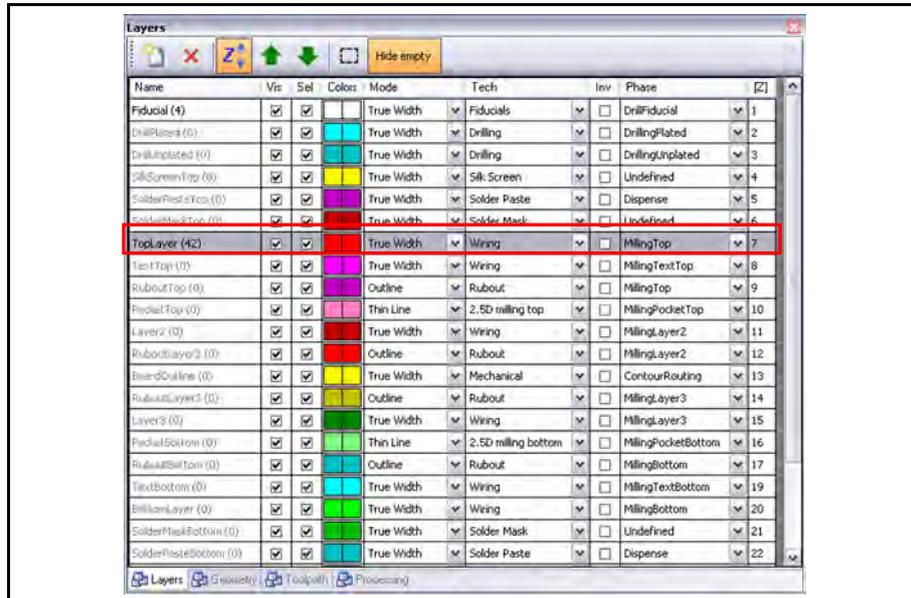


- ➔ The traces were assigned to the top layer.
5. Assign the drill holes to the layer "DrillUnplated":
 - a) Highlight all drill holes.
 - b) Follow steps 2 and 3.
 - c) Click on "DrillUnplated"
- ➔ The drill holes were assigned to the layer "DrillUnplated".
 - ◆ The objects were assigned to the corresponding layers.

- Converting the objects to polygons

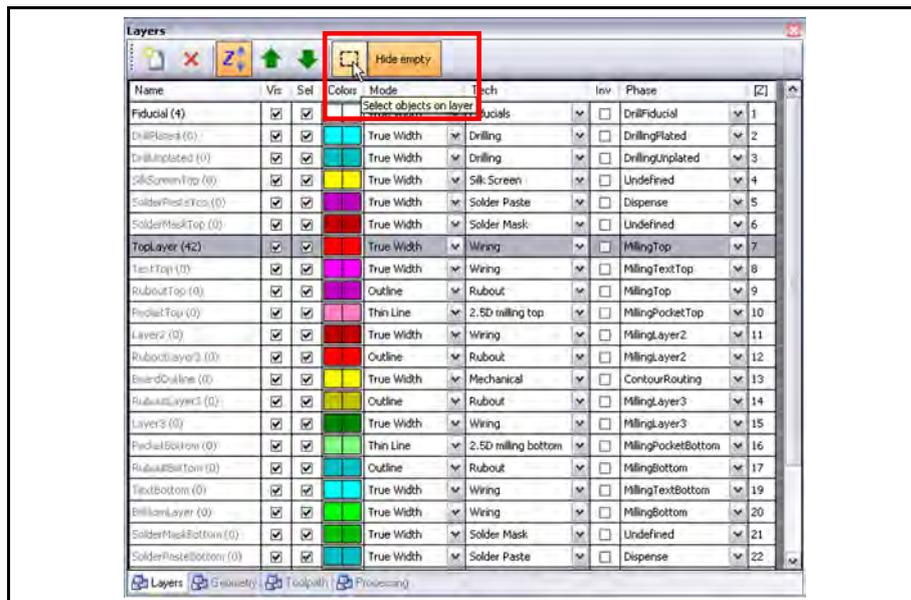
1. Single left click on the "TopLayer" in the pane "Layers":

Fig. 314: Pane
"Layers"



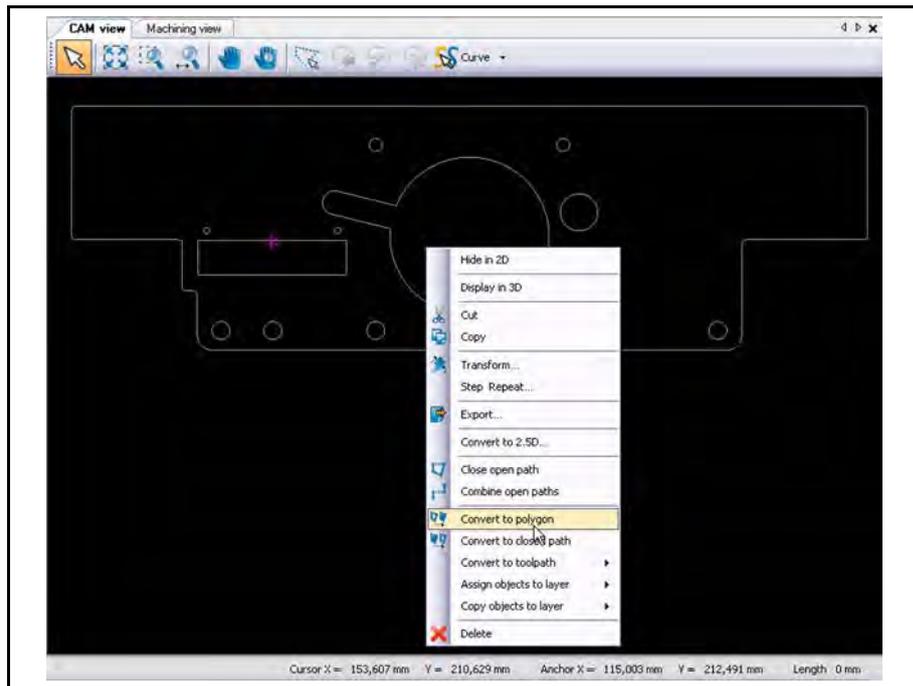
2. Click on the icon "Select objects on layer" in the pane "Layers":

Fig. 315: Select
objects on layer



3. Press the right mouse button.
4. In the context menu, select "Convert to polygon":

Fig. 316: Convert
to polygon



This will combine all segments and fill the object.

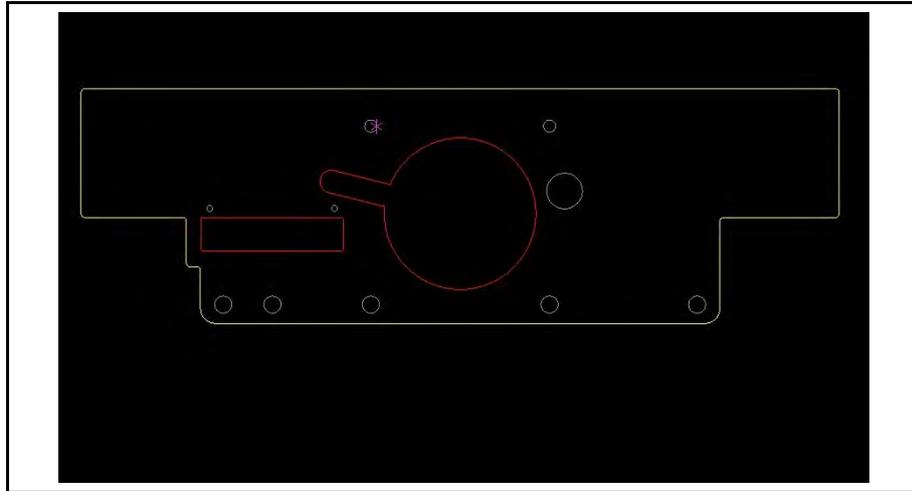
Note

- ◆ The objects were converted to polygons.

