# CircuitPro PM 2.1

## How-to guides



Order Code: 999 999



## CircuitPro PM 2.1

How-to guides Version 1.0

English

LPKF Laser & Electronics AG Osteriede 7 D-30827 Garbsen Germany

Phone	+49 5131-7095-0
Fax	+49 5131-7095-90
Email	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.	999 999
File name	CircuitPro PM 2.1_HB_V1.0_ENG.docx
Version	1.0
Date created	28.10.2013
Date printed	28.10.2013
Copyright	© 2013 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 installation, putting into operation, operation, and maintenance or trouble-shooting of the system delivered. This document is intended for persons with basic knowledge of installation and operation of softwarecontrolled machines. General knowledge of operational safety as well as basic knowledge of using PCs running Microsoft Windows® are required.

## Availability

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

## Notations

To facilitate the reading and understanding of the document information text attributes, text notations and text structures are used.

Various text attributes, notations, and structures are used to facilitate reading and understanding the document. The text attributes (highlighting) inside this document are defined as follows:

Attribute	Function
Bold	Important information
Italics	Brand name
Bold italics	LPKF brand name
[]	Button
\\	Input/display field
<>	Check box
{}	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 an action.

## Tables

Data, facts, and important contexts are 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.



## Procedures

In this document, procedures or workflows are compiled to step-by-step operation sequences. An individual operation sequence consists of at least three components: Title, step, and result

Component	Description
■ title	Description of the objective of the procedure marked by a preceding "∎"
1. step	Consecutively numbered sequence of an procedure
➡ intermediate result	Intermediate result of an operation step. The procedure continues.
♦ result	Result of the procedure marked by a preceding "◆"

## Symbols and signal words

The following symbols are used to mark important information:

Symbol	Description
	Safety note WARNING – Hazard for persons CAUTION – Damage to the system
	Note A note is an information on the optimum usage of a feature.
	Tip A tip contains additional information.
©	Copyright
R	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	Description
DIN 5008 05-2005	Rules for writing and layouting
VDI 4500 BI.1,2 11-2006	Technical documentation - Definitions and legal basics
DIN 62079 11-2001	Preparation of instructions - Structuring, content and presentation
ISO 12100-2 02-2003	Safety of machinery - Basic concepts, general principles for design - Part 2: Technical principles
DIN EN 60204 01- 2005	Safety of machinery - Electrical equipment of machines - Part 1: General requirements



## Contents

## Part I: First steps in CircuitPro PM

1	Hov	v to produce a PCB	9
	1.1	Switching on the machine and starting CircuitPro	10
	1.2	Selecting a template and creating a new document	11
	1.3	Importing files	12
	1.4	Inserting rubout areas	15
	1.5	Multiplying the design if needed	17
	1.6	Inserting fiducials	18
	1.7	Creating toolpaths	19
	1.8	Loading the tool magazine and assigning the tools to positions	21
	1.9	Starting processing	23
2	Dis	pensing solder paste using the ProtoMat S63 or S103	. 31
	2.1	Starting the machine and CircuitPro	32
	2.2	Selecting a template and creating a new document	33
	2.3	Importing data	34
	2.4	Creating solder paste paths	37
	2.5	Creating toolpaths	39
	2.6	Dispense preparation	40
	2.7	Starting dispensing	46

## Part II: How to do...

1	Cre	ating a 3D part	53
	1.1	Starting the machine and CircuitPro	
	1.2	Selecting a template and creating a new document	55
	1.3	Creating the layout	57
	1.4	Creating 2.5D objects	62
	1.5	Creating toolpaths	65
	1.6	Loading the tool magazine and assigning tools to holder positions	67
	1.7	Starting processing	70
2	Cre	ating a 3D part from a STEP file	77
	2.1	Starting the machine and CircuitPro	
	2.2	Selecting a template and creating a new document	79
	2.3	Importing data	81
	2.4	Creating 2.5D toolpaths	85
	2.5	Inserting fiducials	87
	2.6	Creating toolpaths for fiducials	89
	2.7	Loading the tool magazine and assigning tools to holder positions	90
	2.8	Starting processing	



3	Cre	ating a	a multi-layer PCB with galvanic through-hole plating	ı 101
	3.1	Starting	g the machine and CircuitPro	102
	3.2	Selecti	ng a template and creating a new document	103
	3.3	Importi	ng data	105
	3.4	Insertin	ng rubout areas	107
	3.5	Multiply	ying the design (if necessary)	109
	3.6	Creatin	ng toolpaths	111
	3.7	Loadin	g the tool magazine and assigning tools to holder positions	113
	3.8	Starting	g the processing	115
4	Cre	ating a	a front panel	127
	4.1	Starting	g the machine and CircuitPro	128
	4.2	Selecti	ng a template and creating a new document	129
	4.3	Creatin	ng the front panel design	130
	4.4	Creatin	ng toolpaths	150
	4.5	Loading	g the tool magazine and assigning tools to holder positions	153
	4.6	Start p	rocessing	155
5	Cre	ating a	a polyimide stencil	161
	5.1	Starting	g the machine and CircuitPro	162
	5.2	Selecti	ng a template and creating a new document	163
	5.3	Importi	ng data	164
	5.4	Proces	sing data	166
	5.5	Creatin	ng toolpaths	168
	5.6	Loading	g the tool magazine and assigning tools to holder positions	172
	5.7	Starting	g the processing	174
6	Cre	ating a	a flex-rigid PCB	181
	6.1	Starting	g the machine and CircuitPro	182
	6.2	Selecti	ng a template and creating a new document	183
	6.3	Importi	ng data	187
	6.4	Proces	sing the individual physical layers	192
		6.4.1	Drilling and milling the flexible material	193
		6.4.2	Processing the FR4 material	203
		6.4.3	Processing the prepreg material	212
	6.5	Bondin	g the individual physical layers	225
	6.6	Contou	Ir routing the bonded PCB	226





## **Part III: Basics**

1	Circ	cuitPro: Basic CAM operations	245
	1.1	Executing Process planning wizard	246
	1.2	Importing the Gerber files	250
	1.3	Importing the drill file	253
	1.4	Inserting a rubout area (optional)	256
	1.5	Inserting fiducials (optional)	258
	1.6	Creating toolpaths	260
2	Circ	cuitPro: Basic machining operations	267
	2.1	Switching to Machining view	268
	2.2	Loading the tool magazine	269
	2.3	Starting processing	272
3	Pro	cessing DXF files in CircuitPro	281
3	<b>Pro</b> 3.1	cessing DXF files in CircuitPro Importing the DXF file	<b>281</b>
3	<b>Pro</b> 3.1 3.2	Cessing DXF files in CircuitPro Importing the DXF file Converting the DXF file	281 282 285
3	<b>Pro</b> 3.1 3.2 <b>Pro</b>	Importing the DXF files in CircuitPro Importing the DXF file Converting the DXF file cessing Gerber and Excellon files	281 282 285 293
3 4	<b>Pro</b> 3.1 3.2 <b>Pro</b> 4.1	cessing DXF files in CircuitPro         Importing the DXF file         Converting the DXF file         cessing Gerber and Excellon files         Selecting Gerber and Excellon files	281 282 285 293 294
3	Pro 3.1 3.2 Pro 4.1 4.2	Cessing DXF files in CircuitPro         Importing the DXF file         Converting the DXF file         cessing Gerber and Excellon files         Selecting Gerber and Excellon files         Selecting the file format	281 282 285 293 294 296
3	Pro 3.1 3.2 Pro 4.1 4.2 4.3	cessing DXF files in CircuitPro         Importing the DXF file         Converting the DXF file         cessing Gerber and Excellon files         Selecting Gerber and Excellon files         Selecting the file format         Selecting the desired target layer	
3	Pro 3.1 3.2 Pro 4.1 4.2 4.3 4.4	cessing DXF files in CircuitPro         Importing the DXF file         Converting the DXF file         cessing Gerber and Excellon files         Selecting Gerber and Excellon files         Selecting the file format         Selecting the desired target layer         Setting/correcting size and format	
3	Pro 3.1 3.2 Pro 4.1 4.2 4.3 4.4 4.5	Importing the DXF files in CircuitPro         Importing the DXF file         Converting the DXF file         Converting the DXF file         Selecting Gerber and Excellon files         Selecting Gerber and Excellon files         Selecting the file format         Selecting the desired target layer         Setting/correcting size and format         Viewing/modifying aperture properties	
3	Pro 3.1 3.2 Pro 4.1 4.2 4.3 4.4 4.5 4.6	Importing the DXF files in CircuitPro Importing the DXF file Converting the DXF file cessing Gerber and Excellon files Selecting Gerber and Excellon files Selecting the file format Selecting the desired target layer Setting/correcting size and format Viewing/modifying aperture properties Using the layer name defined in the Gerber file	



1



## 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 fiducialsCreating toolpaths
- vii. Loading the tool magazine and assigning the tools to positions
- viii. Starting production

## Following material is required:

• Base material FR4 copper-clad (18µm) on both sides (order no. 115967)

1



## **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	Connection steps
	Connecting the machine.
	Checking the machine, machine type, firmware.
	Reading settings from the machine.
	Synchronizing the settings.
	Checking if there was an abnormal termination and fixing it.
	Referencing the machine.
	Moving to the startup position.
	Onnecting the camera.

- ➡ 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

SingleSided.cbf SingleSided_Top.cbf DoubleSided_PoConduct.cbf DoubleSided_SilvanicTHP.cbf DoubleSided_SilvanicTHP.cbf DoubleSided_NOTHP.cbf Ruler_ProConduct.cbf Ruler_PoConduct.cbf	PCB with one predefined layer. PCB with one predefined layer on top side. PCB with predefined Top and Bottom layers, prepared for ProConduct process. PCB with predefined Top and Bottom layers, prepared for palvanic THP process. PCB with predefined Top and Bottom layers, prepared for Ean-Contac process. PCB with predefined Top and Bottom layers, promote for Ean-Contac process. PCB with predefined Top and Bottom layers, no THP. PCB with predefined Top and Bottom layers, no THP.
SingleSided.chf SingleSided_Too.chf DoubleSided_PorConduct.chf DoubleSided_SilvaricTP#.chf DoubleSided_SilvaricTP#.chf DoubleSided_SilvaricTP#.chf DoubleSided_NOTIPE.chf 4.aver_PorConduct.chf 4.aver_PorConduct.chf	PCB with one predefined layer. PCB with one predefined layer. PCB with predefined Top and Bottom layers, prepared for PoConduct process. PCB with predefined Top and Bottom layers, prepared for galvanic THP process. PCB with predefined Top and Bottom layers, or prepared for East-Contac process. PCB with predefined Top and Bottom layers, no THP. PCB with predefined Top and Bottom layers, no THP. PCB with predefined Top and Bottom layers, no THP.
IngleSided_Top.dbf DoubleSided_TopConduct.dbf DoubleSided_SalvarcTHP.dbf DoubleSided_SalvarCTHP.dbf DoubleSided_NoTHP.dbf Harry ProConduct.dbf Harry ProConduct_MatTressS.dbf	PCB with one predefined Toye and Bottom layers, prepared for ProConduct process. PCB with predefined Top and Bottom layers, prepared for galvaric TH-P process. PCB with predefined Top and Bottom layers, prepared for EaseContac process. PCB with predefined Top and Bottom layers, no TH-P. PCB with predefined Top and Bottom layers, no TH-P. PCB with predefined Top and Bottom layers, no TH-P.
DoubleSided_ProConduct.cbf DoubleSided_GalvanicTHP.cbf DoubleSided_GalvanicTHP.cbf DoubleSided_RovTHP.cbf Quarter_ProConduct.cbf 4Layer_ProConduct_MultPressS.cbf	PCB with predefined Top and Bottom layers, prepared for ProConduct process. PCB with predefined Top and Bottom layers, prepared for paivance The process. PCB with predefined Top and Bottom layers, prepared for FancContac process. PCB with predefined Top and Bottom layers, no THP. PCB with predefined Top and Bottom layers, no THP. PCB with four predefined layers, prepared for Proconduct process.
DoubleSided_GalvanicTHP.cbf DoubleSided_EasyContac.cbf PoubleSided_NoTHP.cbf 4Usyer_ProConduct.cbf 4Layer_ProConduct_MultiPressS.cbf	PCB with predefined Top and Bottom layers, prepared for galaxiic THP process. PCB with predefined Top and Bottom layers, prepared for Fax-Contac process. PCB with predefined Top and Bottom layers, no THP. PCB with puredefined Tayers, prepared for Proconduct process.
DoubleSided EasyContac.cbf DoubleSided NoTHP.cbf 4.layer_ProConduct.cbf 4.layer_ProConduct.cbf	PCB with predefined Top and Bottom layers, prepared for EasyContac process. PCB with predefined Top and Bottom layers, no THP. PCB with Tour predefined layers, prepared for ProConduct process.
DoubleSided_NoTHP.cbf 4Layer_ProConduct.cbt 4Layer_ProConduct_MultiPressS.cbf	PCB with predefined Top and Bottom layers, no THP. PCB with four predefined layers, prepared for ProConduct process.
4Layer_ProConduct.cbf 4Layer_ProConduct_MultiPressS.cbf	PCB with four predefined layers, prepared for ProConduct process.
4Layer_ProConduct_MultiPressS.cbf	
	PCB with four predefined layers, prepared for ProConduct, MultiPress S process.
4Layer_ProConduct_MultiPressS_DoubleCore.cbf	PCB with four predefined layers, double core, prepared for ProConduct, MultiPress S process.
4Layer_GalvanicTHP.cbf	PCB with four predefined layers, prepared for galvanic THP process.
6Layer.cbf	PCB with six predefined layers.
6Layer_MultiPressS.cbf	PCB with six predefined layers, MultiPress S process.
8Layer.cbf	PCB with eight predefined layers.
8Layer_MultiPressS.cbf	PCB with eight predefined layers, MultiPress S process.
25D_Bottom.cbf	Template for 2.5D operations on bottom side that are completely processed without depth limiter.
25D_Top.cbf	Template for 2.5D operations on top side that are completely processed without depth limiter.
25D_Double.cbf	Template for 2.5D operations on both sides that are completely processed without depth limiter.
Stencil QR 266x380.cbf	Stencil for ProtoPrint
Stencil QR 266x380 short.cbf	Stencil for ProtoPrint, short version
	>

- 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

	The LPKF tutor data are stored in "My Document\LPKF Laser & Electronics\ LPKF CircuitPro 1.5\Example Data\UseCase_DoubleSidedPCB".
Tip	_

- Importing data
- 1. Click on File > Import...
- ➡ Following dialog is displayed:





- 2. Select all files in the folder "UseCase\_DoubleSidedPCB".
- 3. Click on [Open].

1

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

File	Layer	Usage
.BOA	BoardOutline	This layer contains the data for the board outline of the PCB.
.BOT	BottomLayer	This layer contains the data for the bottom side of the PCB.
.TOP	TopLayer	This layer contains the data for the top side of the PCB.
.DRL	DrillUnplated	This layer contains the data for the drillings.

#### 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.



rectangle

#### Inserting rubout areas 1.4

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.
- Click on Insert > Rubout area > RuboutTop. 1.
- Following dialog is displayed: ⇒



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



Fig. 8: IC pads





- 3. Click on [Close].
- The dialog is closed.
- The rubout area was inserted.



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 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



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

Note

- 2. Click on Modify > Step & Repeat.
- ➡ Following dialog is displayed:

Fig. 9: Step & Step Repeat 
Repetition X 2 Y 1 Apply
Distance X 100 mm Y 55 mm Close
Combine to flash, list:

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

- 3. Enter "2", in the  $\Repetition X\$  field.
- 4. Enter "100", in the \Distance  $X \setminus$  field.
- 5. Enter "55" in the \Distance Y\ field.



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

Note

- 6. Click on [Apply].
- The design is multiplied in X direction
- 7. Click on [Close].
- ➡ The dialog is closed.



The design was multiplied.

Fig. 10: Layout in CAM view



## **1.6 Inserting fiducials**

For aligning the top and bottom sides of the circuit board you need fiducials. Fiducials are optical marks or drill holes on the surface of the circuit board.

The fiducials are drilled into the board and have a diameter of 1.5 mm. They are recognised by the cameras of the ProtoMat systems.



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

Note



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. 11:Create circle

Create	circle		
Layer:	Fiducial	~	Apply
	Absolute     O Relative to anchor po	int	Clore
Center:	X: 0 mm 🚔 Y: 0 m	m 🔿 ø: 1,5 mm 🗘	Cluse

- 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. 12:Four fiducials

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



1

## 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:





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 rubout" method is selected.
- 3. Disable the following function by clicking on the corresponding checkmark:
- Pockets





- 4. Click on [Start].
- ➡ The results for the generated toolpaths are displayed:



In the computation results is a warning displayed. This means that there are no source objects for certain functions. This is not a malfunction but merely a hint for the user.



Note

#### Required tools

CircuitPro makes a recommendation to the computation results, which tools should be used for the creation of the calculated toolpaths.

If some of the recommended tools are not available, you can subsequently assign other tools to the calculated toolpaths.

Please note that using tools with another diameter can cause deviations from the computations results.

In the pane "Toolpath" you can assign other tools to the toolpaths.

- 5. Click on [Close].
- ➡ The dialog with the computation results is closed.
- The toolpaths were created.

1



1

## 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. 15: Tool	🔛 Tool magazine		
magazine	Please check if all required tool	s are assigned to the holders.	
	Required tools	Machine tools	
	Spiral Drill 0,4 mm	Click O to pick up the tool with the machine head	
	🗙 Spiral Drill 0,6 mm	Click $\Phi$ to pick up the tool to the corresponding magazine spot	
	🗙 Spiral Drill 1 mm		
	Spiral Drill 2 mm	Holder 🖸 Tool	Tool life spent
	Contour Router 2 min		
		2 🖸 🚺 NONE 💌	
		Please use tool holder check-hoxes to make these functions avail	hle
		Check milling width Discard tool	Drop tool
			ОК
	Note	ond to your needed tools. These tools	tools which are
	required for the	ich are marked with a red "Y"	
	required for the		
	2. Insert the requi	red tool into the tool magazine:	
Fig. 16: 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. 17: 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. 18: Machining > Process all		Machining Process all Start processing all or single toolpath objects.
	Note	Make sure that <process All&gt; 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.</process 
		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.



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. 19: Material settings	Material Settings					8
	Application					Continue
	PCB					Close
	O Front panel/Engraving	(2.5D)				Close
	Properties					Abort
	Material Type	FR4			~	
	Copper Thickness [µm]	18,0				
	Material thickness	10 mm			*	
	Underlay plate thickness	2 mm			-	
	Click into the machine ar milling head to the assor Use the buttons to set th rear comer of the materi Please make sure that th limiter of the machine he the tape used to fix the r	ea to move the clated position. e front left and right al. e milling depth ad does not touch material.	X: Y: Z:	Current head position Omm Omm Omm		
	Material width	80,22 mm			-	
	Material length	107,55 mm			-	
	Surface level [mm]	<undefined></undefined>				
	Material Comers [mm]	(153,06 / 72,41) : (2	33,28	/ 179,96)		

- 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.



1

- 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 rear corner of your material:

Fig. 20: Right rear	Material Settings						8	BOOQNA 中 Z 当
corner	Application					Continue		
	PCB							
	O Front panel/Engraving	(2.5D)				Close		
	Properties					Abort		1 42 -X 20 and -
	Material Type	FR4			~			
	Copper Thickness [µm]	18,0						
	Material thickness	10 mm			٢			
	Underlay plate thickness	2 mm						
	Location	ea to move the lated position. e front left and right al. e milling depth rad does not touch naterial. 217,27 mm 159,27 mm (6,50 / 20,50) : (223	C X: Y: Z: 3,77 / 18	223,77 mm 199,27 mm 0 mm 9,27)				

/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":

Application			Continue	
PCB				
Front panel/Engraving	(2.5D)		Close	
Properties			Abort	X 30 com
Material Type	FR4		~	
Copper Thickness [µm]	18,0		*	
Material thickness	10 mm			
Underlay plate thickness	2 mm		•	
Please make sure that the limiter of the machine he the tape used to fix the limit the tape used to fix the limit the tape used to fix the limit tape used to fix	he milling depth X ead does not touch X material. Y Z	223,77 mm 189,27 mm		
Material width	217,27 mm		•	
Material length	159,27 mm			
Surface level [mm]	<undefined></undefined>			
	(6,50 / 20,50) : (223,77	/ 189,27)		
Material Corners [mm]				





- ➡ The processing area is adapted.
  - d) Using your mouse in the machining view, click on the lower left corner of your material:

Fig. 22:	Lower	left
corner		

aterial Settings					😒 🦻 ○ ⊖ ♀ 斜 /\. 中
Application				Continue	
PCB					
> Front panel/Engraving	(2.5D)			Close	
				Abort	1 2/2 - X. 2/2 carve 1
Properties					
Material Type	FR4		*		
Copper Thickness [µm]	18,0		٢		
Material thickness	10 mm				
Underlay plate thickness	2 mm				
Please make sure that t limiter of the machine h the tape used to fix the	he milling depth lead does not touch material.	X: Y: Z:	149,49 mm 💿 84,69 mm 💿		
Material width	217,27 mm				
Material length	159,27 mm		<b>A</b>		24
	sundafinada				
Surface level [mm]	<undernied></undernied>				

➡ The machine head moves to this position.

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

Fig. 23	Defined
proces	sing area

laterial Settings			8	S O O Q R A # Z 2
Application			Continue	
PCB				
Front panel/Engraving	(2.5D)	J	Close	
			Abort	
Properties				
Material Type	FR4	*		
Copper Thickness [µm]	18,0	* *		
Material thickness	10 mm			
Underlay plate thickness	2 mm	* *		
Please make sure that th limiter of the machine he the tape used to fix the r	he milling depth ead does not touch material.	X:         149,49 mm         Image: Compared and the second and the se		
Material width	74,28 mm			
Material length	104,58 mm			►
Surface level [mm]	<undefined></undefined>			
Material Corners [mm]	(149,49 / 84,69) : (2	223,77 / 189,27)		$\mathbf{\lambda}$

/1/ Defined processing area

➡ The processing area was adapted to the material.

V1.0 - 10.2013

- 4. Click on [Continue].
- The material settings were entered.

1

1

#### Phase "Placement"

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

Following dialog is displayed:

Fig. 24: Placement	Placement	$\mathbf{x}$
	Relative translation [mm]	Apply
	Set center	Continue
	Rotation	Close
	Angle 🛛 ° 💮 🔿 🗘	
	Step and Repeat	
	Number of copies	
	X: 1 Y: 1	
	Spacing between copies [mm]	
	X: 0 mm 🐑 Y: 0 mm 🐑	
	Reset	

- 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.
- If desired, multiply the job by using the settings in the "Step and Repeat" 3. 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.

## 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.

1



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:

iducials:	Phase "DrillFiducial"		~	
	DrillFiducial			
- Alignmer	nt parameters			Find and Center
Fiducia	ls for step-and-repeat	O Use fiducials of original project only Subsection Use outer fiducials of entire multiplied project		Accept Current Position
	Diameter Tolerance	0,3 mm		Retry
	Search Area Length	10 mm		Abort
Mi	nimum Fiducial Quality	90 %	-	Abort
Distance f	for Direct Acceptance	0,1 mm	-	
	Show Fiducial Time	0 s	-	

- 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.



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".

Note



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.

# 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



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



Note

## **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 (3 bar)



For a more uniform dispensing of solder paste, we recommend to use the LPKF honeycomb plate (Order no. 116 148)

Tip



## 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:



- 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. 27: New document

remplace nie	Description		
CircuitPro installed templates			
📓 SingleSided.cbf	PCB with one predefined layer.		
SingleSided_Top.cbf	PCB with one predefined layer on top side.		
DoubleSided_ProConduct.cbf	PCB with predefined Top and Bottom layers, prepared for ProConduct process.		
DoubleSided_GalvanicTHP.cbf	PCB with predefined Top and Bottom layers, prepared for galvanic THP process.		
DoubleSided EasyContac.cbf	PCB with predefined Top and Bottom lavers, prepared for EasyContac process,		
DoubleSided_NoTHP.cbf	PCB with predefined Top and Bottom layers, no THP.		
4Layer_ProConduct.cbf	PCB with four predefined layers, prepared for ProConduct process.		
ALayer_ProConduct_MultiPressS.cbf PCB with four predefined layers, prepared for ProConduct, MultiPress S process.			
#Layer_ProConduct_MultiPressS_DoubleCore.cbf	PCB with four predefined layers, double core, prepared for ProConduct, MultiPress S process.		
# 4Layer_GalvanicTHP.cbf	PCB with four predefined layers, prepared for galvanic THP process.		
🔠 6Layer.cbf	PCB with six predefined layers.		
6Layer_MultiPressS.cbf	PCB with six predefined layers, MultiPress S process.		
BLayer.cbf	PCB with eight predefined layers.		
BLayer_MultiPressS.cbf	PCB with eight predefined layers, MultiPress S process.		
25D_Bottom.cbf	Template for 2.5D operations on bottom side that are completely processed without depth limiter.		
25D_Top.cbf	Template for 2.5D operations on top side that are completely processed without depth limiter.		
25D_Double.cbf	Template for 2.5D operations on both sides that are completely processed without depth limiter.		
📷 Stencil QR 266x380.cbf	Stencil for ProtoPrint		
🗃 Stencil QR 266x380 short.cbf	Stencil for ProtoPrint, short version		
۱ د.	-		

- 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



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




- 2. Select all files in the folder "UseCase\_Dispensing".
- 3. Click on [Open].
- 4. Assign the imported files to the corresponding layers according to the following table:

File	Layer	Usage
.BOA	BoardOutline	This layer contains the data for the board outline of the PCB.
.BOT	BottomLayer	This layer contains the data for the bottom side of the PCB.
.SPT	SolderPasteTop	This layer contains data for dispensing solder paste.
.TOP	TopLayer	This layer contains the data for the top side of the PCB.
.DRL	DrillPlated	This layer contains the data for the drillings.

For dispensing you only need the layers "Solder Paste Top" to dispense solder paste and "DrillPlated" to recognise the position. The data on the other layers may help you to determine the side/alignment of the circuit board.

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

- File N • BottomLayer
   SolderPasteTop 66,42 x 34 mm
   69,13 x 38,76 mm Tutor.BOT V Tutor.SPT - 76,4 x 40,53 mm Tutor.TOP TopLayer DrilPlater Tutor.DR ▼ 76.54 x 42.57 mm Tutor, DRI ns Size 80,63 x 45,72 mm Unit Inches Values Absolut Decima Omit leading zer \* 4 Digits m.n 2 1 Ready
- The files are shown in the table:



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.
- 5. Click on [OK].
- ➡ The data are displayed in the CAM view:

Fig. 29: Assign layer



## Fig. 30: CAM view



## • The data are imported.



In the "Layers" pane you can hide the following for a clear display of the dispense data:

- TopLayer
  - DrillPlated

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. 31: Dispense





The dialog "Dispense" allows you to make different settings concerning solder paste paths. For a detailed description, see the chapter "Dispense" in the CircuitPro compendium or in the online help.

- 2. In the drop down list "Source" select the layer "SolderPasteTop". This layer was assigned to the solder paste data.
- 3. Enable the desired dispenser tool you want to use by marking the corresponding checkbox.

The following table describes the properties of the dispenser 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 $\mu m$ 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 dispenser tool.
	The dispenser tools relate to different pad sizes, which can be filled with the respective tool. This is realized by means of different parameter sets. For all dispenser tools the same dispenser and needle are used.



- 4. Click on [Run].
- ➡ The solder paste paths are created.
- 5. Click on [Close].
- ➡ The dialog is closed.
- 6. Click on the machining view tab.
- ➡ The created solder paste paths are shown in the machining view:

Fig. 32: Machining view with solder paste paths

		•	• •	•	•	•	•	•
• • • •		•	• •					•
1 1 								
÷ ÷ • •	••							
•••••		•	•••		•	•	•	•
		 •	• •	•	•	•	•	•

• The solder paste paths were created.



# 2.5 Creating toolpaths

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:



Material type FR4	✓ ■ RF application	
- Insulate	Topologican Maskaul	
	basic	
	Description	
	Insulation with a single insulation channel.	
1/4	Shortest processing time.	
Process		
Show Details		
- Contour Routing	Contour Pouring Method	
	Corner gap	
	Description	
	Contour Routing with one gap in each corner.	
5/6		
Process		
Show Details		
Convert to Toolpath		
Drills Show Details		
Fiducials Show Details		
Chan Dataile	Start	
Pockets Show Details		_



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 chapter "Technology Dialog" in the CircuitPro compendium or online help.

- 2. Disable the following functions by clicking on the corresponding checkmarks:
- Insulate
- Contour Routing
- Fiducials (only if you do not use your own data including 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:



The following dialog is displayed: •





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

Note

Fig. 35: Prepare

dispenser



- 2. Prepare the solder paste as described in the wizard.
- 3. Assemble the dispenser as described in the wizard:

Fig. 36: 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:





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



Fig. 38: Position

- Setting the material height
- Move the wizard screen off to the side. 1.
- 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:





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. 39: Head touching the paper

icon



- 4. Take off the sheet of paper.
- 5. Click on [Next].
- The material height was set. ٠
- Cleaning the dispenser needle (or prepare needle)
- 1. Click on the dispense icon in the wizard:



- 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. ٠



Fig. 41: Camera offset

## Calculating the dispenser offset

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

	Camera offset	
Prepare dispenser Mount dispenser Mount auxiliary board Move to auxiliary board Auxiliary board height Dispenser Up height Dispenser cleaning Camera offset Finished		Determine the offset between camera and dispenser.

- 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. 42:

Alignment



Fig. 43: Dispense

preparation finished



## 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 in the pane "Processing".

2. If the dot is displayed centered in the pane "Camera" click on [Find and Center] in the dialog "Alignment".

 $\rightarrow$  The camera zooms in the dot.

- 2. Click on [Accept Current Position].
- ➡ The view changes as follows:

Dispense preparation wizard UPKF CircuitPro Dispense preparation finished Overview Prepare dispenser The dispense preparation wizard has finished the preparation steps. Feel free to remove the auxiliary board. Mount dispenser Mount auxiliary board Move to auxiliary board Start the dispense process wizard to dispense on a board. Auxiliary board height Dispenser tip height Dispenser cleaning Camera offset Finished Previous Cancel

- 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. Placement
- iii. Cleaning the needle
- iv. Dispensing the solder paste

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



➡ The following dialog is displayed:

Dispensing wizard LINKE Mount board Overview Mount board Mount the circuit board onto the machine. Measure the height and enter it into the input field. Select toolpath Move camera Select toolpath Move camera Measure board height Dispenser cleaning Process dispensing Dispensing process f... \* Cancel

Fig. 45: 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. 46: Enter the thickness



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

Fig. 47: Select drill hole

Fig. 48: Message "Select drill hole"



## Placement

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



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:

?	Confirmation of focu	is height	
	The focus search has finished the calculated focus position sharp. Click 'Yes' to accept th click 'Cancel' to abort the ope	d and the machine has . Please check whethe he position, click 'No' to eration.	s now moved into r the image is o retry focusing or
	Yes	No	Cancel

## 1. Follow the instructions displayed in the wizard:

Overview	Select a sin	gle drilled	hole		
Mount board Select toolpath					In the following step, you need to select drilled hole close to a corner of the board
Move camera					by clicking on it. You may zoom in to mak the selection easier.
ect toolpath ve camera					Click "Next" to select the drilled hole.
easure board height					
ispenser cleaning					
rocess dispensing	=				
spensing process I	8.			100	

- 2. Click on [Next].
- ➡ Following message is displayed:





- 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. 49: Alignment

Alignmen	t		
Check align center of th	ment result. Click 'Acc he fiducial.	pt Current Position' if the result is OK. Click 'Find and center' to find and then move to th	e
Fiducials:	Phase "DrillFiducial"		~
	DrillFiducial		
- Alignment	t parameters	O Use fiducials of original project only	Find and Center
Fiducials	s for step-and-repeat	Use outer fiducials of entire multiplied project	Accept Current Position
	Diameter Tolerance	0,3 mm	Retry
	Search Area Length	10 mm	Abort
Min	nimum Fiducial Quality	90 %	ADDIT
Distance fi	or Direct Acceptance	0,1 mm {	
	Show Fiducial Time	0 s {	÷
Maximum N	lo. of centering Tries	3	•
Manual Ce	entering If Not Found	Yes O No	
	Ask for Confirmation	O Yes ⊚ No	J



## Read in the drill hole manually if centering failed

Note

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".

ightarrow The camera zooms in the drill hole

- 4. Click on [Accept Current Position].
- 5. Repeat the steps 1-4 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. 50: Dispense	Dispensing wizard	©
	GircuitPro	Laser & Electronics
		and the second sec
	Dispenser cleaning	
	Overview Mount board Select toolpath Move camera Measure board he Dispenser cleaning Process dispensing Dispensing proce	Click the button below to dean the dispenser tip. Clean the surface after this. Click "Next" to continue.

- 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

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]:



- 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



1

# 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.



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

Note

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

## The following material is required:

• 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 used material has at least the dimensions 100 mm x 100 mm and a thickness of 5 mm +/- 1 mm.