■ Converting the drill holes

1. Highlight all drill holes:

Fig. 317:
Highlighting the
drill holes



Tip

You can also single click on the drill holes. Hold the control key down to select multiple items.

2. Click on Modify > Draw to flash:

◆ The drill holes were converted.



Tip

The .dxf file was successfully imported and converted. Now you can insert rubout areas or/and fiducials before you start processing your board. Please refer to the tutorial “Basic CAM operations” in this case.

If you want to start processing your board without inserting rubout areas/fiducials, please refer to the tutorial “Basic machining operations”.



Processing Gerber and Excellon files



4 Processing Gerber and Excellon files

This tutorial shows you how to process Gerber and Excellon files in CircuitPro. The following steps are necessary to complete the tutorial successfully:

- i. Selecting Gerber and Excellon files
- ii. Selecting the file format
- iii. Selecting the desired target layer
- iv. Setting/correcting size and format
- v. Setting apertures and tools
- vi. Using the layer name defined in the Gerber file



Tip

You can perform these steps by using the virtual machine. Thus, you can work without a real machine. Click on Machining > Connect > Virtual and [Connect].

4.1 Selecting Gerber and Excellon files



CircuitPro must be running.

Note

■ Selecting files



The LPKF tutor data are stored in “My Documents\LPKF Laser & Electronics\ LPKF CircuitPro 1.5\Example Data”.

Tip

1. Click the icon “Import” on the toolbar:

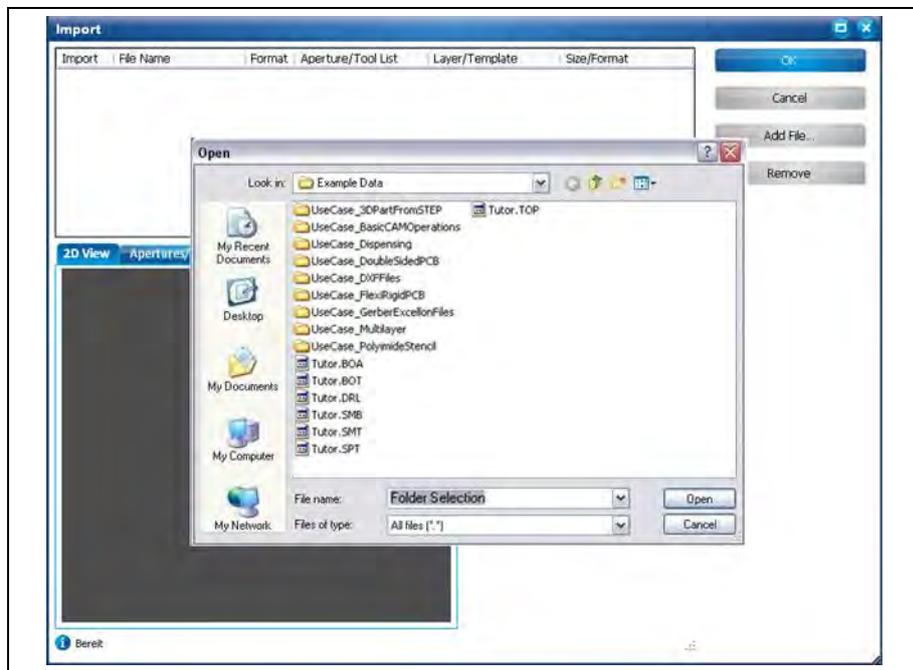
Fig. 318: Icon import



Or

1. Click on File > Import...
- ➔ The following dialog is displayed:

Fig. 319: Import dialog



2. Select the files to be imported.



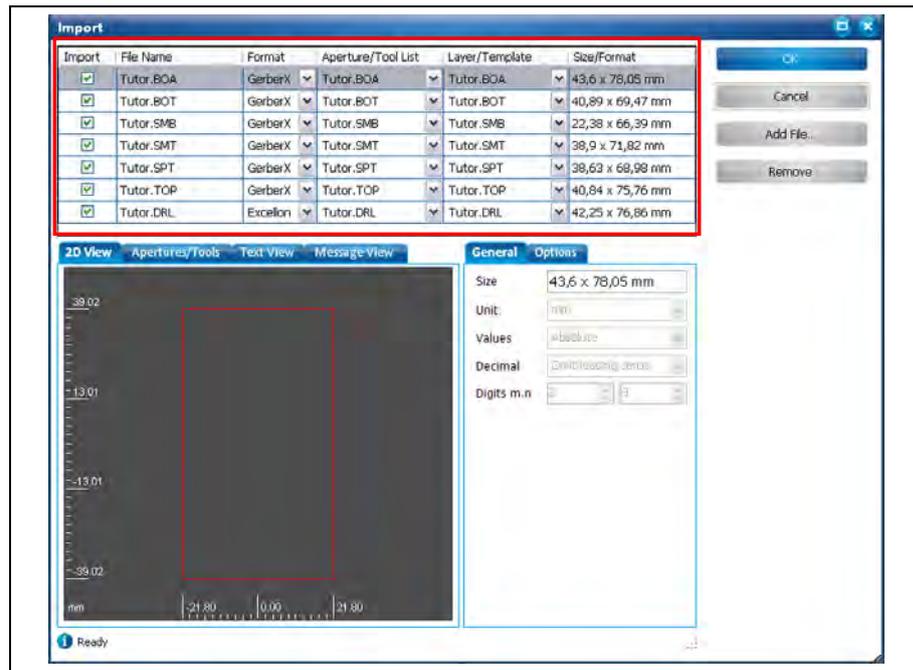
Note

Instead of selecting individual Gerber or Excellon files, you can select a whole folder. In this case, all files in the selected folder are displayed in the table (see figure below).

3. Click on [Open].

➔ The files to be imported are displayed in the table:

Fig. 320:
Imported files



The columns of the table highlighted contain the following information/settings:

Column	Description
Import	Checkmark the files that you want to import.
File name	The name of the selected file is displayed.
Format	The format of the selected file is displayed. If CircuitPro has not recognised the file format correctly, you can assign the correct format in the corresponding drop-down list (for more information please refer to chapter "Selecting the file format").
Aperture/Tool List	The Aperture list is usually part of the Gerber or Excellon file. The apertures contained are displayed in the "Apertures/Tools" tab.
Layer/Template	Assign the layer that is to contain the imported data (for more information please refer to the chapter "Selecting the desired target layer").

Column	Description
Size/Format	The size of the imported design is displayed in this column (for more information please refer to the chapter "Setting/correcting size and format").

- ◆ The files are selected.