# **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:



- 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. 53: New document

emplate file	Description
CircuitPro installed templates	
SingleSided.cbf	PCB with one predefined layer.
SingleSided_Top.cbf	PCB with one predefined layer on top side.
DoubleSided_ProConduct.cbf	PCB with predefined Top and Bottom layers, prepared for ProConduct process.
DoubleSided_GalvanicTHP.cbf	PCB with predefined Top and Bottom layers, prepared for galvanic THP process.
DoubleSided_EasyContac.cbf	PCB with predefined Top and Bottom layers, prepared for EasyContac process.
DoubleSided_NoTHP.cbf	PCB with predefined Top and Bottom layers, no THP.
4Layer_ProConduct.cbf	PCB with four predefined layers, prepared for ProConduct process.
4Layer_ProConduct_MultiPressS.cbf	PCB with four predefined layers, prepared for ProConduct, MultiPress S process.
4Layer_ProConduct_MultiPressS_DoubleCore.cbf	PCB with four predefined layers, double core, prepared for ProConduct, MultiPress S process.
4Layer_GalvanicTHP.cbf	PCB with four predefined layers, prepared for galvanic THP process.
6Layer.cbf	PCB with six predefined layers.
6Layer_MultiPressS.cbf	PCB with six predefined layers, MultiPress S process.
8Layer.cbf	PCB with eight predefined layers.
8Layer_MultiPressS.cbf	PCB with eight predefined layers, MultiPress S process.
25D_Bottom.cbf	Template for 2.5D operations on bottom side that are completely processed without depth limiter
25D Top.cbf	Template for 2.5D operations on top side that are completely processed without depth limiter.
25D_Double.cbf	Template for 2.5D operations on both sides that are completely processed without depth limiter.
Stencil QR 266x380.cbf	Stencil for ProtoPrint
Stencil QR 266x380 short.cbf	Stencil for ProtoPrint, short version
	III.
Set as default	
Location:	

- 2. Select the template "25D\_Top.cbf".
- 3. Click on [OK].
- ➡ The following dialog is displayed:

Fig. 54: Material properties		Material	properties	*
		Please de	fine target material properties.	
		Туре:	AL (EN AW-6012)	•
		Length:	100 mm	
		Width:	100 mm	
		Height:	10 mm	
		C Save a	as default properties	
			OK Cancel	
	L			

- 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  $\$  ,  $\$  ,  $\$  and  $\$  .
- 6. Click on [OK].



The CAM view is changing:



- 7. Click on File > Save As...
- 8. Enter a file name for the new file.
- 9. Click on [Save].
- The new document is created.



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

# **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. 56: Create	Create rec	tangle	8
rectangle	Layer:	2.5D MillingTop	Apply
		Absolute O Relative to anchor point	Close
	Start point:	x: 0 mm 💽 Y: 0 mm	
	End point:	x: 0 mm 💽 y: 0 mm	

- 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







- 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. 58: Second rectangle



- 9. Click on [Close].
- The rectangles for the pockets are created.



1

- Creating circles for drill holes
- 1. Click on Insert > Circle.
- The following dialog is shown

Fig. 59: Create circle

Layer:	2.5	D DrillingTop			~		Apply
	0	Absolute 🔵 Rel	ative to ancl	nor point			Close
Center:	X:	0 mm	Y:	0 mm	ø	0 mm	

- 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  $\emptyset$ .



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  $\langle \emptyset \rangle$ .

- 6. Click on [Apply].
- ➡ The circle is shown in the 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...".



➡ The following dialog is shown:



- 4. Enter "5" in the field  $\mathbb{R}$
- 5. Enter "5" in the field Distance X.



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

Note

- 6. Click on [Apply].

value.

The drawn circle is multiplied in X direction.

Fig. 62: Multiplying the circle in X direction

1

- 7. Click on [Close].
- ➡ The dialog is closed.
- The drawn circle was multiplied.

Assign layer



Note

The layer which contain the drawn objects can be changed

manually, if necessary.1. Highlight the objects.

- 2. Processing the objects.
- Press your right mouse button.
   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.

 $\rightarrow$  The objects were moved to the desired layer.



### **Creating 2.5D objects** 1.4

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
- Select all objects in the CAM view: 1.

Fig. 63: Select all objects



- 2. Press your right mouse button.
- The context menu is displayed.

Fig. 64: Context Hide in 2D Display in 3D \* \* Cut Сору 金. Transform.. Step Repeat. 📑 Export. Convert to 2.5D. Y Close open path Combine open paths 7 Convert to polygon **V** Convert to closed path Convert to toolpath ۲ Assign objects to layer ۲ Copy objects to layer ۲ Delete × Z Material properties...

- Click on "Convert to 2.5D...". 3.
- The following dialog is displayed:

menu



Fig. 65: Convert into 2.5D

Destination layer:	Same as source layer	*	Convert
Z value:	0,01 mm	*	dere



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.

- Note
- 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. 66:	Properties		X
Properties of the	Dimensions		
2.5D object	Z depth	3 mm	
	Name	2.5D Object	
	ID	946	
	General		
	📑 Camera 📑 Navigation	Properties	

- 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.
- $\rightarrow$  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].
- $\rightarrow$  The Z value was changed.
- 6. Click on [Close].
- $\rightarrow$  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:



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



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: ⇒



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

Fig. 69: 3D view

#### The 3D tab displays the 3D part: ⇒





activating this icon in the toolbar.



The 2.5D toolpaths were created.

magazine

1

### Loading the tool magazine and assigning tools to holder positions 1.6

- Loading the tool magazine and assigning the tools to holder positions
- Click on Edit > Tool magazine... 1.
- The following dialog is displayed:



correspond to your needed tools. The tools are examples.

- 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
- 2. Insert the required tools into the tool magazine:

by a green checkmark.

Fig. 71: Inserting the tool

Note



© 2013 LPKF AG

1



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

Fig. 72: Assigning the tools in the dialog	Please check if all required to Required tools C End Hill 2 mm Spiral Drill 2 mm	Nols are assigned to the holders. Machine tools The machine clamp is empty at the moment. Click  O to pick up the tool with the machine head. Click  O to put the tool to the corresponding magazine spot.	
		Holder         Tool         Tool life spent           1         O         End Mil 2 mm (0,00%)         X         0,00%           2         NCNE         X         0,00%         X         0,00%           3         O         Spatal Drill 2 mm (0,00%)         X         0         0,00%         X         0           4         O         Spatal Drill 2,2 mm (0,00%)         X         0	
		Please use tool holder check-boxes to make these functions available.  Check miling width.  Discard tool  CK	

The tools which are displayed in the section "Required tools" are marked with a green checkmark:

✓ Required tools – ✓ End Mill 2 mm ✓ Spiral Drill 2 mm	Machine tools The machine of Click <b>O</b> to p Click <b>Φ</b> to p	<ul> <li>Machine tools</li> <li>The machine clamp is empty at the moment.</li> <li>Click O to pick up the tool with the machine head.</li> <li>Click O to put the tool to the corresponding magazine spot.</li> </ul>						
	Holder 🗖	Tool	Tool life spent	^				
	1 🗖 🖸	End Mil 2 mm (0,00%)	<b>X</b>	0,00%				
	2 🗖 🖸	) Spiral Drill 2 mm (0,00%)	<ul><li>✓ ✓</li></ul>	0,00%				
	3 🗆 🖸	NONE		11				
	4 🗆 🖸	NONE	▼					
	5 🗆 🖸	NONE	▼					
	6 🗆 🖸	NONE	▼					
	7 🗆 🖸	NONE	<b>v</b>					
	Please use too	I holder check-boxes to make these fu	nctions available.					
		miling width Discard	tool Drop tool					

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



Fig. 73: A tools



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

Fig. 74:	Tool magazine							
Assigning additional tools	Please check if all required tools are assigned to the holders.							
	Required tools ———————————————————————————————————	Machine to	iols –					_
	V End Mill 2 mm	The machin	e cla	mp is empty at the moment.				
	V Spiral Drill 2 mm	Click O	to pic	k up the tool with the machine head.				
		Click 🕁	to pu	t the tool to the corresponding magazine	spot.			
		Holder		Tool	Tool life	spent		^
		1	0	End Mil 2 mm (0.00%)	¥ (		0.00%	
						_	0,0070	
		2	0	Spiral Drill 2 mm (0,00%)	v		0,00%	
		3 🗖	0	End Mil 2 mm (0,00%)	X		0,00%	Ш
		4	0	NONE				
		5	0	NONE				
		6	0	NONE				
		70	0	NONE				
			-					~
								-
		Please use	tool h	older check-boxes to make these function	ons available.			
					Dro	p tool		

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



The tools were loaded.



# **1.7 Starting processing**

Starting processing

1. Click on Machining > Process all.

Fig. 76: Machining > Process all		Machining Process all Start processing all or single toolpath objects.					
	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".</process>					
		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].


Fig. 77: Material placement

### Phase "Placement Top"

➡ The following dialog is displayed:

<ul> <li>Material properties –</li> </ul>		Machine properties ————————————————————————————————————	Cont	
Type: AL (EN AW	-6012)	Material side: Top		
Length: 100,00 mr	n		Clo	se
Width: 100,00 mr	n			
Height: 5,00 mm			Ab	ort
Location				
Use the buttons to se	t the front left and	rear right comer	$\mathbf{P}$	
Material corners [mm Angle [°]:	]: (0,18 / 0,00) : (0,00)	: (100,18 / 100,00)		
Material corners [mm Angle [°]: - Surface level	]: (0,18 / 0,00) : (0,00)	: (100,18 / 100,00)		
Material corners [mm Angle [°]: > Surface level Move camera onto m	]: (0,18 / 0,00) : (0,00) aterial.	: (100,18 / 100,00)		
Material corners [mm Angle [°]: > Surface level Move camera onto m Click on the auto focu	]: (0,18 / 0,00) : (0,00) aterial. s button on the pro	: (100,18 / 100,00)		
Material corners [mm Angle [°]: Surface level Move camera onto m Click on the auto focu Use the button to set	]: (0,18 / 0,00) : (0,00) aterial. is button on the pro the surface level.	: (100,18 / 100,00)		

- 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. 78: Pane "Processing"	X/Y - position ng       X/Y - position ng       ID mm       X/Y - position ng       X/Y - position ng       X/Y - position ng       Y: 0,000 mm       Y: 0,000 mm
	Z - positioning Q,1 mm Z: 0,000 Depth limiter removed Select a Head Operate Operate Process All>



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:

- Select a Hea	d b	1
ų	Ŗ	8

1





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. 79: Autofocus

Fig. 80: Cross hair of camera



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





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

Note

Click on following icon in the dialog "Material Placement": 4. The coordinates for the lower left corner were saved.



Following message is displayed:





- 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



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:

Move camera onto material.	
Click on the auto focus button on the processing panel.	
Use the button to set the surface level.	
Material surface level: <undefined></undefined>	

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:

Move to r	measuring pos	sition	
Should the ma determining t	achine move the he he height of the ma	ead to the measuri aterial's surface?	ng position for
		Yes	No

- 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".



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

Note

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



Fig. 82: Move to measuring position Fig. 83: Material surface level defined



➡ The material height is saved:

Type: AL (EN AW-6012) Length: 100,00 mm Width: 100,00 mm Height: 5,00 mm Location Use the camera to teach the material position. Use the buttons to set the front left and rear right comer of the material. Material comers [mm]: (0,00 / 0,00) : (28,00 / 18,00) Angle [°]: (0,00) Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.		Machine properties	Continue
Length: 100,00 mm Width: 100,00 mm Height: 5,00 mm Location Use the camera to teach the material position. Use the buttons to set the front left and rear right comer of the material. Material comers [mm]: (0,00 / 0,00) : (28,00 / 18,00) Angle [°]: (0,00) Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.	Type: AL (EN AW-6012)	Material side: Top	Continue
Width: 100,00 mm Height: 5,00 mm Location Use the camera to teach the material position. Use the buttons to set the front left and rear right comer of the material. Material comers [mm]: (0,00 / 0,00) : (28,00 / 18,00) Angle [°]: (0,00) Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.	Length: 100,00 mm		Close
Height:       5,00 mm       Abort         Location	Width: 100,00 mm		
Use the camera to teach the material position. Use the buttons to set the front left and rear right comer of the material. Material comers [mm]: (0,00 / 0,00) : (28,00 / 18,00) Angle [°]: (0,00) Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.	Height: 5,00 mm		Abort
Use the camera to teach the material position. Use the buttons to set the front left and rear right comer of the material. Material comers [mm]: (0,00 / 0,00) : (28,00 / 18,00) Angle [°]: (0,00) Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.	- Location		
Material corners [mm]: (0,00 / 0,00) : (28,00 / 18,00) Angle [°]: (0,00) Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.	Use the camera to teach the mater Use the buttons to set the front lef of the material.	rial position. t and rear right corner	
Angle [*]: (0,00) Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.	Material corners [mm]: (0,00 / 0	1,00) : (28,00 / 18,00)	
- Surface level Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.			
Move camera onto material. Click on the auto focus button on the processing panel. Use the button to set the surface level.	Angle [°]: (0,00)		
Click on the auto focus button on the processing panel.	Angle [°]: (0,00) Surface level		
Use the button to set the surface level.	Angle [°]: (0,00) Surface level Move camera onto material.		
	Angle [°]: (0,00) Surface level Move camera onto material. Click on the auto focus button on ti	he processing panel.	

- 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. 84: Dismount the depth limiter



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



Fig. 85: Pneumatic Depth limiter

- Dismounting the pneumatic depth limiter (S103)
- 1. Pull down the depth limiter:



/1/ Pneumatic depth limiter

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

Fig. 86: Fasten depth limiter



/1/ Pneumatic depth limiter

/2/ Hose clamp

• The pneumatic depth limiter was dismounted.



- Dismounting the mechanical depth limiter (S63)
- 1. Pull down the 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 dismounted.

### 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.



# 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.



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

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

### The following material is required:

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.



# 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:



- 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
- Click on File > New... 1.
- The following dialog is displayed:

#### Fig. 89: New document

Idefined layer. Idefined layer on top side. Ided Top and Bottom layers, prepared for ProConduct process. Ided Top and Bottom layers, prepared for Parkanic THP process. Ided Top and Bottom layers, prepared for EasyContac process. Idefined layers, prepared for ProConduct process. Idefined layers, prepared for ProConduct, MultiPress S process. Idefined layers, prepared for ProConduct, MultiPress S process. Idefined layers, prepared for ProConduct, THP process. Idefined layers, prepared for ProConduct, MultiPress S process. Idefined layers. Idefined layers.
defined layer, defined layer, red Top and Bottom layers, prepared for ProConduct process. red Top and Bottom layers, prepared for RayConta process. red Top and Bottom layers, prepared for RayConta process. red Top and Bottom layers, no THP. defined layers, prepared for ProConduct proces. defined layers, prepared for ProConduct, MultiPress S process. defined layers, prepared for ProConduct, MultiPress S process. defined layers, NultiPress S process.
defined layer on top side. red Top and Bottom layers, prepared for ProConduct process. red Top and Bottom layers, prepared for galvanic TH-P process. red Top and Bottom layers, prepared for Eso/Contac process. red Top and Bottom layers, no TH-P. defined layers, prepared for ProConduct, MultiPress S process. defined layers, prepared for ProConduct, MultiPress S process. defined layers, prepared for galvanic TH-P process. defined layers. lefined layers.
red Top and Bottom layers, prepared for ProConduct process. red Top and Bottom layers, prepared for alwarch "Phy process. red Top and Bottom layers, prepared for EasyContac process. red Top and Bottom layers, no THP. defined layers, prepared for ProConduct, process. defined layers, prepared for ProConduct, MultiPress S process. defined layers, prepared for galvaria: THP process. lefined layers. Hered layers. MultiPress S process.
red Top and Bottom layers, prepared for galvanic THP process. red Top and Bottom layers, prepared for EarlyContac process. red Top and Bottom layers, pro THP. defined layers, prepared for ProConduct process. defined layers, prepared for ProConduct process. defined layers, prepared for plant. THP process. defined layers. hethed layers. MultiPress S process.
red Top and Bottom layers, prepared for EaryContac process. red Top and Bottom layers, no THPP. redefined layers, prepared for ProConduct, publierses S process. defined layers, prepared for ProConduct, MultiPress S process. defined layers, prepared for galvanic THP process. lefined layers. MultiPress S process.
red Top and Bottom layers, no THP. defined layers, prepared for ProConduct, process. defined layers, prepared for ProConduct, NultiPress S process. defined layers, double core, prepared for ProConduct, MultiPress S process. defined layers, prepared for galvanic THP process. efined layers. NultiPress S process.
defined layers, prepared for ProConduct process. stefined layers, prepared for ProConduct, MultiPress S process. defined layers, onepared for ProConduct, MultiPress S process. defined layers, prepared for galvanic THP process. lefined layers, MultiPress S process.
edefined alyers, prepared for ProConduct, MultiPress S process. edefined alyers, prepared for proConduct, MultiPress S process. edefined layers, prepared for galvanic THP process. efined layers. MultiPress S process.
defined layers, double core, prepared for ProConduct, MultPress S process. defined layers, prepared for galvanic THP process. lefined layers, MultPress S process.
adefined layers, prepared for galvanic THP process. lefined layers. lefined layers, MultiPress S process.
lefined layers. lefined layers, MultiPress S process.
lefined layers, MultiPress S process.
redefined layers.
redefined layers, MultiPress S process.
D operations on bottom side that are completely processed without depth limiter.
D operations on top side that are completely processed without depth limiter.
D operations on both sides that are completely processed without depth limiter.
rint
rint, short version

- 2. Select the template "25D\_Double.cbf".
- 3. Click on [OK].
- The following dialog is displayed: -

Fig. 90: Material properties	Material	properties	
	Please de	efine target material properties.	
	Type:	AL (EN AW-6012)	~
	Width:	100 mm	
	Length:	100 mm	
	Height:	10 mm	
	🗹 Save a	as default properties	
		ОК	
			-

- 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:



- 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



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:



- 2. In the folder "UseCase\_3DPartFromSTEP", select the file "Tutor\_STEP.stp".
- 3. Click on [OK].

Creating a 3D part from a STEP

➡ The 3D view changes as follows:



file



- 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:



3. Click on "Rotate [auto]".

2



The view changes as follows: •

Rotate	Transform		
	Translate Rotate [au	to] Rotate [user] Invert	Apply
	Selected objects: 0		
	Angle	0 ° 💮 🗶	Close
	Standard axes	Intrinsic axes	
	ΘX	O First axis	
	OY	O Second axis	
	ΟZ	O Third axis	

- 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. 96: 3D view rotatet 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. 97: Move in X and Y direction

Fig. 98: Move object in Z 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:



- 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
- Click on Toolpath > 2.5D milling... 1.
- The following dialog is displayed:



- 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.



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.

Note

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.



à

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







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 aligning the top and bottom sides of the circuit board you need fiducials. Fiducials are optical marks or drill holes on the surface of the circuit board.

The fiducials are drilled into the board and have a diameter of 1.5 mm. They are recognised by the cameras of the ProtoMat systems.



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

Note



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.

- Inserting fiducials
- 1. Click on Insert > Fiducial > Create new layer...
- A new layer "Fiducial" is created. The following dialog is displayed:



- 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. 102: CAM view with fiducials



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.

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

Technology Dialog Material type FR4 -RF application Insulate Insulation Method Basic Description Insulation with a single insulation channel. Shortest processing time. Process Show Details Contour Routing Contour Routing Method Corner gap Description Contour Routing with one gap in each corner. Process Show Details Convert to Toolpath Show Details 🔽 Drills Fidu Show Details Pockets Show Details

- 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.

Fig. 103: Technology Dialog



# 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:

Jazine	Please check if all required t Required tools C End Hill 2 mm Spiral Drill 2 mm	Machine t Machine t Tool currer Click O Click Φ	to the cools — ntly loa to pic to pu	holders. Ided is from magazine spot {0}. k up the tool with the machine head. t the tool to the corresponding magaz	ine spot.		
		Holder C	ו	Tool		Tool life spent	^
		1	0	End Mill 2 mm 🗸	<b>v</b>	0,70%	
		2 🗖	Ф	Spiral Drill 2 mm 🗸	<b>v</b>	0,70%	=
		3 🗖	0	NONE			
		4 🗖	0	NONE			
		5	0	NONE			
		6 🗖	0	NONE			
		7 🗖	0	NONE			
		8 🗖	$\cap$	NONE	ſ	l l	×
		Please use	tool h	older check-boxes to make these fun	ctions avai	lable.	
		Che		ling width Discard 1		Drop tool	
						CH OK	
						1	
	The tools	shown ii	n th	ne tool magazine	dialo	a must not	

Note

- 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. 105: 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. 106: Tools in the tool magazine



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



# 2.8 Starting processing

Starting processing

1. Click on Machining > Process all.



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



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 table using double-sided adhesive tape.
- 3. Click on [OK].

2



#### Phase "Placement Bottom"

➡ The following dialog is displayed:



- 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. 109: Pane "Processing"	X/Y - position ng       10 mm       X: 0,000 mm       X: 0,000 mm       X: 0,000 mm       Y: 0,000 mm       Y: 0,000 mm
	Select a Head     Head actions       Operate     Tool information <process all="">     No tool information</process>



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:







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".
  - b) Click on [Autofocus] in the "Processing" pane:

Head actions

Fig. 110: Autofocus

Fig. 111: Cross hair of camera



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





If you use dark/reflective materials the camera view could be too dark. In this case please adjust the camera lens.

- - -

- 4. Click on following icon in the dialog "Material Placement":
- The coordinates for the lower left corner were saved.





2

Following message is displayed:



- 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



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:

Move camera onto mat	erial.		A
Click on the auto focus	button on the processing panel.	$\bigcirc$	
Use the button to set th	ne surface level.		

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:

?	Move to measuring position
	Should the machine move the head to the measuring position for determining the height of the material's surface?
	Yes No

- 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.

Fig. 113: Move to measuring position



2. Check if the autofocus has been performed successfully and if the focused material surface is visible in the pane "Camera".



- 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. 115: Dismount the depth limiter



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

2



Fig. 116: Pneumatic Depth limiter

- Dismounting the pneumatic depth limiter (S103)
- 1. Pull down the depth limiter:



/1/ Pneumatic depth limiter

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

Fig. 117: Fasten depth limiter



/1/ Pneumatic depth limiter

/2/ Hose clamp

• The pneumatic depth limiter was dismounted.



- Dismounting the mechanical depth limiter (S63)
- 1. Pull down the mechanical depth limiter:

Fig. 118: 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 dismounted.

### Phase "Drill Fiducial"



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

Note

• The Spiral Drill 1.5 mm is picked up and the fiducials are drilled.

### Phase "Flip Material "

- ➡ A message prompts you to flip the material.
- 1. Flip the material along the machine's X-axis.
- 2. Click on [OK].



The display in the machining view changes. The position of the layout is adjusted to the position of the work piece. The side of the 3D part to be processed is now the top side.



#### Phase "Material Placement Top"

➡ See phase "Material Placement Bottom".



Note

During the phase "Material Placement Top" a rough alignment of the material is done. The exact alignment is done in the following phase "Read Fiducial Top".

#### Phase "Read Fiducials\_Top"

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.

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:

iducials:	Phase "DrillFiducial"		~	
	DrillFiducial			
Alignmer	it parameters	O the fiducials of adviced exclusion		Find and Center
Fiducia	ls for step-and-repeat	Use nuccais or original project only     Use outer fiducials of entire multiplied project		Accept Current Position
	Diameter Tolerance	0,3 mm	*	Retry
	Search Area Length	10 mm		Abaut
Mi	nimum Fiducial Quality	90 %	*	Abort
Distance 1	for Direct Acceptance	0,1 mm	*	
	Show Fiducial Time	0 s	*	
		2		



Fig. 119: Alignment Fig. 120:

Centering the fiducial



- Manually detecting the fiducial using the camera
- 1. Using the X and Y arrows in the pane "Processing", move the machine head, until the cross hair is placed centered on the fiducial:





For centering the fiducial in the pane "Camera" you need to activate the cross hair. To do so, click on the menu item Camera > Overlay > Switch Crosshair State.

- 2. Click on [Accept Current Position].
- The fiducials were recognized by the camera.

### Phase "Processing 2.5D Top"

The required tools are 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.

# 3 Creating a multi-layer PCB with galvanic throughhole 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

### The following material is required:

- 1-mm-thick multi-layer core with 18-µm-thick copper layers on both sides (order no. 119574)
- 200-µm-thick laminate with a 5-µm-thick copper layer on one side for the outer layers (order no. 119571)



# 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:



- 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. 122: New document

remplate file	Description
CircuitPro installed templates	
SingleSided.cbf	PCB with one predefined layer.
SingleSided_Top.cbf	PCB with one predefined layer on top side.
DoubleSided_ProConduct.cbf	PCB with predefined Top and Bottom layers, prepared for ProConduct process.
DoubleSided_GalvanicTHP.cbf	PCB with predefined Top and Bottom layers, prepared for galvanic THP process.
DoubleSided_EasyContac.cbf	PCB with predefined Top and Bottom layers, prepared for EasyContac process.
DoubleSided NoTHP.cbf	PCB with predefined Top and Bottom layers, no THP.
4Layer_ProConduct.cbf	PCB with four predefined layers, prepared for ProConduct process.
4Layer_ProConduct_MultiPressS.cbf	PCB with four predefined layers, prepared for ProConduct, MultiPress S process.
4Laver_ProConduct_MultiPressS_DoubleCore.cbf	PCB with four predefined layers, double core, prepared for ProConduct, MultiPress S process.
4Layer_GalvanicTHP.cbf	PCB with four predefined layers, prepared for galvanic THP process.
6Layer.cbf	PCB with six predefined layers.
6Layer_MultiPressS.cbf	PCB with six predefined layers, MultiPress S process.
BLayer.cbf	PCB with eight predefined layers.
BLayer_MultiPressS.cbf	PCB with eight predefined layers, MultiPress S process.
25D_Bottom.cbf	Template for 2.5D operations on bottom side that are completely processed without depth limiter.
25D_Top.cbf	Template for 2.5D operations on top side that are completely processed without depth limiter.
25D_Double.cbf	Template for 2.5D operations on both sides that are completely processed without depth limiter.
Stencil QR 266x380.cbf	Stencil for ProtoPrint
🔝 Stencil QR 266x380 short.cbf	Stencil for ProtoPrint, short version
<	
Set as default	

- 2. Select the template "4Layer\_GalvanicTHP.cbf".
- 3. Click on [OK].
- ➡ The template is displayed in the CAM view:



Fig. 123: 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



Importing data

- 1. Click on File > Import...
- ➡ The following dialog is displayed:

Fig. 124: Import

Import   File Name	Format   Aperture/Tool L	ist Laver/Templat	e Size/Format		OK
angest (The fight			- Taken and		UK.
					Cancel
					Add Elo
	Open			2 🗙	AUU FIIE
	Look in: 😂 Use Case_Mult	layer	💌 O 🗊 💷 🖽 -		Remove
	Tutor_MultiLaye	r_4_Lagen.BOA			
	My Recent Tutor_MultiLaye	r_4_Lagen.BOT			
2D View Aperture	Documents Tutor_MultiLaye	r_4_Lagen.LY2			
	Tutor_MultiLaye	r_4_Lagen.LY3 r 4_Lagen.TOP			
	Desktop				
	My Documents				
	My Computer				
			11-11-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-		
	File name:	Folder Selection	×	Open	
	My Network Files of type:	All files (*.*)	~	Cancel	
	L			211	

- 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. 125: Assigned layers

		Format		Aperture/Tool List	Layer/Template		Size/Format	
<b>v</b>	Tutor_MultiLayer_4	GerberX	~	Tutor_MultiLayer_4	BoardOutline	~	40,21 x 75,13 mm	
	Tutor_MultiLayer_4	GerberX	~	Tutor_MultiLayer_< 🗸	BottomLayer	~	37,64 x 67,96 mm	Cancel
~	Tutor_MultiLayer_4	GerberX	~	Tutor_MultiLayer_4 🛩	Layer2	×	38,43 x 73,38 mm	add etc.
~	Tutor_MultiLayer_4	GerberX	~	Tutor_MultiLayer_4	Layer3	~	38,43 x 73,38 mm	AUU FIE
~	Tutor_MultiLayer_4	GerberX	~	Tutor_MultiLayer_<	TopLayer	~	36,47 x 70,17 mm	Remove
~	Tutor_MultiLayer_4	Excellon	~	Tutor_MultiLayer_4 🗸	DrilPlated	~	31,78 x 65,61 mm	
<u>-47.44</u> - <u>24.</u> 79 - <u>21.13</u>					Values Decimal Digits m.n	Ab On 2	solute with leading zeros with l	



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:



♦ The files are imported.

Fig. 126: CAM view

106


# 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:



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



- 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.

3





Fig. 129: Rubout Bottom



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



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

Note

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.



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

Note

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. 130: Step &	Step Repeat							
Repeat	Repetition X 2 🗘 Y 1 🚖 Apply							
	Distance X 50 mm 🐑 Y 1 mm 🐑 Close							
	Combine to flash, list:							

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



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.

- 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:



olobul process sectings		
Material type FR4	RF application	
Insulate		
	Insulation Method	
	Basic	
	Description	
	Insulation with a single insulation channel.	
1/4	Shortest processing time.	
Process		
Show Details		
Contour Routing		
	Corner gap	
	Description	
	Contour Routing with one gap in each corner.	
5/6		
Process		
Show Details		
Convert to Toolpath		
Drills Show Details		
Fiducials Show Details		
Deckets Show Details		Start
POCKELS Show Details		Class

In the technology dialog, several settings 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.

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. Disable the function "Pockets".
- 4. Click on [Start].



The computation results are displayed and the CAM view changes as displayed:



- ➡ The dialog with the computation results of the technology dialog is closed.
- The toolpaths are created.



magazine

#### Loading the tool magazine and assigning tools to holder positions 3.7



are missing are marked by a red "X". 2.

Fig. 135: Inserting a tool



- Assign the tools to the corresponding positions in the dialog. 3.
- The tool holders of the machine are loaded:



Fig. 136: 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.



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



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.

Note

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:

Properties         Material Type       FR4         Copper Thickness [µm]       18         Material thickness       1 mm         Underlay plate thickness       2 mm         Location       2 mm         Click into the machine area to move the milling head to the associated position.         Use the buttons to set the front left and rig rear corner of the material.	ght Currer X: Or	rent head position		
Material Type       FR4         Copper Thickness [µm]       18         Material thickness       1 mm         Underlay plate thickness       2 mm         Location       2 mm         Click into the machine area to move the milling head to the associated position.         Use the buttons to set the front left and rig rear corner of the material.	ght Currer X: []]	rent head position		
Copper Thickness [µm]       18         Material thickness       1 mm         Underlay plate thickness       2 mm         Location       2         Click into the machine area to move the milling head to the associated position.         Use the buttons to set the front left and rig rear corner of the material.	ght Currer X: []]	rent head position		
Material thickness       1 mm         Underlay plate thickness       2 mm         Location	ght Currer X: Or	rent head position		
Underlay plate thickness 2 mm Location Click into the machine area to move the milling head to the associated position. Use the buttons to set the front left and rig rear comer of the material.	ght Currer X: []]	rent head position	n	
Location Click into the machine area to move the milling head to the associated position. Use the buttons to set the front left and rig rear corner of the material.	ght Currer X: Or	rent head position	'n	
	Y: 0 r Z: 0 r	0 mm 0 mm		
Material width 305 mm				
Material length 229 mm			(x)	
Surface level [mm] <undefined></undefined>				
Material Corners [mm] (13,50 / 0,50) : (3	(318,50 / 229,50)	50)		

- 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 rear corner of your material:



The machine head moves to this position.



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

 Fig. 142: Defined processing area
 Location

 Click into the machine area to move the milling head to the associated position.
 Use the buttons to set the front left and right rear comer of the material.

 Use the buttons to set the front left and right rear comer of the material.
 Current head position

 X:
 14,23 mm

 Y:
 3,27 mm

 Z:
 0 mm

- The coordinates of the current head position are saved and 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.



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....

dX: 186,146 r	mm 🏩 dY: 51,541 mm 🌒	Apply
	Set center	Continue
		Close
- Rotation		
Angle 0 °		
Step and Rep	eat	
	Number of copies	
X: 1	Y: 1	
Sp	acing between copies [mm]	
X: 0 mm	🔹 Y: 0 mm 🌨	
	Reset	
	)	

Following dialog is displayed:

Fig. 143: 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.



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.



# 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.

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

?	Confirmation of focus	s height	
	The focus search has finished the calculated focus position. sharp. Click 'Yes' to accept the click 'Cancel' to abort the ope	and the machine ha Please check whethe e position, click 'No' t ration.	s now moved into er the image is o retry focusing or
	Yes	No	Cancel

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:

ducials:	Phase "DrillFiducial"	8	
	DrillFiducial		
- Alignmer	nt parameters —		Find and Center
Fiducia	ls for step-and-repeat	Use nouclais or original project only     Use outer fiducials of entire multiplied project	Accept Current Position
	Diameter Tolerance	0,3 mm 😂	Retry
	Search Area Length	10 mm	Abort
Mi	inimum Fiducial Quality	90 %	Abort
Distance	for Direct Acceptance	0,1 mm	
	Show Fiducial Time	0 s	
Maximum I	No. of centering Tries	3	
Manual C	enterina If Not Found	Yes O No	

1. Enlarge the search area by increasing the value of the field \Search Area Length\.

V1.0 - 10.2013

- 2. Restart the search by clicking on [Start].
- 3. Repeat steps one and two if necessary.