4.2 Selecting the file format

If CircuitPro does not recognise the file format, the corresponding row in the dialog is grayed-out:

Fig. 321:
Unrecognised
files

Import	File Name	Format	Aperture/Tool List	Layer/Template	Size/Format
<input checked="" type="checkbox"/>	Tutor.BOA	GerberX	Tutor.BOA	Tutor.BOA	43,6 x 78,05 mm
<input checked="" type="checkbox"/>	Tutor.BOT	GerberX	Tutor.BOT	Tutor.BOT	40,89 x 69,47 mm
<input checked="" type="checkbox"/>	Tutor.SMB	GerberX	Tutor.SMB	Tutor.SMB	22,38 x 66,39 mm
<input checked="" type="checkbox"/>	Tutor.SMT	GerberX	Tutor.SMT	Tutor.SMT	38,9 x 71,82 mm
<input checked="" type="checkbox"/>	Tutor.SPT	GerberX	Tutor.SPT	Tutor.SPT	38,63 x 68,98 mm
<input checked="" type="checkbox"/>	Tutor.DRL	Excellon	Tutor.DRL	Tutor.DRL	42,25 x 76,86 mm
<input type="checkbox"/>	Tutor.TOP	Undefined			

In such a case, the information on aperture list, layer, and format are also missing.

You can select the file format manually.

■ Selecting the file format

1. In the “Format” column, click on the arrow button of the file concerned.

➔ The format selection list is displayed:

Fig. 322:
Selecting the file
format manually

Import	File Name	Format	Aperture/Tool List	Layer/Template	Size/Format
<input checked="" type="checkbox"/>	Tutor.BOA	GerberX	Tutor.BOA	Tutor.BOA	43,6 x 78,05 mm
<input checked="" type="checkbox"/>	Tutor.BOT	GerberX	Tutor.BOT	Tutor.BOT	40,89 x 69,47 mm
<input checked="" type="checkbox"/>	Tutor.SMB	GerberX	Tutor.SMB	Tutor.SMB	22,38 x 66,39 mm
<input checked="" type="checkbox"/>	Tutor.SMT	GerberX	Tutor.SMT	Tutor.SMT	38,9 x 71,82 mm
<input checked="" type="checkbox"/>	Tutor.SPT	GerberX	Tutor.SPT	Tutor.SPT	38,63 x 68,98 mm
<input checked="" type="checkbox"/>	Tutor.DRL	Excellon	Tutor.DRL	Tutor.DRL	42,25 x 76,86 mm
<input type="checkbox"/>	Tutor.TOP	Undefined			

2. Click on the corresponding file format (“GerberX” in this case).

➔ The information in the columns “Aperture/Tool List”, “Layer/Template”, and “Size/Format” is displayed automatically if you have selected the correct file format.

Fig. 323: File
details

<input checked="" type="checkbox"/>	Tutor.TOP	GerberX	Tutor.TOP	TopLayer	40,84 x 75,76 mm
-------------------------------------	-----------	---------	-----------	----------	------------------

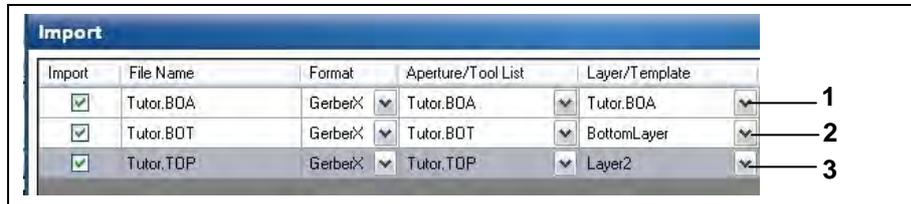
➔ The checkmark in the “Import” column is set automatically by CircuitPro.

◆ The file format is selected.

4.3 Selecting the desired target layer

Once you have selected the files to be imported, these files have to be assigned to layers in CircuitPro. You can assign the files in the “Layer/Template” column:

Fig. 324:
“Layer/Template”
column



There are three ways to assign the layer:

1. Keeping the layer that is created during file selection:

In this case, CircuitPro creates a new layer during the file import if it does not exist in the template (see /1/).

2. Selecting a layer that already exists in the template file of CircuitPro:

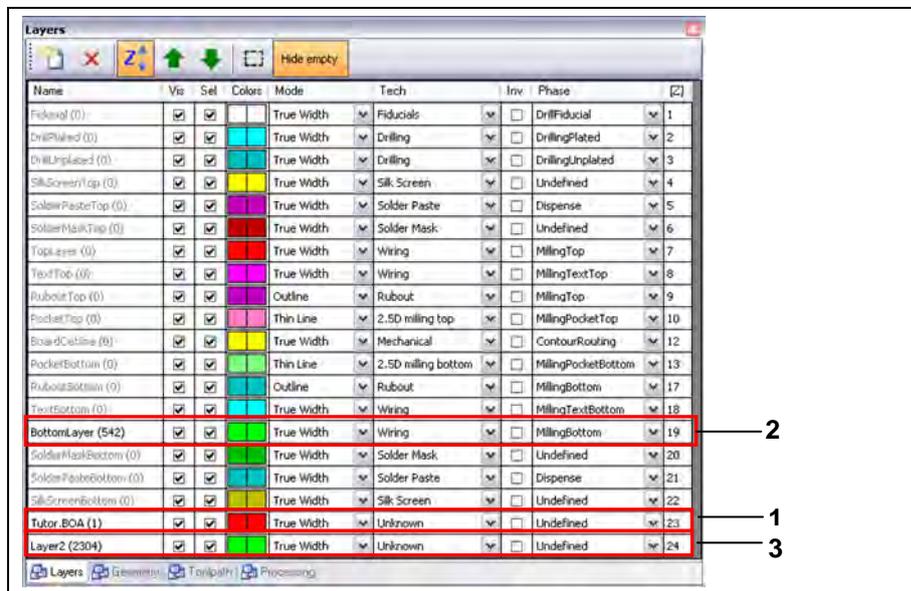
These layers are in the templates of CircuitPro and appear in the drop-down lists that are shown when you click on the list's arrow button (see /2/).

3. Creating a new layer:

If you want to create a new layer, you can enter a name for the layer in the “Layer/Template” column. CircuitPro creates a new layer for this file during the file import (see /3/).

After the file import, the layers are listed in the “Layers” pane of CircuitPro:

Fig. 325: “Layers”
pane



4.4 Setting/correcting size and format

In some cases, the design of the file is not displayed correctly. There are four possible causes for this:

- Wrong measurement unit: A wrong measurement unit was selected when importing the file (inch instead of mm).
- Wrong number of decimal digits: The number of decimal digits entered does not match the file's contents.
- Wrong declaration of the value (relative/absolute)
- Wrong null suppression (decimal)

■ Checking and modifying the measurement unit

The dimensions of the imported files are listed in the “Size/Format” column. Millimeters are used by default.



Note

If the imported files are GerberX files, the measurement unit is preset by default and cannot be modified.

You can modify the measurement unit for all other file types (Gerber, Excellon etc.) in the “General” tab of CircuitPro’s Import dialog.