Fig. 144: Alignment



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.



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

Note

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.



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

Note

# 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.



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.



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].



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.

3

# 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.





# 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

# The following material is required:

• 2-mm-thick aluminum sheet with the dimensions 250 mm x 120 mm



# 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:



- 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. 146: New document

CircuitPro installed templates	
	personal de la companya de la company
CarlsCided Tes off	PLB with one predefined laver.
SingleSided_Top.com	PCB with and afred and Detters large and for DefCenture received
Doublesided_proconduct.cbr	PCB with predefined Top and Bottom layers, prepared for Proconduct process.
DoubleSided_GalvanicTHP.com	PCB with predefined Top and Bottom layers, prepared for gavanic THP process.
DoubleSided_EasyContac.cbr	PCB with predefined Top and Bottom layers, prepared for EasyContac process.
Doublesided NoTHP.com	PCB with precenned Top and Bottom layers, no THP.
4 aver_Proconduct.com	PCB with four predefined layers, prepared for Proconduct process.
4 aver_Proconduct_MultiPressS.com	PCB with four predefined layers, prepared for Proconduct, MultiPress 5 process.
4 aver_Proconduct_MultiPresss_Doublecore.com	PCB with four predefined layers, double core, prepared for Proconduct, MultiPress S process.
4Layer_Galvanic ( HP.cor	PCB with the readefined layers, prepared for gavanic THP process.
Claver. M. H. Draw Cale	PCB with six predefined layers.
BLayer_MultiPressS.cot	PLB with six predefined layers, MultiPress S process.
BLayer.cor	PCB with eight predefined layers.
BLayer_MultiPresss.com	PLB with eight predefined layers, MultiPress S process.
250_Bottom.com	Template for 2.50 operations on bottom side that are completely processed without depth limiter.
250_10p.cor	Template for 2.50 operations on top side that are completely processed without depth limiter.
Changel on Decision and	Template for 2.50 operations on both sides that are completely processed without depth limiter.
Sterici QR 200X380.001	Stendi for ProtoPrint, death unarian
Stenci QR 200X380 short.com	Stenci for ProtoPrint, short version
٢	
Set as default	
Stendi QR 266x380 short.cbf	Stencil for ProtoPrint, short version

- 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.

- Creating a rectangle
- 1. Click on Insert > Rectangle.
- ➡ Following dialog is displayed:

Fig. 147: Create	Create rec	tang	gle				×
rectangle	Layer:	Bo	ardOutline			×	Apply
		0	Absolute O Relative to ancho	r point			Clava
	Start point:	X:	0 mm	¥:	0 mm	\$	Close
	End point:	X:	260 mm	¥:	140 mm		

- 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].

4

130



➡ The rectangle is created:

Fig. 148: Rectangle

		*

- 8. Click on [Close].
- ➡ The dialog is closed.
- The dimensions of the front panel are defined.



- Displaying objects within the BoardOutline
- 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":



• The mode has been changed:

Fig. 150: Mode "Thin Line"

		4

- The BoardOutline and all objects within are now displayed in the mode "Thin Line".
- Changing the zero point of the BoardOutline

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.
- The zero point has been moved to the anchor point and is now positioned on the lower left corner.

4



# Inserting cutouts for switches

Two cutouts are needed for switches. Therefore rectangles are created.

- Creating rectangles.
- 1. Click on Insert > Rectangle.
- ➡ Following dialog is displayed:

Fig. 151: Create	Create rectangle						
rectangle	Layer:	Boa	ardOutline			~	Apply
		0	Absolute O Relative to anchor p	oint			Clore
	Start point:	X:	20 mm 🚔	Y:	85 mm	*	Close
	End point:	x:	40 mm 🚖	Y:	115 mm	-	

- 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 Xfield.
- 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. 152: Rectangles in CAM view





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 polygon	is into closed paths
------------------------	----------------------

1. Select the rectangles that you have created.

2. Click on Modify > Convert to path.

 $\rightarrow$  The rectangles are converted to closed paths.

• The cutouts for the switches were inserted.



# Creating and multiplying drill holes for LEDs

In this step holes are created for inserting LEDs. These holes must be multiplied afterwards. The drill holes are created by inserting and multiplying circles on the layer "DrillUnplated".

- Creating a circle
- 1. Click on Insert > Circle.
- ➡ Following dialog is displayed:



Layer:	DrillUnplated			~		Apply
	Absolute	Relative to anc	hor point			Close
Center:	X: 20 mm	🚔 Y:	20 mm	Ø:	5 mm	Cluse

- 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 ø\ diameter field.
- 7. Click on [Apply] to create the circle.
- ➡ The circle is created:

Fig. 154: Created circle	
	•

- 8. Click on [Close].
- ➡ The dialog is closed.
- The circle was created.

© 2013 LPKF AG



- Multiplying the circle
- 1. Select the circle to be multiplied.
- 2. Click on Modify > Step & Repeat.
- 3. Enter "9" into the \Repetition X\ field.
- 4. Enter "7" into the \Distance X\ field.
- 5. Click on [Apply] to multiply the drill hole.
- ➡ The circles were multiplied:

Fig. 155: Multiplied circles

- 6. Click on [Close].
- ➡ The dialog is closed.
- The circle was multiplied.



Creating another series of drill holes

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 ø\ diameter field.
- 7. Click on [Apply] to create the circle.
- ➡ The circle is created.

Fig. 156: Another series of drill holes

 ●K	

- 8. Click on [Close].
- ➡ The dialog is closed.
- Another series of drill holes was created.

Fig. 157: Drill holes



- Multiplying the circle
- 1. Multiply the drill hole in Y direction by using the "Step & Repeat" function.
- 2. Enter "5" into the  $\ Pertine Y$  field.
- 3. Enter "20" into the Distance Y field.
- 4. Click on [Apply] to multiply the drill holes in Y direction.
- ➡ The design now looks as follows:

	20 view / CAM view   Machinig view   Kan Kan Kan Kan Kan Kan Kan Kan Kan Kan	4
• • • • • • • • • • • • • • • • • • • •		•
	••••••	•

• The circle was multiplied.

Fig. 158: Create Polygon

# Inserting markings

An indication of the charge status is to be created above the lower row of LEDs. This will be realized by a polygon.

- Creating a polygon
- 1. Click on Insert > Polygon.
- ➡ Following dialog is displayed:

Layer:	TopLay	/er			~	Apply
	Abs	olute O Relative	to anchor po	int	0	Close
Start point:	X:	17,5 mm	Ç Y	30 mm		Close
Segment	X:	80 mm	Ş Y	30 mm	-	Set
O Arc	Clockw	ise			~	
	Radius	: 0 mm	<u>)</u> a	0 •	1	Remove last

- 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 Xfield.
- 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. 159: Polygon	30 view CAM view Machining view	4 Þ 🗙
	🔀 🔀 4. 🕲 🕲 🖓 🖓 🖓 🖓 🖓 100	
	*	

• 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  $\emptyset$ \ diameter field.
- 7. Click on [Apply] to create the circle.
- ➡ The circle is created:

Fig. 160: Circular mark

•		
•	•	
	•	
•	••••••	

- 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:



• The circular marks are created around the drill holes.

4



# **Creating legends**

In this step the elements on the front panel are labeled.

- 1. Click on Insert > Text.
- ➡ Following dialog is displayed:

# Fig. 162: Define text

Arial	🖌 10 mm 🚔	Standard-Zeichensatz	✓ B 1	1 <u>U</u> <del>S</del>	Apply
Power					Close
			Characters 5 of I	maximal 1000	
Text object	<create new=""></create>			~	
Destination	Layer "TextTop"			~	
Mode	Filled Area			~	
Mode Character spacing	Filled Area 0,1 mm			×	
Mode Character spacing Line spacing	Filled Area 0,1 mm 2,5 mm			*	
Mode Character spacing Line spacing Position	Filled Area 0,1 mm 2,5 mm X 15 mm	Y 120	0	<ul> <li>4)</li> <li>4)</li> <li>4)</li> </ul>	
Mode Character spacing Line spacing Position Alignment	Filled Area           0,1 mm           2,5 mm           X           15 mm           X           Left	<ul> <li>Y 120</li> <li>Y Bo</li> </ul>	) ttom	> () () () () () ()	

- 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.

142


- ➡ 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. 163: Text in CAM view

20 view / CAM view Machining view	4
🔀 🔯 🔍 🗶 🧶 🏹 🎧 🖓 🖓 Curve -	
Power Laser	
Gate	
• Trigger	
Charge Overload	
Charge	
Error	
•••••	

• The legends were created.



### Creating holes for mounting the front panel

In this step drill holes are created for mounting the front panel later. Therefore circles must be created on the layer "Drill Unplated".

- Creating circles
- 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 ø\ 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:

• Bower Lasor	
Fower Laser	
Gate	
Trigger	
obarra a Overload	
Charge	
•••••••	

- 9. Click on [Close].
- ➡ The dialog is closed.
- The circles for mounting the front panel were created.

Fig. 1 CAM



### Creating cutouts for a fan

This task is to create cutouts for a fan to be able to mount a fan later. The required objects are created by using the function "Open path".

- Creating a open paths
- Click on Insert > Open Path... 1.
- Following dialog is displayed: -



- 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.



# path



9. Repeat the steps 4-9 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	48 227	
161	61	114	
161	79	114	
183	30,5	244,5	92
196	26 249		79
214	4 26 249		61

➡ The new design now looks as follows:

Fig. 167: Cutouts for the fan



10. Click on [Close].

- ➡ The dialog is closed.
- The open paths were created.

■ Changing the line width afterwards

If you want to change the width of the open path afterwards, act as follows: Now the width of the open paths has to be changed to create the cutouts.

- 1. Select one of the created paths.
- 2. Click in the pane "Properties".
- 3. In the pane "Properties" select the field \Width\.
- 4. Overwrite the value by entering 5 mm.
- 5. Press "Enter" on your keyboard.

4



Fig. 168: Pane "Properties"

Pr	operties	<del>-</del> ₽ X
Ξ	Object data	
	Туре	Open path
	Layer	BoardOutline
	Aperture list	
	Aperture	
	Shape	Undefined
Ξ	Dimensions	
	Width	5 mm
	Length	49,497 mm
	Area	267,133 mm <sup>2</sup>
	Perimeter	164,2 mm
÷	Elements	
Ŧ	Bounding box	
Ŧ	Transformation	
ŧ	Points	
ŧ	General	
2.0	(idth	
	ath width	
14	arriviuuri	

➡ The line width has changed.



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.

- 6. Repeat the steps 10-14 to change the line width of the remaining six open paths to 5 mm.
- ➡ The layout changes as follows:



The line width was changed.

Fig. 169:Line width changed



### Creating drill holes for mounting the fan

This task is to create drill holes for mounting the fan. The holes are created by using the function "Create circle".

- Creating circles
- 1. Click on Insert > Circle.
- ➡ The following dialog is displayed:



- 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:

Fig. 171: Created circle



- ➡ The circle was created.
- 8. Repeat the steps 4-7 using the following values to create three more circles:

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

Table. 3: Values for the circles



➡ The layout changes as follows:

Fig. 172: Created circles	• Power Laser		•
		Laser	
		Gate	
		• Trigger	
	Charge	Overload	
		Error	• // // •
	•		٠

- 9. Click on [Close].
- ➡ The dialog is closed.
- The circles 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. 173: Pane "Layers"

Name	Vis	Sel	Color	s Mode		Tech		Inv	Phase	_	[Z]
Fiducial (0)				True Width	~	Fiducials	~		DrillFiducial	~	1
DrillPlated (0)				True Width	~	Drilling	~		DrillingPlated	~	2
DrillUnplated (3)		•		True Width	~	Drilling	~		DrillingUnplated	~	3
SilkScreenTop (0)				True Width	~	Silk Screen	~		Undefined	~	4
SolderPasteTop (0)				True Width	~	Solder Paste	~		Dispense	~	5
SolderMaskTop (0)				True Width	~	Solder Mask	<		Undefined	~	6
TopLayer (0)		•		True Width	~	Wiring	~		MillingTop	~	7
TextTop (8)		•		True Width	~	Wiring	~		MillingTextTop	~	8
RuboutTop (0)				Outline	~	Rubout	~		MillingTop	~	9
PocketTop (0)	V	◄		Thin Line	~	2.5D milling top	<		MillingPocketTop	~	10
BoardOutline (0)	V	◄		Thin Line	~	Mechanical	<		ContourRouting	~	12
PocketBottom (0)	⊻	◄		Thin Line	~	2.5D milling bottom	<		MillingPocketBottom	~	13
RuboutBottom (0)		•		Outline	~	Rubout	~		MillingBottom	~	17
TextBottom (0)		•		True Width	~	Wiring	~		MillingTextBottom	~	18
BottomLayer (0)		•		True Width	~	Wiring	~		MillingBottom	~	19
SolderMaskBottom (0)		•		True Width	~	Solder Mask	~		Undefined	~	20
SolderPasteBottom (0)	•	✓		True Width	~	Solder Paste	~		Dispense	~	21
SilkScreenBottom (0)				True Width	~	Silk Screen	~		Undefined	~	22

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.

4

Fig. 174: Technology Dialog

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

Material type AL (EN AW-6012) Insulate	RF application
Material type AL (EN AW-6012)	RF application
Insulate	
	Insulation Method
	Basic
	Description
	Insulation with a single insulation channel.
	Shortest processing time.
Show Details	
SHOW DECONS	
Contour Routing	
	Contour Routing Method
	Basic
	Description
	Contour Routing without gaps.
1/6	
Process	
Show Details	
Convert to Toolpath	
Drills Show Details	
Fiducials Show Details	
	Start
Pockets Show Details	
	Close



In the technology dialog, several settings 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.

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.



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. 175: Created toolpaths

Power Laser	<ul> <li>Laser</li> <li>Gate</li> <li>Trigger</li> <li>Overload</li> <li>Error</li> </ul>	
-------------	---	--

• 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:





- 3. In the dialog, assign the tools to the respective tool magazine positions used.
- ➡ The tool holders of the machine are loaded:



- 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.



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



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:

Application     PCB     Front papel/Engraving	(2.5D)			Close
Properties Material Type	AL (EN AW-6012)			Abort
Copper Thickness [µm] Material thickness	18,0 2 mm		A N	
Location Click into the machine an head to the associated po Z-buttons in the processi tool tip to the surface. Use the buttons to set the corner of the material. Please make sure that the the machine head does n to fix the material.	ea to move the milling osition. Use the ng view to lower the e front left and right rear e milling depth limiter of ot touch the tape used	X: Y: Z:	Current head position 0 mm 4 0 m 4	
Material width 3 Material length 2 Surface level [mm] ()	003,5 mm 119,5 mm (-1,00) (6,50 / 20,50) : (310,00 / 24(	0,00)	4 v 4 v	

- 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 as necessary.

4

- 3. Define the processing area:
  - a) Move the dialog "Material Settings" off to the side.
  - b) Click on the position in the machining view that represents the right rear corner of your material:

Fig. 181: Defining	Material Settings		×	1
the right rear	Application		Continue	
corner	O PCB		Continue	
	Front namel/Engradient	(2.5D)	Close	
	Pront panel/Engraving	(230)		
	Properties		nodA	
	Material Type	AL (EN AW-6012) -		
	Copper Thickness [µm]	18,0		
	Material thickness	2 mm		
	Underlau plate thickness	2 mm		
	Condenay plate thickness	· · · · · · · · · · · · · · · · · · ·		
	_ Location			
	Cicki into the machine a head to the associated p 2-buttons in the process tool lip to the surface. Use the buttons to set the comer of the material. Please make sure that the the machine head does to fix the material. Material width Material length Surface level [mm] Material Corners [mm]	rea to move the milling ostion. Use the ing view to lower the e front left and right rear emilling depth limiter of not touch the tape used 284.93 mm 286,04 mm 21 286,04 mm 286,04 mm 2		
				•

- The machine head moves to this position. •
  - c) Click on the corresponding icon in the dialog "Material Settings":

Fig. 182: Click on the icon	Location Click into the machine area to move the milling head to the associated position. Use the Z-buttons in the processing view to lower the tool tip to the surface. Current head position
	Use the buttons to set the front left and right rear corner of the material.
	Please make sure that the milling depth limiter of the machine head does not touch the tape used Y: 206,54 mm
	to fix the material. Z: 4 mm

The coordinates of the current head position are saved and the processing ⇒ area is adjusted.

corner



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



- The machine head moves to this position. ⇒
  - e) Click on the corresponding icon in the dialog "Material Settings":

Fig. 184: Defined processing area		Click into the machine area to move the milling head to the associated position. Use the Z-buttons in the processing view to lower the tool tip to the surface.				
		Use the buttons to set the front left and right rear corner of the material.	X: 19	9,26 mm	▲ ▼	
		Please make sure that the milling depth limiter of the machine head does not touch the tape used	Y: 45	5,02 mm	×	
		to fix the material.	Z: 4	mm		
	· · · ·					

- The coordinates of the current head position are saved and the processing • area has been fit to the material.
- 4. Click on [Continue].
- The material settings were entered.