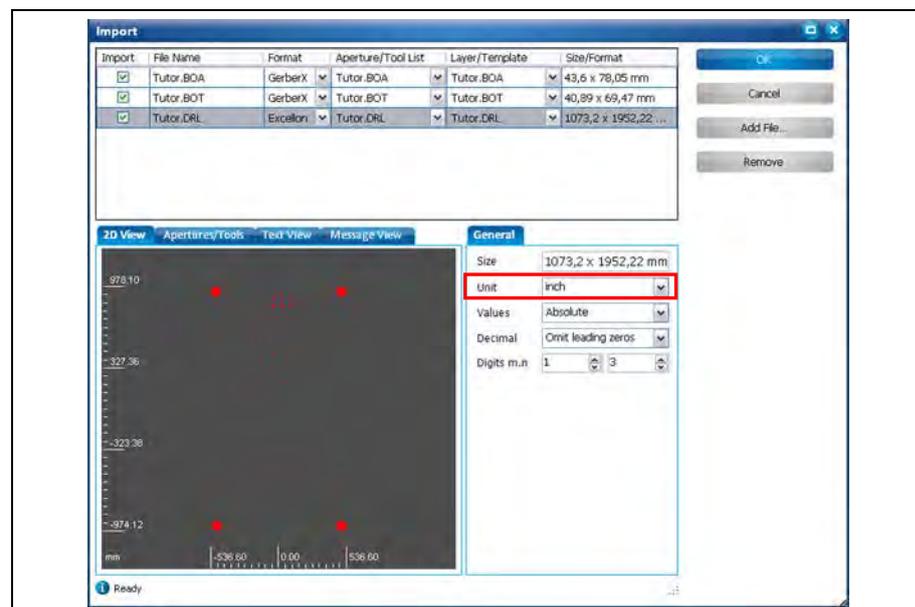
If the design size of some files looks peculiar, you should check the measurement unit:

Fig. 326:
Abnormal design
size

Import	File Name	Format	Aperture/Tool List	Layer/Template	Size/Format
<input checked="" type="checkbox"/>	Tutor.BOA	GerberX	Tutor.BOA	Tutor.BOA	43,6 x 78,05 mm
<input checked="" type="checkbox"/>	Tutor.BOT	GerberX	Tutor.BOT	Tutor.BOT	40,89 x 69,47 mm
<input checked="" type="checkbox"/>	Tutor.DRL	Excellon	Tutor.DRL	Tutor.DRL	1073,2 x 1952,22 mm

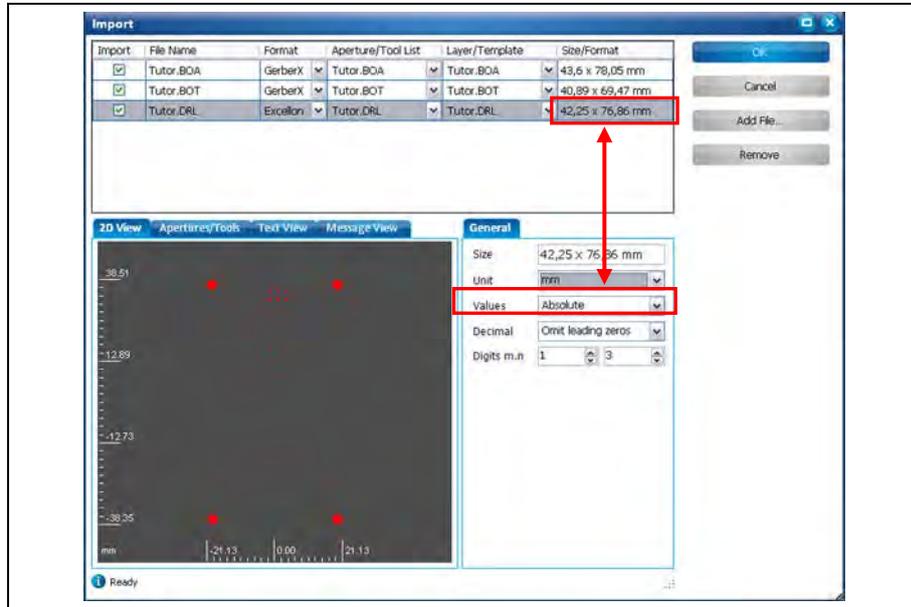
1. Click on the tab “General”.

Fig. 327: Wrong
measurement
unit



2. Check the measurement unit.
3. If the measurement unit is set to "inch", click on the arrow button.
 - ➔ The measurement unit selection list is displayed.
4. Select "mm" as measurement unit.
 - ➔ The design's dimensions change automatically.

Fig. 328:
Corrected
measurement
unit

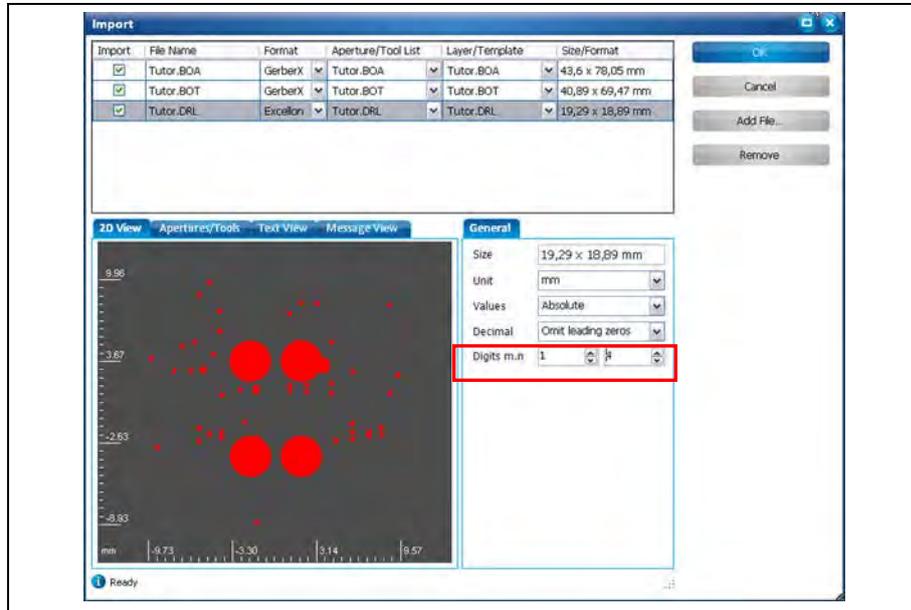


- ◆ The measurement unit has been checked and corrected.

■ Checking and correcting the number of decimal digits

If the design is still not displayed correctly in the “2D View” tab, you should check the number of decimal digits. An incorrect display of the design data may look as follows:

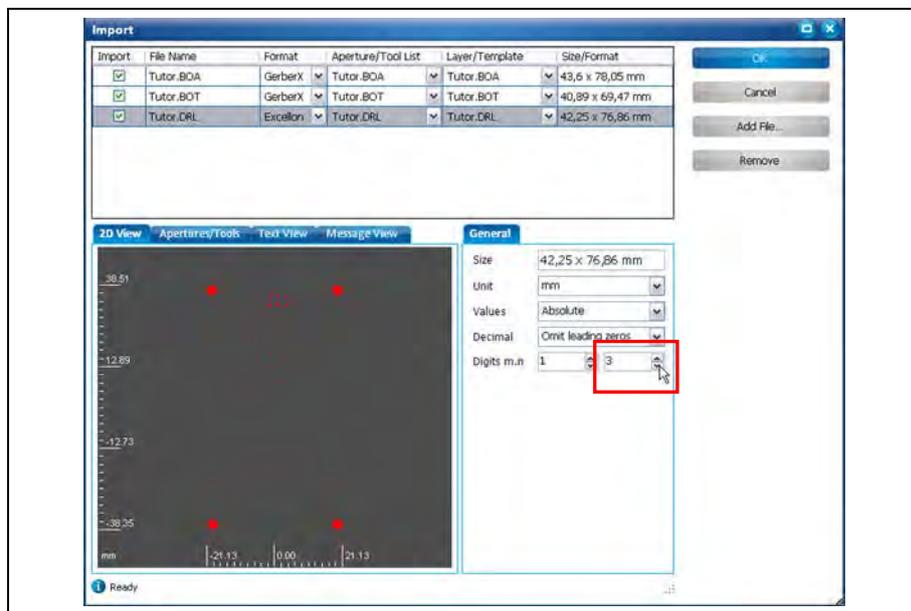
Fig. 329:
Incorrect preview



1. Click on the tab “General”.
2. Change the “n” digit count from “4” to “3”.

➔ The design's preview changes as follows:

Fig. 330: Correct
preview



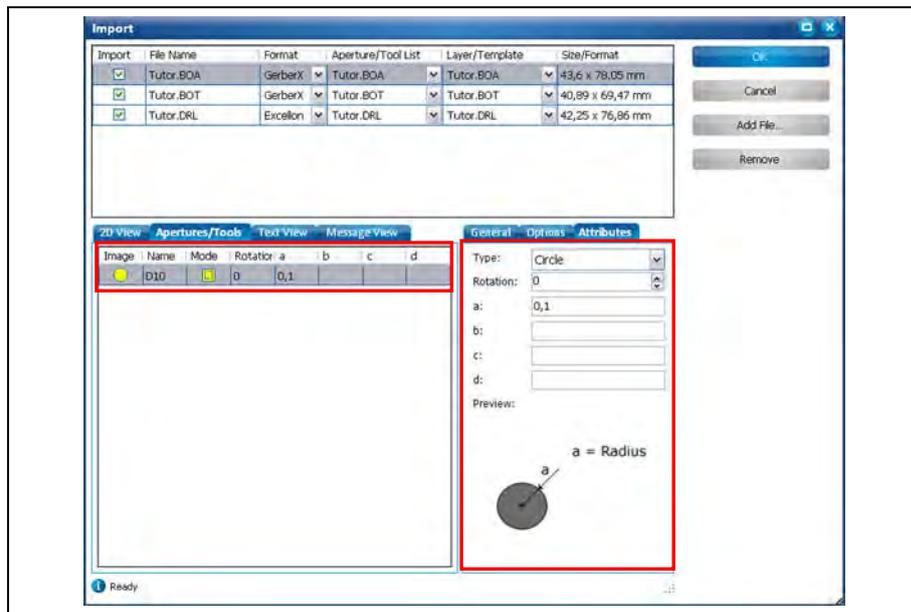
◆ The number of decimal digits has been checked and corrected.

4.5 Viewing/modifying aperture properties

You can view the properties of the apertures contained in the files. You can also modify the aperture properties according to your needs.

- Modifying aperture properties
 1. Select the file whose apertures you want to regard.
 - ➔ The aperture contained in the file is displayed in the “Apertures/Tools” tab.
 - ➔ The aperture properties are displayed in the “Attributes” tab.

Fig. 331: Aperture properties



The following aperture properties can be modified in the “Attributes” tab:

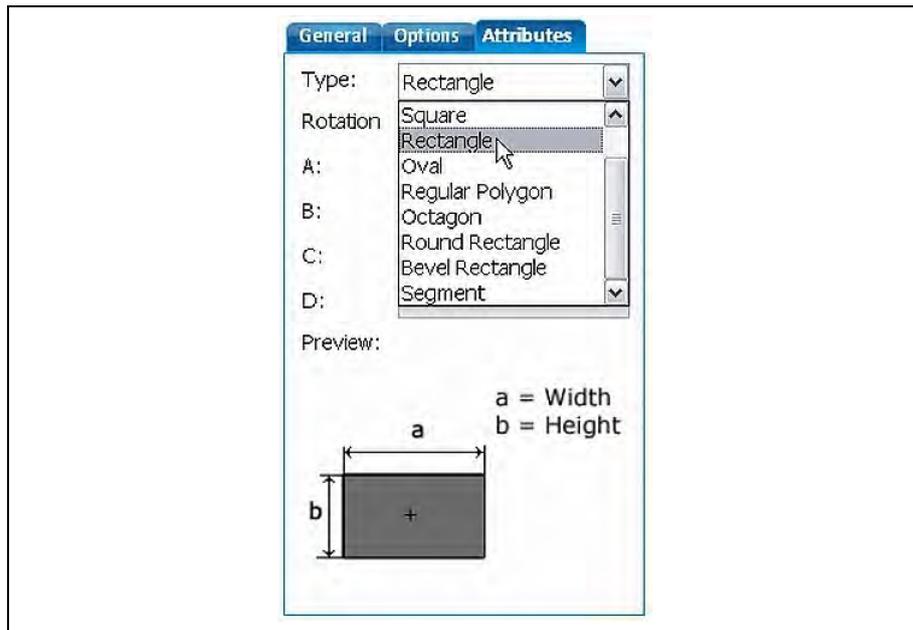
- Shape of the aperture (circle, square, oval, etc.)
- Rotation of the aperture
- Parameters that define the aperture's geometry

If you change the aperture's attributes, the “Attributes” tab displays a preview of the aperture.

In this example, the aperture's shape is to be changed from “circle” to “rectangle”.

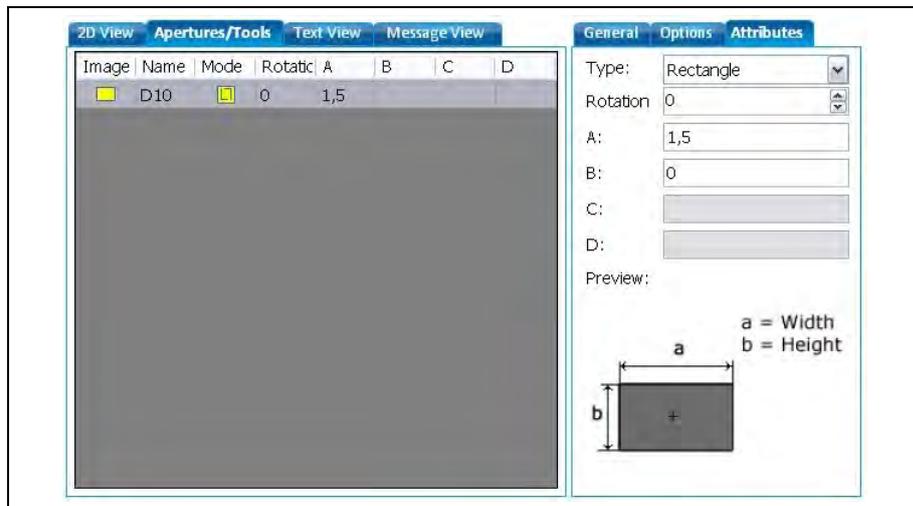
2. Click on the arrow button of the drop-down list “Type”.
 - ➔ The aperture shape selection list is displayed:

Fig. 332:
Available
aperture shapes



3. Click on "Rectangle" in the drop-down list.
 4. Enter "1.5" in the field \A:\
- ➔ The aperture's shape and size are changed according to your input:

Fig. 333: Modified
aperture shape
and size

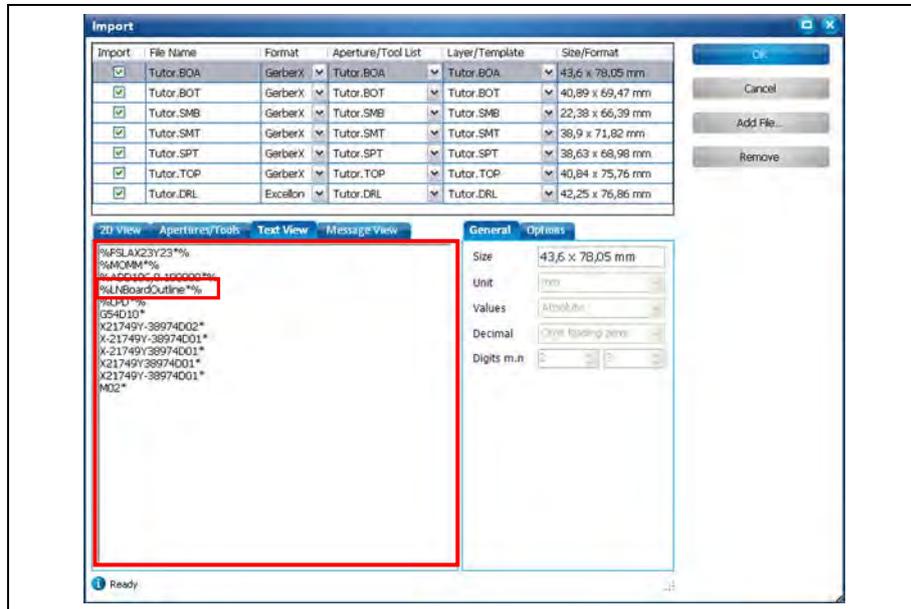


- ◆ The aperture's properties are modified.

4.6 Using the layer name defined in the Gerber file

The Gerber files contain the name of the layer. You can see the layer name in the “Text View” tab of the Import dialog:

Fig. 334: Layer name in Gerber file



The option “Use layer name” is only available for importing Gerber files as the layer name is defined in the Gerber file.

Note

■ Using the layer name for import

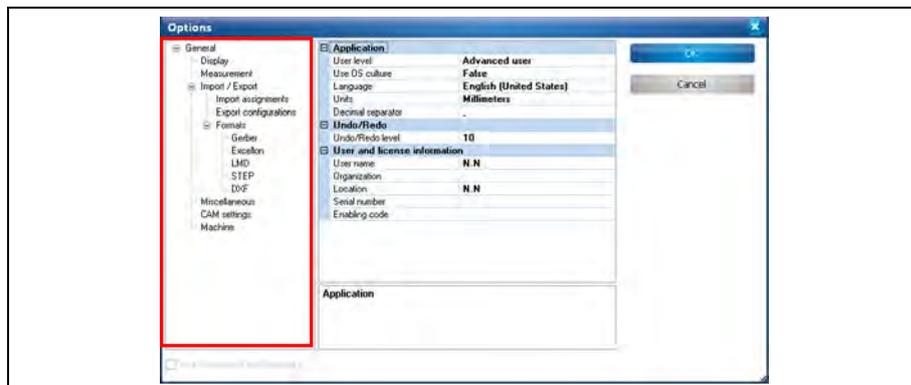
In CircuitPro, you can either

- define globally that the layer name is read from the Gerber files during import or
- you activate this option only once during import.

How to activate the option globally:

1. Click on Extras > Options...
- ➔ The following dialog is displayed:

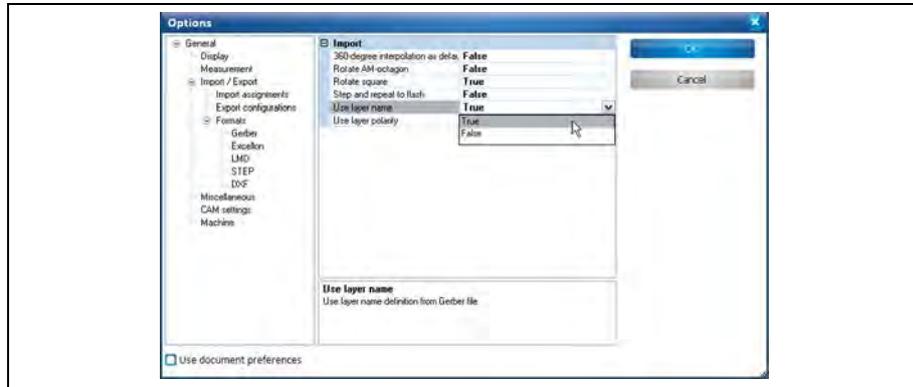
Fig. 335: Extras > Options



- Click in the category tree on the left on General > Import/Export > Formats > Gerber.

➔ The dialog changes as follows:

Fig. 336: Extras > Options



- Click on the arrow button of the drop-down list "Use layer name".
 - If the value of "Use layer name" is set to "True", the correct layer is assigned to the Gerber files automatically in CircuitPro:

Fig. 337: Import using option "Use layer name"

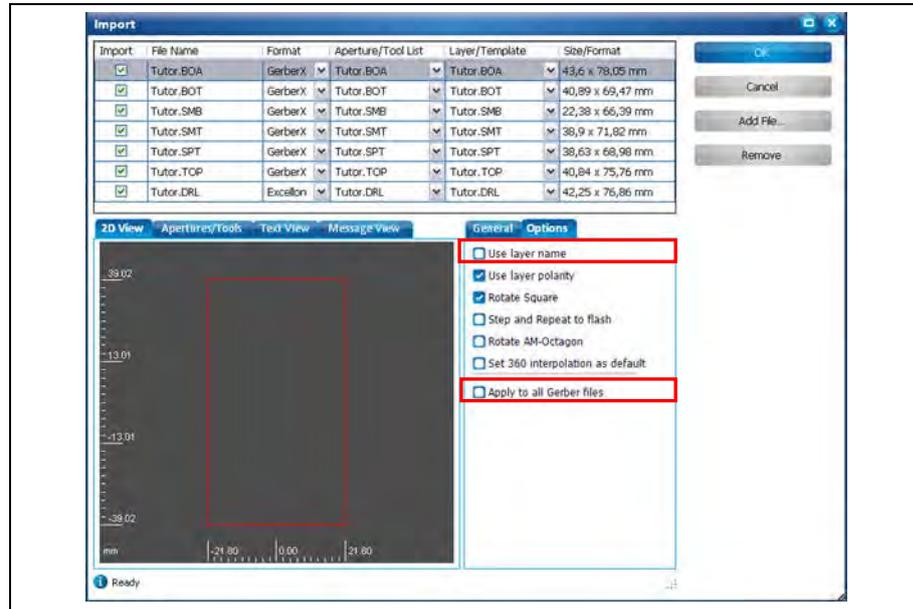
Import	File Name	Format	Aperture/Tool List	Layer/Template	Size/Format
<input checked="" type="checkbox"/>	Tutor.BOA	GerberX	Tutor.BOA	BoardOutline	43,6 x 78,05 mm
<input checked="" type="checkbox"/>	Tutor.BOT	GerberX	Tutor.BOT	BottomLayer	40,89 x 69,47 mm
<input checked="" type="checkbox"/>	Tutor.SMB	GerberX	Tutor.SMB	SolderMaskBottom	22,38 x 66,39 mm
<input checked="" type="checkbox"/>	Tutor.SMT	GerberX	Tutor.SMT	SolderMaskTop	38,9 x 71,82 mm
<input checked="" type="checkbox"/>	Tutor.SPT	GerberX	Tutor.SPT	SolderPasteTop	38,63 x 68,98 mm
<input checked="" type="checkbox"/>	Tutor.TOP	GerberX	Tutor.TOP	TopLayer	40,84 x 75,76 mm
<input checked="" type="checkbox"/>	Tutor.DRL	Excellon	Tutor.DRL	Tutor.DRL	42,25 x 76,86 mm

- If the value is left at "False", the option is not globally activated. In the latter case, you can define each time again when importing Gerber files whether to use the layer name.