Fig. 185: Placement

#### Phase "Placement"

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

Following dialog is displayed:

dX: 186,146 m	m 🛃 dY: 51,541 mm		
			Continue
	Set center		Close
Rotation ——			
Angle 0 °	800	00	
Step and Repe	ət		
	Number of copies		
X: 1	Y: 1	*	
Spa	ing between copies [mm]		
X: 0 mm	Y: 0 mm		
	Posst		

- 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.





# 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).



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

Note

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

### The following material is required:

• Polyimide film 125 µm (order no. 108321)



### 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:



- 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. 187: New document

CircuitDes installed termilates	Description
Circuit Pro installed templates	
SingleSided.cbf	PCB with one predefined layer.
SingleSided_Top.cbf	PCB with one predefined layer on top side.
DoubleSided_ProConduct.cbf	PCB with predefined Top and Bottom layers, prepared for ProConduct process.
DoubleSided_GalvanicTHP.cbf	PCB with predefined Top and Bottom layers, prepared for galvanic THP process.
DoubleSided_EasyContac.cbf	PCB with predefined Top and Bottom layers, prepared for EasyContac process.
DoubleSided NoTHP.cbf	PCB with predefined Top and Bottom layers, no THP.
4Layer_ProConduct.cbf	PCB with four predefined layers, prepared for ProConduct process.
4Layer_ProConduct_MultiPressS.cbf	PCB with four predefined layers, prepared for ProConduct, MultiPress S process.
4Layer_ProConduct_MultiPressS_DoubleCore.cbf	PCB with four predefined layers, double core, prepared for ProConduct, MultiPress S process.
4Layer_GalvanicTHP.cbf	PCB with four predefined layers, prepared for galvanic THP process.
6Layer.cbf	PCB with six predefined layers.
6Layer_MultiPressS.cbf	PCB with six predefined layers, MultiPress S process.
BLayer.cbf	PCB with eight predefined layers.
BLayer_MultiPressS.cbf	PCB with eight predefined layers, MultiPress S process.
3 25D_Bottom.cbf	Template for 2.5D operations on bottom side that are completely processed without depth limiter.
25D_Top.cbf	Template for 2.5D operations on top side that are completely processed without depth limiter.
a 25D Double.cbf	Template for 2.5D operations on both sides that are completely processed without depth limiter.
Stencil QR 266x380.cbf	Stencil for ProtoPrint
Stencil QR 266x380 short.cbf	Stencil for ProtoPrint, short version
4	
Set as default	

- 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



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

Fig. 188: Import

								Cancel	
	(a							Add File	
	Look in:	C UseCase_Pol	yimide Stenci	1	<b>x</b>	) 🗊 🕑 🛄	· ·	Remove	
		Tutor.BOA							
2D View Apertures/	My Recent Documents	Tutor.DRL							
		Tutor.TOP							
	Desktop								
	My Documents								
	My Computer								
		File name:	Folder Se	lection		~	Open		
	My Network	Files of type:	All files (*	.")		~	Cancel		

- 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

5





Import	File Name	Format	Aperture/Tool List	Layer/Te	mplate	Size/Format		OK
V	Tutor.SPT	GerberX	Tutor.SPT	<ul> <li>SolderPas</li> </ul>	teTop	<ul> <li>69,13 x 38,76 mm</li> </ul>		
<b>V</b>	Tutor.TOP	GerberX	Tutor.TOP	TopLayer	8	💌 76,4 x 40,53 mm		Cancel
								Add File
								Kemove
2D View	Apertures/Tools	Text View N	1essage View		General	Options		
					Size	76,4 x 40,53 mm		
					Unit	Inches	-	
					Values	Absolute	÷	
<u>50.</u> 90				2	Decimal	Omit leading zeros	*	
					Digits m.n	2 4	(A) (W)	
- - 30.64								
-				8				
-								
-								
<u>- 10.</u> 38				<b>.</b>				
mm	2.12	27.59		78.52				
~								



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:

The data is imported.

Fig. 190: CAM view



### 5.4 Processing data

Fig. 191: Identical

proportion

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:



In order to achieve better resulty 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.



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.

5





4. Select the tab "Scale":

Fig. 192: Trans-
formation >Scale

Individue Notate St	ale Mirror	Apply
Selected objects: 3		
Scale group		Close
Scale each object individent of the second secon	dually	
New size	100 %	

- 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:



• 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:

Layers	t	+		Show empty							×
Name	Vis	Sel	Colors	Mode		Tech	T	Inv	Phase		[Z]
SolderPasteTop (150)	•	•		True Width	~	Wiring	~	•	CuttingSide	~	5
TopLayer (2304)	◄			True Width	~	Unknown	~		CuttingSide	~	7
Frame (1)	◄			Thin Line	~	Unknown	~		Undefined	~	23
Perforation (42)	✓			Thin Line	~	Wiring	~	◄	CuttingSide	~	24
Layers 🔂 Geometry	<b>2</b> 1	oolpal	h   🔁 P	rocessing							

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:

🎦 Technology Dialog	
Global process settings	
Material type Polyimide	
Insulate	
	Insulation Method
	Basic
	Description
	Insulation with a single insulation channel.
1/4	Shorest processing time.
Process	
Show Details	
Contour Routing ————————————————————————————————————	
	Contour Routing Method
	Corner gap
	Description
	Contour Routing with one gap in each corner.
5/6	
Process	
Show Details	
Convert to Toolpath	
Drills Show Details	
Fiducials Show Details	
Pockets     Show Details	Start
Show Details	Close



Fig. 194: pane



Fig. 196: Insulation details

No	ote	<ul> <li>In the technology dialog, several settings can be modified by clicking on the [Show details] buttons.</li> <li>For a detailed description of the individual functions of the technology dialog see the corresponding chapter in the CircuitPro compendium.</li> <li>This example just requires the "Basic" contour routing method.</li> </ul>								
2.	In the o	drop down list "Ma	aterial type" select "Polyimide".							
3.	Click o	n [Show Details] i	n the "Insulate" section.							
⇒	The de	tails for insulatior	are displayed.							
	Insulate	► 55 ₩5	Insulation Method Basic Description Insulation with a single insulation channel. Shortest processing time.							
Si Pi Ai	iource Irimary wailable tools	<wring> layers  Micro Cutter 0,1 mm  Universal Cutter 0,2 mm  Micro Cutter 0,1 mm  End Mill (RP) 0,15 mm  End Mill (RP) 0,15 mm</wring>	Insulation width 0,1 mm  Pads insulation 0,05 mm  Insulation 0,05 mm  Insulation 0,05 mm  Insulation Insulatio							

In the drop down list "Primary", select the tool "Micro Cutter 0.1 mm. 4.

Force insulation

Perform inner insulation 💌 🗹 Replace existing toolpath



Note

5. Click on [Hide details].

End Mill (A') 0,4 End Mill 0,8 mm End Mill 1 mm End Mill 2 mm

- The details are hidden. ⇒
- Deactivate all other functions of the technology dialog except insulation. To 6. achieve this, click on the check marks next the individual sections:



Fig. 197: Deactivating functions

Maturial trans		
Material type Polyimide	KF application	
/ Insulate		
	Insulation Method	
	Basir	
	Description	
	Insulation with a single insulation channel. Shortest processing time.	
1/4		
Process		
Show Details		
Contrast Resting		
Contour Routing	Contour Routing Method	
	Corner gap	
	Description	
	Contour Routing with one gap in each corner.	
Process		
Convert to Toolpath		
Drills Show Details		
Fiducials Show Details		_
	Start	
Pockets Show Details		

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





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.

Toolpa	ith 🛛 🛛
	Phases
•	Toolpaths
	Insulate_SolderPasteTop_Micro Cutter 0,1 mm     Insulate_SolderPasteTop_Micro Cutter 0,1 mm_1     Insulate_Perforation_Micro Cutter 0,1 mm
	Insulate_Perforation_Micro Cutter 0,1 mm_1
	I OOIS
•	
🛃 La	yers 🛃 Geometry 🛃 Toolpath 🛃 Processing

The material is cut through when insulation 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

-ig. 198: Tool nagazine	■ Loading the to     1. Click on Edit >     The following (     Tool magazine     Please check if all required tools	Tool magazine a dialog is displa	e yed:		
	<ul> <li>✓ Required tools</li> <li>✓ Micro Cutter 0,1 mm</li> </ul>	Machine tools The machine clamp is Click $\bigcirc$ to pick up Click $\bigoplus$ to put the Holder $\bigcirc$ 1 $\bigcirc$ $\bigcirc$ $\bigcirc$ $\bigcirc$ 3 $\bigcirc$ $\bigcirc$ $\bigcirc$ 4 $\bigcirc$ $\bigcirc$ $\bigcirc$ $\bigcirc$ 5 $\bigcirc$ $\bigcirc$ $\bigcirc$ $\bigcirc$ 6 $\bigcirc$ $\bigcirc$ $\bigcirc$ $\bigcirc$ 7 $\bigcirc$ $\bigcirc$ $\bigcirc$ 8 $\bigcirc$ $\bigcirc$ $\bigcirc$ $\bigcirc$ Please use tool holder Check milling	empty at the moment. the tool with the machine head. icool to the corresponding magazin Tool icro Cutter 0,1 mm (0,00%) DNE DNE DNE DNE DNE DNE DNE Check-boxes to make these function width	re spot. Tool life spot Tool	ent 0,00%
	Note The tools requare missing ar 2 Insert the required	I shown in the needed tools. ired for the job e marked by a ired tools into	tool magazine di These tools are d are displayed. T red "X". the tool holders of	alog must not co examples. <sup>•</sup> ools required for	rrespond



- 3. Assign the tools to the corresponding positions in the dialog.
- The tool holder in the machine is loaded: ⇒





- 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.



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



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:

Application Application PCB Front panel/Engravin Properties Material Type Copper Thickness [um] Material thickness Underlay plate thickness Underlay plate thickness Underlay plate thickness Use the buttons to set rear comer of the material	g (2.5D) Polyimide 0,1 2 mm s 2 mm area to move the ocitated position. the front left and right rial.	v a a a a a a a a a a a a a	CK Close
Please make sure that limiter of the machine the tape used to fix the	the milling depth head does not touch : material.	X:         0 mm         *           Y:         0 mm         *           Z:         0 mm         *	
Material width Material length Surface level [mm] Material Corners [mm]	303,5 mm 219,5 mm <undefined> (6,50 / 20,50) : (310</undefined>	,00 / 240,00)	

- 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.

4. Define the processing area:

- a) Move the dialog "Material Settings" off to the side.
- Click on the position in the machining view that represents the right rear b) corner of your material:



- The machine head moves to this position.
  - Click on the corresponding button in the dialog "Material Settings": c)



The coordinates of the current head position are saved and the processing ⇒ area is adjusted.

176



Fig. 205: Front left corner d) Click on the position in the machining view that represents the front left corner of your material:

-	(a. a.c.)		Close	
O Front panel/Engraving	(2.5D)			
Properties				
Material Type	Polyimide	2		
Copper Thickness (um)	0,1	조		
Material thickness	2 mm	충		
Underlay plate thickness	2 /mm	33		
use the purchas to set th rear corner of the materi	e more serc and nght al.	Current head position           X:         13,44 mm         32           Y:         10,97 mm         32           Z:         0 mm         33		
Material width	296,26 mm	3		
	206,57 mm	소		
Material length				

### The machine head moves to this position.

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



- The coordinates of the current head position are saved and the processing area has been fit to the material.
- 5. Click on [Continue].
- The material settings were entered.

Fig. 207: Placement



### Phase "Placement"

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

Following dialog is displayed:

dx: 186.146 n	ation [mm]	Apply
		Continue
	Set center	Close
Step and Repe	Number of copies	
Spa	acing between copies [mm]	
X: 0 mm	Y: 0 mm	

1. Click on the job and drag it to the desired position using the mouse.

- Or
- 2. Enter the new position in the dialog.



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.





# 6 Creating a flex-rigid PCB

This tutorial shows you how to produce a flex-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



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

#### Following material is required:

- Base material FR4 18/18µm with the dimensions 229 x 305 mm (order no. 119 574)
- Polyimide film 150µm copper clad (18µm) with the dimensions 210 x 297 mm.



## 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:



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



Fig. 209: New document

### 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:

Template life	Description
CircuitPro installed templates	
SingleSided.cbf	PCB with one predefined layer.
SingleSided_Top.cbf	PCB with one predefined layer on top side.
DoubleSided_ProConduct.cbf	PCB with predefined Top and Bottom layers, prepared for ProConduct process.
DoubleSided_GalvanicTHP.cbf	PCB with predefined Top and Bottom layers, prepared for galvanic THP process.
DoubleSided_EasyContac.cbf	PCB with predefined Top and Bottom layers, prepared for EasyContac process.
DoubleSided_NoTHP.cbf	PCB with predefined Top and Bottom layers, no THP.
Haver_ProConduct.cbf	PCB with four predefined layers, prepared for ProConduct process.
# 4Layer_ProConduct_MultiPressS.cbf	PCB with four predefined layers, prepared for ProConduct, MultiPress S process.
4Layer_ProConduct_MultiPressS_DoubleCore.cbf	PCB with four predefined layers, double core, prepared for ProConduct, MultiPress S process.
#Layer_GalvanicTHP.cbf	PCB with four predefined layers, prepared for galvanic THP process.
6Layer.cbf	PCB with six predefined layers.
6Layer_MultiPressS.cbf	PCB with six predefined layers, MultiPress S process.
BLayer.cbf	PCB with eight predefined layers.
BLayer_MultiPressS.cbf	PCB with eight predefined layers, MultiPress S process.
25D_Bottom.cbf	Template for 2.5D operations on bottom side that are completely processed without depth limiter.
25D_Top.cbf	Template for 2.5D operations on top side that are completely processed without depth limiter.
25D_Double.cbf	Template for 2.5D operations on both sides that are completely processed without depth limiter.
Stencil QR 266x380.cbf	Stencil for ProtoPrint
Stencil QR 266x380 short.cbf	Stencil for ProtoPrint, short version
1	-

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 flex-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].

Fig. 210: CAM

view



➡ The template is displayed in the 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 flex-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 arrow symbol next to "Phases".
- ➡ The phases are displayed:



V Pł	nases	-
	1. MountMaterial	
6	2. MaterialSettings	
6	3. Placement	
	4 Drilleichuria	
144	5 Dran tracCorp	
	5. MillioTovit auco	
	0. Milling TextCayers	
	7. MillingLayer3	
	8. FlipMaterial	
e	9. ReadFiducialLayer2	
Ø.	10. MillingTextLayer2	
	11. MilingLayer2	
6	12. DismountMaterial	
6	13. MountLaminat	
6	14. MaterialSettinos 1	
v 13	15. Prenarel aminatBottom	
100	Renard aminatRottom Shiral Dr. 2 mm	
	Propared aminatoritation (contra a mark 2 mm)	
20	Preparezaninacionation contour router 2 min	
	to thing extended in	
•	17. MillingBottom	=
e e	18. MillingPocketBottom	
0	19. DismountMaterial_1	
6	20. MountLaminat_1	
V	21. PrepareLaminatTep	
	PrepareLaminatTop Spiral Drill 2 mm	
	PrepareLaminatTop Contour router 2 mm	
1		
	23 MillingTon	
	24 MillionDerketTen	
	25 Discountertop	
	23. District investering_2	
	20. Messuager top_bottom	
	27. MountMaterial op	
0	28. MaterialSettings_2	
0	29. ReadFiducialsTop	
C.	30. MarkingDrills	
ď	31. DrillingUnplated	
18	32. ContourRouting	
1	33. DismountMaterial 3	
	34 ApplyProtectionElin	
	OF MeyerProtectorm	
	33. Muuritmateriariup_1	
<u> </u>	30. Read-Iduciais top_1	
a,	37. DrillingPlated	
d.	38. DismountMaterial_4	
Ċ.	39. ProcessProConduct	×

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



Fig. 212: Moving fiducials





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.
- 6. Select the memory location.
- 7. Click on [Save].
- The new document is created.



#### 6.3 Importing data

In this section you will pass through following steps:

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



The LPKF tutor data are stored in folder "My documents\LPKF Laser & Electronics\LPKF CircuitPro 1.5\Example Data\ UseCase\_FlexiRigidPCB".