Using the layer name automatically when importing:

1. Switch to the “Options” tab in the “Import” dialog.

Fig. 338:
“Options” tab



2. Checkmark the check boxes “Use layer name” and “Apply to all Gerber files”.

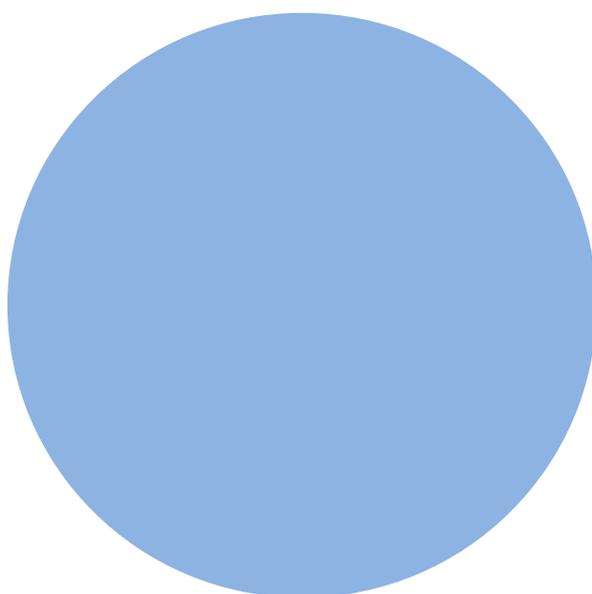
- ◆ The name contained in the Gerber files is used as the layer name.



Note

You can also import Gerber files without using the option “Use layer name”. In this case, you have to assign the files manually to the corresponding layer.

Appendix





Appendix

Fig. 1: Connection steps.....	14
Fig. 2: New document	15
Fig. 3: Import	16
Fig. 4: Assigned layers	17
Fig. 5: CAM view	18
Fig. 6: Create rectangle.....	19
Fig. 7: Rectangle around the pads	19
Fig. 8: Step & Repeat	21
Fig. 9: Layout in CAM view.....	22
Fig. 10: Create circle	23
Fig. 11: Two fiducials.....	24
Fig. 12: Technology Dialog.....	25
Fig. 13: Tool magazine.....	26
Fig. 14: Inserting the tool.....	26
Fig. 15: Tools in the tool magazine	27
Fig. 16: Machining > Process all	28
Fig. 17: Material settings	29
Fig. 18: Right corner.....	30
Fig. 19: Click on button.....	30
Fig. 20: Left corner	31
Fig. 21: Defined processing area	31
Fig. 22: Placement	32
Fig. 23: Alignment	34
Fig. 24: Connection steps.....	40
Fig. 25: New document	41
Fig. 26: Import	42
Fig. 27: Assign layer.....	43
Fig. 28: Dispense	45
Fig. 29: Machining view with solder paste paths.....	46
Fig. 30: Technology Dialog.....	47
Fig. 31: Wizards > Dispense preparation wizard	49
Fig. 32: Prepare dispenser	49
Fig. 33: Assembling the dispenser	50
Fig. 34: Enter the thickness.....	50
Fig. 35: Position of auxiliary board	51
Fig. 36: Head touching the paper	52
Fig. 37: Dispense icon.....	52
Fig. 38: Camera offset.....	53
Fig. 39: Alignment	53
Fig. 40: Dispense preparation finished.....	54
Fig. 41: Wizards > Dispense process wizard	55
Fig. 42: Mount board	55
Fig. 43: Enter the thickness.....	56
Fig. 44: Select drill hole	57

Fig. 45: Message "Select drill hole"	57
Fig. 46: Alignment	58
Fig. 47: Dispense icon	59
Fig. 48: Process dispensing	60
Fig. 49: Connection steps.....	66
Fig. 50: New document	67
Fig. 51: Material properties.....	67
Fig. 52: CAM view	68
Fig. 53: Create rectangle.....	69
Fig. 54: Rectangle in CAM view	69
Fig. 55: Second rectangle	70
Fig. 56: Create circle	70
Fig. 57: Circle in CAM view	71
Fig. 58: Step Repeat	71
Fig. 59: Multiplying the circle in X direction	72
Fig. 60: Select all objects	73
Fig. 61: Context menu	73
Fig. 62: Convert into 2.5D	74
Fig. 63: Properties of the 2.5D object.....	74
Fig. 64: Create 2.5D milling.....	76
Fig. 65: CAM view	77
Fig. 66: 3D view.....	77
Fig. 67: Tool magazine.....	78
Fig. 68: Inserting the tool.....	78
Fig. 69: Assigning the tools in the dialog.....	79
Fig. 70: Assigned tools	79
Fig. 71: Assigning additional tools.....	80
Fig. 72: Tools in the tool magazine	80
Fig. 73: Machining > Process all	81
Fig. 74: Material placement	82
Fig. 75: Pane „Processing“	82
Fig. 76: Autofocus	83
Fig. 77: Cross hair of camera	83
Fig. 78: Message for second position	83
Fig. 79: Move to measuring position	84
Fig. 80: Material surface level defined	85
Fig. 81: Dismount the depth limiter.....	85
Fig. 82: Pneumatic Depth limiter	86
Fig. 83: Fasten depth limiter.....	86
Fig. 84: Mechanical depth limiter.....	87
Fig. 85: Connection steps.....	90
Fig. 86: New document	91
Fig. 87: Material properties.....	91
Fig. 88: CAM view	92
Fig. 89: Importing the STEP file	93
Fig. 90: 3D view.....	94
Fig. 91: Transformation	94



Fig. 92: Rotate [auto].....	95
Fig. 93: 3D view rotatet around Z axis	95
Fig. 94: Move in X and Y direction	96
Fig. 95: Move object in Z direction	96
Fig. 96: Create 2.5D milling.....	97
Fig. 97: 3D view.....	98
Fig. 98: Phases in pane "Toolpath"	99
Fig. 99: Context menu	99
Fig. 100: Create new phase	99
Fig. 101: New phase created	100
Fig. 102: Creating circles.....	101
Fig. 103: CAM view with fiducials.....	101
Fig. 104: Pane "Layers".....	103
Fig. 105: Phase selection list.....	103
Fig. 106: Technology Dialog.....	104
Fig. 107: Alignment	105
Fig. 108: Selecting the fiducials.....	105
Fig. 109: Tool magazine	106
Fig. 110: Inserting the tool.....	106
Fig. 111: Tools in the tool magazine	107
Fig. 112: Connection steps.....	112
Fig. 113: New document	113
Fig. 114: CAM view - multi-layer	113
Fig. 115: Import	115
Fig. 116: Assigned layers	116
Fig. 117: CAM view	116
Fig. 118: Create Rectangle	117
Fig. 119: Rubout Top.....	117
Fig. 120: Rubout Bottom	118
Fig. 121: Step & Repeat	119
Fig. 122: Design in the CAM view	120
Fig. 123: Technology Dialog.....	121
Fig. 124: Tool magazine	122
Fig. 125: Inserting a tool.....	122
Fig. 126: Loaded tool holder.....	123
Fig. 127: Machining > Process all	124
Fig. 128: Material Settings.....	125
Fig. 129: Defining the left corner	126
Fig. 130: Click on the icon	126
Fig. 131: Defining the left corner	127
Fig. 132: Defined processing area	127
Fig. 133: Placement	128
Fig. 134: Alignment	130
Fig. 135: Connection steps.....	138
Fig. 136: New document	139
Fig. 137: Create rectangle.....	140
Fig. 138: Create rectangle.....	141