Tip

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

Fig. 213: Import

Import   File Name	Format	Aperture/Tool List	Layer/Template	Size/Format		ОК	
						Cancel	
	Öffnen				? 🛛	Add File	
	Suchen in:	🗀 UseCase_FlexiF	RigidPCB	💌 🔇 🌶 📂 🛙		Remove	
2D View Apertures,	Zuletzt verwendete D	Tutor_Starrflex.ca	n				
	Desktop						
	igene Dateien						
	Arbeitsplatz						
	Netzwerkumgeb	Dateiname: Fo Dateityp: Al	lder Selection I files (*.*)	<b>v</b>	Öffnen Abbrechen		
🕽 Bereit					.:		

- 2. Select the file "Tutor\_Starrflex.cam".
- 3. Click on [OK].
- The data are displayed in the CAM view:



Fig. 214: 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.



- 2. Delete the following phases in the "toolpath" pane:
  - MillingTextLayer3 through to MillingTextBottom (phases 6-16)
  - MillingPocketBottom through to MarkingDrills (phases 18-30)
  - DismountMaterial3 through to ProcessProConduct (phases 33-39)
  - ReadTopography through to Dispense (phases 41-42)
- 3. Select the desired phases.
- 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 layer "CuttingLaminat":

Fig. 216: "Layers" pane

	<b>T +</b>		Show en	npty								
Name	Vis	Sel	Colors	Mode		Tech		Inv	Phase		[Z]	
Fiducial (4)		•		True Width	•	Fiducials	-		DrillFiducial	•	1	
BoardOutline (1)	•	•		True Width	•	Mechanical	-		ContourRouting	•	13	
BottomLayer (213)	•	•		True Width	•	Wiring	•		Undefined	•	20	
Prepreg (53)	•	V		Thin Line	•	Unknown	-		Undefined	•	24	
BaseMaterial (40)	•	V		True Width	•	Unknown	•		Undefined	•	25	
CuttingLaminat (4)				Thin Line	•	Unknown			Undefined	•	26	
CuttingInside (1)	V	V		Outline	•	Unknown	-		Undefined	•	27	
CuttingLaminat+Core (6)		V		Thin Line	•	Unknown	•		Undefined	•	28	

- 3. In the layer row, click on the arrow in the "Phase" column.
- The phase selection list is displayed:

Fig. 217: Phase selection list

🛅 🗙 Z‡	+ +		Show en	npty								
Name	Vis	Sel	Colors	Mode		Tech		Inv	Phase		[Z]	3
Fiducial (4)		•		True Width	•	Fiducials	-		DrillFiducial	-	1	
BoardOutline (1)	•	•		True Width	•	Mechanical	-		ContourRouting	-	13	
BottomLayer (213)	•	•		True Width	•	Wiring	-		Undefined	-	20	
Prepreg (53)	•	•		Thin Line	•	Unknown	-		Undefined	-	24	
BaseMaterial (40)				True Width	•	Unknown	-		Undefined	-	25	
CuttingLaminat (4)	•	✓		Thin Line	•	Unknown			Undefined	-	26	
CuttingInside (1)	~	•		Outline	•	Unknown	-		Undefined		27	
CuttingLaminat+Core (6)	V	◄		Thin Line	•	Unknown			PrepareCore	N	28	

4. Assign the phase according to the following table:

Layer	Phase
CuttingLaminat	DrillingUnplated
CuttingInside	ContourRouting

- 5. Repeat the steps 2-4 for the layer "CuttingInside".
- The data are assigned to production phases.

6

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.
- $\rightarrow$  The context menu is displayed.
- 3. Click on "Edit" in the context menu.

 $\rightarrow$  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 flex-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 flex-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

#### **Creating toolpaths**

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

Material type PK4	✓ U KF application
Insulate	Insulation Method
	Basic
	Description
	Insulation with a single insulation channel.
1/4	Shortest processing time.
Process	
Show Details	
<ul> <li>Contour Routing</li> </ul>	
	Contour Routing Method
	Corner gap
	Description
	Contour Routing with one gap in each corner.
5/6	
Process	
Show Details	
Convert to Toolpath	
✓ Drills Show Details	
Fiducials Show Details	Stad
Pockets Show Details	Start



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.

Fig. 218: Technology Dialog



- 2. In the dropdown list "Material type" select "Polyimide".
- 3. Click on [Show Details] in the "Insulate" section.
- ➡ The details for insulation are displayed.

Fig. 219:	II Technology Dialog	
Insulation details	C Global process settings	
	Material type Polyimide	
	∠ Insulate	
	Insulation Method	
	Einfach	
	Description	
	Isolierung mit einfachem Isolationskanal Kürzeste Bearbeitungszeit	
	1/4 ☑ Process	
	Hide details	
	Source Layer "BottomLayer" v Insulation width 0,2 mm	
	Primary Universal Cutter 0,2 mm 👻 Pads insulation 0,05 mm 👘 0 👘 channels	
	Available tools 🔯 Universal Cutter 0,2 mm 🔶 Rubout <kein rubout=""> 🗸 Konzentrisch 👻</kein>	
	End Micro Cutter 0,1 mm     End Mill (RF) 0,15 mm     Tolerance     0,002 mm	
	End Mill (RF) 0,25 mm     Generate optimized rubout     Perform inner insulation	
	End Mill Q.8 mm Z Force insulation Z Remove spikes	
	End Mill 1 mm	

- 4. Select the layer "BottomLayer" in the "Source" selection list.
- 5. Click on [Hide details].
- ➡ The details are hidden.
- 6. Click on [Show Details] next to "Fiducials".
- ➡ The details for fiducials are displayed.

Fig. 220: Fiducials - dotails	Convert	to Toolpath ———			
Fiduciais - details	🗹 Drills	Show Details	Source	<fiducials> Layer</fiducials>	-
			Tool	<automatisch zugewiesen=""></automatisch>	-
	riducia	Hide details	Phase	<automatisch zugewiesen=""></automatisch>	-
	Pocket	s Show Details	Tolerance	0,002 mm	
			Replace	existing toolpath	J

- 7. Select the "Fiducials" layer in the "Source" selection list.
- 8. Click on [Hide details].
- ➡ The details are hidden.



- 9. Deactivate the following functions by clicking on the corresponding check marks:
  - Contour Routing
  - Drills
  - Pockets

Fig. 221: Deactivate	Convert to To	olpath	
functions	Drills	Show Details	
	Fiducials	Show Details	
	Pockets	Show Details	

- 10. 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



Required tools     Spiral Drill 1,5 mm     Contour Router 2 mm     Spiral Drill 2 mm     Universal Cutter 0,2 mm	Machine tools The machine clamp is empty at the moment. Click O to pick up the tool with the machine head. Click O to put the tool to the corresponding magazine spot.						
	Holder		Tool		Tool life spent	^	
	1	0	Spiral Drill 1,5 mm (0,00%)	<b>v v</b>	)	0,00%	
	2 🗖	0	Contour Router 2 mm (0,00%)	<b>v</b>	)	0,00%	
	3 🗖	0	Spiral Drill 0,2 mm (0,00%)	M (		0,00%	
	4 🗆	0	Spiral Drill 1,4 mm (0,00%) Spiral Drill 1,6 mm (0,00%)		<b>^</b>		
	5 🗖	0	Spiral Drill 1,7 mm (0,00%)				
	6 🗖	0	Spiral Drill 1,9 mm (0,00%)				
	7 🗆	0	Spiral Drill 2 mm (0,00%) Spiral Drill 2,1 mm (0,00%)		4		
		-	Spiral Drill 2,2 mm (0,00%)		~	~	
	Please us	e tool l	holder check-boxes to make these	functions avai	lable.		
	0				Drop tool		
						OV	
						UN.	

- 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. 223: Inserting a tool





- 3. Assign the tools to the corresponding positions in the dialog.
- The tool holders of the machine are loaded: •



The tools are inserted into the tool holders and assigned accordingly. ٠



### Starting the processing

	1. Click on Machining > Process all.								
Fig. 225: Machining > Pro- cess all		Machining Process all Start processing all or single toolpath objects.							
	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.</process>							

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, with the copper side up, onto the processing area of the machine.
- 2. Fasten the base material onto the table top using the adhesive tape.
- 3. Click on [OK].



6

#### Phase "Material Settings"

➡ The following dialog is displayed:

J. 226: Material	Material Settings					8
ungs	Application ————————————————————————————————————					Continue
	PCB					
	O Front panel/Engraving	(2.5D)				Close
	Properties					Abort
	Material Type	Polyimide		-		
	Copper Thickness [µm]	18		-		
	Material thickness	0,15 mm				
	Underlay plate thickness	k mm				
	Construction					
	Click into the machine a head to the associated p Use the buttons to set th corner of the material. Please make sure that th the machine head does to fix the material.	rrea to move the milling loosition. he front left and right rear he milling depth limiter of not touch the tape used	<b>X:</b> <b>Y:</b> Z:	Current head position 0 mm 0 mm 0 mm		
	Material width	303,5 mm			-	
	Material length	219,5 mm			-	
	Surface level [mm] Material Corners [mm]	<undefined></undefined>	0 501			

- Entering the material settings
- 1. In the dropdown list "Material Type" select "Polyimide".



Note

- 2. Enter "18" µm into the field \Copper Thickness\.
- 3. Enter "0.15" mm into the field \Material Thickness \.

corner



- 4. Define the processing area:
  - Move the "Material Settings" dialog off to the side. a)
  - Click on the position in the machining view that represents the right rear b) corner of your material:



The machine head moves to this position.



- The coordinates of the current head position are saved and the processing area is adjusted.
  - d) Click on the position in the machining view that represents the front left corner of your material:



The machine head moves to this position.

6



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

Fig. 230: Defined processing area

Location Click into the machine area to move the milling head to the associated position. Use the buttors to set the front left and right rear comer of the material.	(	Current head position
Please make sure that the milling depth limiter of the machine head does not touch the tape used to fix the material.	X: Y:	9,58 mm 🗘 23,46 mm
	Z:	0 mm 🜔

- The coordinates of the current head position are saved and 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.



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

Note

use the menu item Modify > Transform....

Following dialog is displayed:

g. 231: lacement	Placement	
	Relative translation [mm]	Apply
	dX: 186,146 mm 🚔 dY: 51,541 mm 🚔	Carthous
	Set center	Continue
		Close
	Rotation	
	Angle O ° 😤 🔿 📿	
	Step and Repeat	
	Number of copies	
	X: 1 Y: 1	
	Spacing between copies [mm]	
	X: 0 mm 🌪 Y: 0 mm	
	Deart	
	Reset	



- 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.



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

Note

#### 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 flex-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

#### 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 arrow 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
MillingPottom	Insulate_BottomLayer_Universal Cutter 0.2 mm
WillingBottom	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. 232: Technology Dialog

Global process settings		
Material type FR4	RF application	
Insulate		
	Insulation Method	
	Basic	
	Description	
	Insulation with a single insulation channel.	
1/4	Shortest processing time.	
Process		
Show Details		
Contour Routing		
	Contour Routing Method	
	Corner gap	
	Description	
	Contour Routing with one gap in each corner	
5/6	contour routing with one gap in each conten.	
Process		
Show Details		
Convert to Toolpath		
Drills Show Details		
Fiducials Show Details		Stad
Pockets Show Details		Statt
		Close



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

Note

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:

	O Tasida	Course	Laver "CuttingInside"	
	<ul> <li>Inside</li> </ul>	Source	Layer Cuttinguiside	•
	O Outside	Tool	Contour Router 2 mm	×
		Tabs position	Equal distance	×
		Gap width	0 mm 💮	
		Distance	0 mm 💮	
		Tolerance	0,002 mm	
			Replace existing toolpath	
			Generate start drills	
		Contos	r Pouting Mathod	
		Contoc	a Rodding Hedrou	
		Basic		
		Descrip	ption	
		Cont	our Routing without gaps.	
1/6				
Process				
Process				

Fig. 233: Contour routing details

6



Fig. 234: Drills details 6

- 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:

Drills	Hide details	Source	Layer "CuttingLami 🗸	Create marking	drills
		Tool	<automatically assi="" td="" 🗸<=""><td>Marking tool</td><td>Universal Cutter 0,</td></automatically>	Marking tool	Universal Cutter 0,
Fiducials	Show Details	Phase	<automatically assi="" td="" 🗸<=""><td>Phase</td><td>MarkingDrills</td></automatically>	Phase	MarkingDrills
Pockets	Show Details	Tolerance	0,002 mm 😂		
		Replace	existing toolpath		

- 8. In the selection list "Source" select the layer "CuttingLaminat", listed below the layer "BaseMaterial".
- 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
  - Fiducials
  - Pockets

Fig. 235: Deactivate functions	Convert to To	oolpath	1	
	🔽 Drills	Show Details	Show Details	
	Fiducials	Show Details		
	Pockets	Show Details		
			)	

- 12. Click on [Start].
- ➡ The toolpaths are created and the computation results are displayed..
- The toolpaths are created.

6



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



Required tools     Spiral Drill 1,5 mm     Contour Router 2 mm     Spiral Drill 2 mm     Universal Cutter 0,2 mm	Machine tools The machine clamp is empty at the moment. Click O to pick up the tool with the machine head. Click $\bigoplus$ to put the tool to the corresponding magazine spot.						
	Holder	כ	Tool		Tool life spent		^
	1	0	Spiral Drill 1,5 mm (0,00%)	• •		0,00%	
	2 🗖	0	Contour Router 2 mm (0,00%)	· ·		0,00%	
	3 🗖	0	Spiral Drill 0,2 mm (0,00%)	•		0,00%	-
	4 🗆	0	Spiral Drill 1,4 mm (0,00%) Spiral Drill 1,6 mm (0,00%)				
	5 🗖	0	Spiral Drill 1,7 mm (0,00%)				
	6 🗖	0	Spiral Drill 1,9 mm (0,00%)				
	7 🗖	0	Spiral Drill 2 mm (0,00%) Spiral Drill 2,1 mm (0,00%)		- R		
		-	Spiral Drill 2,2 mm (0,00%)		×		~
	Please us	e tool l	nolder check-boxes to make these f	unctions avai	lable.		
	C d		ling width Discar		Drop tool		
					1	ОК	
							_

- 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. 237: Inserting a tool





- 3. Assign the tools to the corresponding positions in the dialog.
- ➡ The tool holders of the machine are loaded:



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

6



#### Starting the processing

	<ol> <li>Click on Machining &gt; Process all.</li> </ol>									
Fig. 239: Machining > Pro- cess all		Machining Process all Start processing all or single toolpath objects.								
	Note	Make sure that <process All&gt; 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.</process 								

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].



6

#### Phase "Material Settings"

➡ The following dialog is displayed:

Fig. 240: Material Settings	Application Application PCB Front panel/Engraving Properties Material Type Copper Thickness [µm] Material thickness Underlay plate thickness Underlay plate thickness Location Click into the machine a milling head to the asso Use the buttons to set th rear corner of the mater Please make sure that the limiter of the machine he the tape used to fix the	(2.5D)  FR4  18  1 mm  2 mm  rea to move the clated position.  the front left and right ial.  he milling depth ead does not touch material.	Current head position X: 0 mm	CK Close
	Material width	299 92 mm	Z: 0 mm	
	Material length	216.23 mm	×	
	Surface level [mm]	<undefined></undefined>	×	
	Material Corners [mm]	(9,58 / 23,46) : (309,	50 / 239,69)	
			)	4

- 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 "1" 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 rear corner of your material:



Fig. 241: Defining the right rear corner	Material latings         Veckolini         Veckolini
	<ul> <li>The machine head moves to this position.</li> <li>c) Click on the corresponding icon in the "Material Settings" dialog:</li> </ul>
Fig. 242: Click on the icon	Location         Click into the machine area to move the milling head to the associated position.         Use the buttons to set the front left and right rear corner of the material.         Please make sure that the milling depth limiter of the machine head dies not touch the tape used to fix the material.         X:       207,5 mm         Y:       238,11 mm         Z:       0 mm
	<ul> <li>The coordinates of the current head position are saved.</li> <li>d) Click on the position in the machining view that represents the left corner of your material:</li> </ul>
Fig. 243: Defining the front left corner	Attend Selection         Status

➡ The machine head moves to this position.



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

**CircuitPro PM 2.1** 

Fig. 244: Defined processing area

Laser

Location Click into the machine area to move the milling head to the associated position. Use the buttons to set the front left and right rear corner of the material.	Current head position
Please make sure that the milling depth limiter of the machine head does not touch the tape used to fix the material.	X: 9,12 mm C Y: 19,11 mm C
	Z: [Dimm [d]]

- The coordinates of the current head position are saved and 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 flex-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

#### **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 arrow symbol next to phases "PrepareCore", "DrillingUnplated", and "ContourRouting".
- The toolpaths of the phases are displayed.