Fig. 139: Rectangles in the CAM view	142
Fig. 140: Create circle	143
Fig. 141: Drill holes.....	144
Fig. 142: Create Polygon.....	145
Fig. 143: Polygon	145
Fig. 144: Circular marks	146
Fig. 145: Define text	147
Fig. 146: Text in CAM view	148
Fig. 147: Holes in CAM view	149
Fig. 148: Create closed path	150
Fig. 149: Cutouts for the fan.....	151
Fig. 150: Pane „Properties“	151
Fig. 151: Create Circle	152
Fig. 152: Pane “Layers“.....	153
Fig. 153: Technology Dialog.....	154
Fig. 154: Created toolpaths	155
Fig. 155: Tool magazine.....	156
Fig. 156: Inserting a tool.....	156
Fig. 157: Loaded tool holder.....	157
Fig. 158: Machining > Process all	158
Fig. 159: Material settings	159
Fig. 160: Right corner.....	160
Fig. 161: Click on button.....	160
Fig. 162: Left corner	161
Fig. 163: Defined processing area	161
Fig. 164: Placement	162
Fig. 165: Connection steps.....	166
Fig. 166: New document	167
Fig. 167: Import	168
Fig. 168: Assign layers	169
Fig. 169: CAM view	169
Fig. 170: Identical proportion.....	170
Fig. 171: Transformation >Scale	171
Fig. 172: Dimensions changed.....	171
Fig. 173: “Layers” pane	172
Fig. 174: Technology Dialog.....	172
Fig. 175: Insulation details.....	173
Fig. 176: Deactivating functions	174
Fig. 177: Tool magazine	176
Fig. 178: Insert tool.....	176
Fig. 179: Loaded tool holder.....	177
Fig. 180: Machining > Process all	178
Fig. 181: Material Settings.....	179
Fig. 182: Right corner.....	180
Fig. 183: Click on button.....	181
Fig. 184: Left corner	181
Fig. 185: Defined processing area	182



Fig. 186: Placement	182
Fig. 187: Connection steps.....	188
Fig. 188: New document	189
Fig. 189: CAM view	190
Fig. 190: Phases.....	191
Fig. 191: Phases.....	191
Fig. 192: Import	193
Fig. 193: CAM view	194
Fig. 194: "Toolpath" pane	195
Fig. 195: "Layers" pane	196
Fig. 196: Phase selection list.....	196
Fig. 197: Technology Dialog.....	199
Fig. 198: Insulation details.....	200
Fig. 199: Fiducials - details.....	200
Fig. 200: Tool magazine.....	201
Fig. 201: Inserting a tool.....	201
Fig. 202: Loaded tool holder.....	202
Fig. 203: Machining > Process all	203
Fig. 204: Material Settings.....	204
Fig. 205: Defining the right corner	205
Fig. 206: Click on the icon	205
Fig. 207: Defining the left corner	206
Fig. 208: Defined processing area	206
Fig. 209: Placement	207
Fig. 210: Technology Dialog.....	210
Fig. 211: Contour routing details	210
Fig. 212: Drills details	211
Fig. 213: Tool magazine.....	212
Fig. 214: Inserting a tool.....	212
Fig. 215: Loaded tool holder.....	213
Fig. 216: Machining > Process all	214
Fig. 217: Material Settings.....	215
Fig. 218: Defining the right corner	216
Fig. 219: Click on the icon	216
Fig. 220: Defining the left corner	217
Fig. 221: Defined processing area	217
Fig. 222: Selecting a cutout around a fiducial	220
Fig. 223: Properties of the cutout	220
Fig. 224: Aperture lists	221
Fig. 225: Aperture size	221
Fig. 226: Modifying the aperture size	222
Fig. 227: Technology Dialog.....	223
Fig. 228: Contour routing details	224
Fig. 229: Drills details	224
Fig. 230: Tool magazine.....	226
Fig. 231: Inserting a tool.....	226
Fig. 232: Loaded tool holder.....	227

Fig. 233: Machining > Process all	228
Fig. 234: Material Settings.....	229
Fig. 235: Defining the right corner	230
Fig. 236: Click on the icon	230
Fig. 237: Defining the left corner	231
Fig. 238: Defined processing area	231
Fig. 239: Creating a new phase	234
Fig. 240: Technology Dialog.....	236
Fig. 241: Contour routing details	237
Fig. 242: Fiducials - details.....	237
Fig. 243: Tool magazine	238
Fig. 244: Inserting a tool.....	238
Fig. 245: Loaded tool holder.....	239
Fig. 246: Machining > Process all	240
Fig. 247: Material Settings.....	241
Fig. 248: Defining the right corner	242
Fig. 249: Click on the icon	242
Fig. 250: Defining the left corner	243
Fig. 251: Defined processing area	243
Fig. 252: Alignment	244
Fig. 253: Icon Process Planning Wizard	252
Fig. 254: Process planning wizard	252
Fig. 255: Process planning wizard	253
Fig. 256: Selecting the substrate.....	253
Fig. 257: Through hole plating equipment.....	254
Fig. 258: Surface finishing.....	254
Fig. 259: Summary	255
Fig. 260: Icon import.....	256
Fig. 262: Assigned layers	257
Fig. 263: Icon import.....	258
Fig. 264: Import dialog.....	258
Fig. 265: Changing the layer column	259
Fig. 266: Display correct.....	259
Fig. 267: Design displayed in CAM view	260
Fig. 274: Icon Technology Dialog.....	265
Fig. 275: Technology Dialog.....	265
Fig. 276: Basic insulation	266
Fig. 277: Basic insulation, pads double.....	266
Fig. 278: Partial rubout	267
Fig. 279: Complete rubout.....	267
Fig. 280: Basic routing.....	268
Fig. 281: Horizontal gaps routing	268
Fig. 282: Vertical gaps routing.....	269
Fig. 283: Edge gaps routing	269
Fig. 284: Corner gap routing	270
Fig. 285: Equidis-tant gaps routing	270
Fig. 286: Tool drop-down menu	270



Fig. 287: Computation Results	271
Fig. 288: Tab "Machining view"	276
Fig. 289: Machining view	276
Fig. 290: Icon Tool magazine	277
Fig. 291: Tool magazine	277
Fig. 292: Tool list	278
Fig. 293: Insert tool into holder	278
Fig. 294: Tool magazine	278
Fig. 295: Colors of the tool holders	279
Fig. 296: Icon "Process Planning wizard"	280
Fig. 297: Machining > Process all	280
Fig. 298: Material settings	281
Fig. 299: Right corner	282
Fig. 300: Click on button	282
Fig. 301: Left corner	283
Fig. 302: Defined processing area	283
Fig. 303: Placement	284
Fig. 304: Alignment	286
Fig. 305: Icon import	292
Fig. 306: Import dialog	292
Fig. 307: Change column	293
Fig. 308: Selecting the increment	293
Fig. 309: CAM view	293
Fig. 318: Icon import	304
Fig. 319: Import dialog	304
Fig. 320: Imported files	305
Fig. 321: Unrecognised files	307
Fig. 322: Selecting the file format manually	307
Fig. 323: File details	307
Fig. 324: "Layer/Template" column	308
Fig. 325: "Layers" pane	308
Fig. 326: Abnormal design size	309
Fig. 327: Wrong measurement unit	309
Fig. 328: Corrected measurement unit	310
Fig. 329: Incorrect preview	311
Fig. 330: Correct preview	311
Fig. 331: Aperture properties	312
Fig. 332: Available aperture shapes	313
Fig. 333: Modified aperture shape and size	313
Fig. 334: Layer name in Gerber file	314
Fig. 335: Extras > Options	314
Fig. 336: Extras > Options	315
Fig. 337: Import using option "Use layer name"	315
Fig. 338: "Options" tab	316