The following toolpaths have to be deleted:

Phase	Toolpath	
Droporo Coro	PrepareCore_Contour router 2 mm	
FiepaleCole	PrepareCore_Spiral Drill 2 mm	
Drilling Inplated	Drills_CuttingLaminat_Contour router 2 mm	
DriningOrplated	Drills_CuttingLaminat_Spiral Drill 2 mm	
ContourPouting	ContourRouting_CuttingInside_Contour router 2 mm	
Contour Kouting	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. 245: Selecting a cutout around a fiducial

Fig. 246: Properties cutout



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

🗆 Object data		
Туре	Flash/Drill	
Layer	CuttingLaminat	
Aperture list	4LayerMPS.TOP	
Aperture	D12	
Shape	Circle	
Dimensions		
Diameter	4 mm	
Location		
×	145,5 mm	
Y	-64,5 mm	
Transformation		
Orientation	0*	
Scaling	1	
Mirror	No mirror	
🗆 General		
	Flack - J 12	

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 arrow symbol next to the aperture lists.



I	The aperture lists are displayed:	
Fig. 247: Aperture lists	Geometry	
	Aperture Lists	
	4LayerMPS.TOP	
	D10	
	D11	
	D12	
	D13	
	D14	
	D26	
	D27	
	▶ ApeTutor	
	SiebMeyerTools	
	▶ ToolTutor	
	▶ HPGLPens	
	▶ LPKFQuickLoadTool	
	▶ LpkfDrillingTools	
	▶ LpkfMillingTools	
	▶ LpkfMillingOutlineTool	
	▶ LpkfMillingTextTool	
	LpkfDispenseTools	
	▶ LpkfVacuumTools	
	▶ LpkfSoldStopTool	
	▶ LpkfFilmFoilTool	
	🛃 Layers 🛃 Geometry 🛃 Toolpath 🛃 Processing	

- 6. Click on the arrow 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:

	Pr	operties		
•	E	Aperture properties		
		Aperture list	4LayerMPS.TOP	
		Angle	0.	
	E	Dimensions		
		Form type	Circle	
		A	4 mm	
		В	-1 mm	
		C	-1 mm	
		D	-1 mm	
	E	Measurement		
		Area	0 mm <sup>2</sup>	
	E	BndBox	(0,0) (0,0)	
		Length	0 mm	
		Perimeter	0 mm	
		Show measurement	False	
	E	General		
		Name	D12	
		ID	929	


6

- ➡ 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:

Ξ	Aperture properties		
	Aperture list	4LayerMPS.TOP	
	Angle	0.	
Ξ	Dimensions		
	Form type	Circle	
Т	A	8  mm	
1	В	-1 mm	
	С	-1 mm	
	D	-1 mm	
Ξ	Measurement		
	Area	0 mm <sup>2</sup>	
ŧ	BndBox	(0,0) (0,0)	
	Length	0 mm	
	Perimeter	0 mm	
	Show measurement	False	
Ξ	General		
	Name	D12	
	ID	929	
A Di	imension A of the aperture. Exact mean	ing of this parameter depends on the aperture type	

- 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.



Fig. 250: Technology Dialog



#### Creating toolpaths

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

Material type FR4	RF application
Insulate	
	Insulation Method
	Basic
	Description
	Description
	Insulation with a single insulation channel. Shortest processing time.
1/4	
Process	
Show Details	
Contour Routing	Containe Bradiene Mathead
	Corner gap
	Description
	Contour Routing with one gap in each corner
	contour routing with one gap in each comen
Process	
Show Details	
Convert to Toolnath	
Drills Show Details	
Fiducials Show Details	
	Start
Pockets Show Details	



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 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. 251: Contour	Contour Routing	
routing details	O Inside Source Layer "CuttingInside"	
	Outside Tool Contour Router 2 mm	
	Tabs position Equal distance 👻	
	Gap width 0 mm	
	Distance 0 mm	
	Tolerance 0,002 mm	
	Replace existing toolpath	
	🛃 Generate start drills	
	Contour Routing Method	
	Basic	
	Description	
	Contour Routing without gaps.	
	2/6	
	✓ Process	
	Hide 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.



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:







- 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

Fig. 253: Deactivate functions	Convert to To	Convert to Toolpath				
	🗹 Drills	Show Details				
	Fiducials	Show Details				
	Pockets	Show Details				

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



6

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



- Click on Edit > Tool magazine... 1.
- The following dialog is displayed:



are missing are marked by a red "X". Insert the required tools into the tool holders of the machine: 2.

Fig. 255: Inserting a tool



Fig. 254: Tool magazine



- 3. Assign the tools to the corresponding positions in the dialog.
- ➡ The tool holders of the machine are loaded:



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

6



6

#### Starting the processing

	1. Click on	Machining > Process all.					
Fig. 257: Machining > Proc ess all		Machining Process all Start processing all or single toolpath objects.					
	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.</process>					

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

#### Phase "MountMaterial"

1. Place the base material onto the processing area of the machine. In order to get a leveled result with clear-cut edges, press underlay plates from both sides onto the prepreg material. This prevents the material from slipping.



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

material



#### Phase "Material Settings"

➡ The following dialog is displayed:

Application					
Application					Continue
PCB					Close
O Front panel/Engraving (2	2.5D)				ciose
Properties				_	Abort
Material Type	FR4			•	
Copper Thickness [µm]	0			×	
Material thickness	3,5 mm			*	
Underlay plate thickness	2 mm			-	
Please make sure that the r the machine head does no to fix the material.	milling depth limiter of t touch the tape used	<b>X:</b> <b>Y:</b> Z:	0 mm		
Material width 30	2,09 mm			-	
Material length 20	) mm			-	
Surface level [mm] <1	undefined> ,50 / 30,00) : (308,59 / 50,	00)			

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



PCB is selected as default.

Note

- 2. Enter "0"  $\mu m$  into the field \Copper Thickness\.
- 3. Enter "3.5" mm into the field \Material Thickness\.



- 4. Define the processing area:
  - Move the "Material Settings" dialog off to the side. a)
  - Click on the position in the machining view that represents the right rear b) corner of your material:



The machine head moves to this position.



- The coordinates of the current head position are saved.
  - d) Click on the position in the machining view that represents the front left corner of your material:

Fig. 262: Defining the front left corner	Material Settings	
	Material webh         382,00 mm           Material longh         200 mm           Surface to all family         variaded south           Material Comen (rem)         8,597 / 80,001 (308,597 / 30,00)	

The machine head moves to this position. ⇒



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

Fig. 263: Defined processing area

Click into the machine area to move the milling	Г		0
head to the associated position.		0	$\mathbf{\nabla}$
Use the buttons to set the front left and right rear corner of the material.	_	Current head pos	ition
Please make sure that the milling depth limiter of	X:	6,14 mm	4 5
to fix the material.	Y:	29,88 mm	(A) (V)
	Z	0 mm	

- The coordinates of the current head position are saved and 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.



- 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

 $\rightarrow$  Align the prepreg material to the flexible material. Use the fiducials in the flexible material as reference.

#### III. FR4 material

 $\rightarrow$  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.

- 2. Bond the materials with constant temperature and constant pressure.
- ➡ If you use the MultiPress S system select the press profile "LPKF Set".
- ➡ If you use another press system enter the following parameters:
- Preheating temperature: 250 °C
- Prepressing temperature: 180 °C
- Main pressing temperature: 180 °C
- Prepressing time: 10 min.
- Main pressing time: 60min.
- Prepressing force: 80 N/cm<sup>2</sup>
- Main pressing force: 150 N/cm<sup>2</sup>
- The physical layers are bonded.



## 6.6 Contour routing the bonded PCB

Creating a flex-rigid 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.



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

Inserting a production phase



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. 264: Creating a new phase

Name	ReadFiducials		Apply
Processing Step	Read fiducials top	~	
Link to	<none></none>	~	Close
Processing order	8		
🗖 The data are m	irrored (bottom side data)		
The phase cont	ains toolpath data		

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



Note



- 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.

24

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 PCB.

- Deleting toolpaths
- 1. Click on the "Toolpath" pane.
- 2. Click on the arrow symbol next to "Toolpaths".
- ➡ The toolpaths are displayed.



- 3. Select all existing toolpaths.
- 4. Press the right mouse button in order to open the context menu.
- 5. Click on "Delete" in the context menu.
- 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.



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

Note

Proceed as follows:

- 1. Right-click on the desired phase.
- 2. Click on "Edit" in the context menu.
- 3. Ensure 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:

Material type FR4	<ul> <li>RF application</li> </ul>	
Turniste		
Insulate	Insulation Method	
	Basia	
	Dasic	
	Description	
	Insulation with a single insulation channel.	
1/4	Shortest processing time.	
Process		
Show Details		
Contour Routing	Contour Routing Method	
6-3		
	Equidistant gaps	
	Description	
	Contour Routing with equally spaced page	
	control noting the equily spaced gaps	
0/0		
Show Details		
Convert to Toolpath		
C Drills Show Details		
Eidusiala Chasu Dataita		
a rouciais anow Decails		Start
Pockets Show Details		State



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

Fig. 266: Technology Dialog

CircuitPro compendium.



- 2. Click on the right arrow icon in the "Contour Routing" section until the contour routing method "Equidistant gaps" is displayed.
- 3. Click on [Show details].
- ➡ The details for contour routing are displayed:

Fig. 267: Contour	Contour Routing				
routing - details	O Inside	Source	Layer "BoardOutline"	•	
	Outside	Tool	Contour Router 2 mm	•	
		Tabs position	Equal distance	•	
		Gap width	1 mm		
		Distance	50 mm		
		Tolerance	0,002 mm		
			Replace existing toolpath		
			Generate start drills		
		Equi	ur Routing Method distant gaps		
		Descri	ption		
		Con	tour Routing with equally spaced gaps.		
	5/6				
	Hide details	_			

- 4. Select the "BoardOutline" layer in the "Source" drop-down 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:

Convert to 1	Toolpath ———			
🗹 Drills	Show Details	Source	Layer "Fiducial"	
	1 Edu datala	Tool	<automatically assigned=""></automatically>	
riduciais	Hide details	Phase	<automatically assigned=""></automatically>	
Pockets	Show Details	Tolerance	0,002 mm	
		🛃 Replace (	existing toolpath	
	Convert to Convert to Drills Fiducials Pockets	Convert to Toolpath Drills Show Details Fiducials Hide details Pockets Show Details	Convert to Toolpath Drills Show Details Source Tool Fiducials Hide details Phase Pockets Show Details Tolerance Replace	Convert to Toolpath Drills Show Details Source Layer "Fiducial"  Fiducials Hide details Phase CAutomatically assigned>  Phase CAutomatically assigned>  Replace existing toolpath

- 8. Select the "Fiducial" layer in the "Source" drop-down list.
- 9. Click on [Hide details].
- ➡ The details are hidden.



view

- 10. Deactivate the following functions by clicking on the corresponding check marks:
- Insulate
- Drills
- Pockets

Fig. 269: Deactivating	Convert to Toolpath —	
Tunctions	Drills Show Deta	ils
	Fiducials Show Deta	ils
	Pockets Show Deta	ils

- 11. Click on [Start].
- ➡ The toolpaths are created and the computation results are displayed.
- The CAM view changes as follows: •



The next task is to create the toolpaths for the cutouts around the fiducials.



- Creating the toolpaths for the cutouts around the fiducials
- 1. Select the row "CuttingLaminat" in the "Layers" pane:

Fig. 271: Layer CuttingLaminat

	-					1211			-			
Name	Vis	Sel	Colors	Mode		Tech		Inv	Phase		[2]	
Fiducial (4)		V		True Width	•	Fiducials	-		DrillFiducial	•	1	
BoardOutline (1)		Y		True Width	•	Mechanical	-		ContourRouting	-	13	
BottomLayer (213)	V	Y		True Width	•	Wiring	•		Undefined	-	20	
Prepreg (53)	•	Y		Thin Line	•	Unknown	-		Undefined	-	24	
BaseMaterial (40)	•	V		True Width	•	Unknown	-		Undefined	-	25	_
CuttingLaminat (4)		V		Thin Line	•	Unknown	•		Undefined	•	26	
CuttingInside (1)	•	V		Outline	•	Unknown	-		Undefined	-	27	
CuttingLaminat+Core	~	V		Thin Line	•	Unknown	-		Undefined	•	28	

- 2. Click on Toolpath > Technology dialog.
- ➡ The "Technology Dialog" is displayed.
- 3. Click on the left arrow icon in the "Contour Routing" section until the contour routing method "Basic" is displayed.
- 4. Click on [Show details].
- ➡ The details for the function "Contour routing" are displayed:

Fig. 272: Details	Contour Routing					
contour routing		Inside	Source	<current selection=""></current>	·	
		O Outside	Tool	Contour Router 2 mm	-	
			Tabs position	Equal distance	*	
			Gap width	0 mm		
			Distance	0 mm		
			Tolerance	0,002 mm		
				Replace existing toolpath		
	↓ Ve Process Hide details		Conto Basi Descri	Generate start drills our Routing Method c pition tour Routing without gaps.		

- 5. Select "<Current selection>" in the "Source" drop-down list.
- 6. Activate the radio button "Inside" by clicking on it.
- 7. Deactivate the check box "Replace existing toolpath".
- 8. Click on [Hide details].
- The details are hidden.

6



- 9. Deactivate the following functions by clicking on the corresponding check marks:
- Insulate
- Drills
- Fiducials
- Pockets
- 10. Click on [Start].
- ➡ The toolpaths are created and the computation results are displayed.
- ➡ The CAM view changes as follows:





• The toolpaths for the cutouts around the fiducials have been created.



#### Editing the positions of the breakout tabs

The positions of the breakout tabs have to be moved to achieve a clean-cut edge also around the rigid part of the flexi-rigid circuit board.

- Editing the breakout tabs
- 1. Click on Toolpath > Edit breakout tabs...
- ➡ The following dialog is displayed:



➡ The positions of the breakout tabs are marked by circles in the CAM view.

Fig. 275: Breakout tab position zoomed in



- 2. Click on the breakout tab/circle to be moved.
- 3. Keep the left mouse button pressed and drag the circle to the right to the outer edge of the flexi-rigid design:



Fig. 276: Breakout tab position moved



The X and Y coordinates of the new breakout tab position are displayed in the dialog:

# Fig. 277: Circle moved



4. Repeat the above steps and add another breakout tab position at the left side of the design:

Fig. 278: Left breakout tab

Fig. 279: New breakout tab positions

- 5. Click on [Apply] in the dialog.
- 6. Click on [Close] in the dialog.
- The breakout tabs have been moved and the CAM view changes as follows:



• The breakout tab positions have been edited.



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

- 1. Click on Edit > Tool magazine...
- ➡ The following dialog is displayed:



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. 281: Inserting a tool

Note





6

- 3. Assign the tools to the corresponding positions in the dialog.
- ➡ The tool holders of the machine are loaded:



The tools have been inserted into the tool holders and assigned accordingly.



#### Starting the processing

	1. Click	on Machining > Process all.
Fig. 283: Machining > Process all		Machining Process all Start processing all or single toolpath objects.
	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 pha In the combo box, select the desired phase and green "Start processing" button.</process>	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 green "Start processing" button.</process>
green "Start processing" button. You can also start processing with a specific phase. S desired phase in the combo box and click on the "lade The selected phase and all following phases are proc the correct order.	You can also start processing 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 in individu	have started the processing, the ProtoMat machine executes the job al phases. The phases are displayed in messages:

#### Phase "Mount Material"

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



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

Note

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



#### Phase "Material Settings"

➡ The following dialog is displayed:

Application @ PCB O Front panel/Eng Properties Material Type Copper Thickness Material thickness Underlay plate thick Location Click into the mac milling head to th Use the buttons to rear comer of the Please make sure limiter of the mac the tape used to f	FR4 [J] [I] [J,75 mm] [J,75 m	V Current head position X: 0 mm Y: 0 mm	Close
 Material width	301,37 mm	Z: 0 mm (\$)	
Material length	207,44 mm	\$	
Surface level [mm Material Comers [r	<pre>  <undefined> nm] (6,50 / 20,50) : (307</undefined></pre>	7,87 / 237,44)	

- Entering the material settings
- 1. Select {PCB} in the section "Application".
- 2. Select the material type (FR4).
- 3. Enter "18" µm into the field \Copper Thickness\.
- 4. Enter the thickness of the bonded PCB into the field \Material Thickness\, in this case "1.75 mm".



You can also measure the thickness using callipers.

Тір

corner



- 5. Define the material area:
  - Move the "Material Settings" dialog off to the side. a)
  - b) Click on the position in the machining view that represents the right rear corner of your material:



The milling head moves to this position.



- ⇒ The coordinates of the current milling head position are adopted.
  - d) Click on the position in the machining view that represents the front left corner of your material:



corner



- ➡ The milling head moves to this position.
  - e) Click on the corresponding icon in the "Material Settings" dialog.

Fig. 288: Defined material area

	Location Click into the machine area to move the milling head to the associated position. Use the buttons to set the front left and right rear comer of the machine the milling depth limiter or the machine head does not touch the tape used to fix the material.	Current head position           X:         6,87 mm           Y:         30,25 mm           Z:         0 mm	
--	---	--	--

- The coordinates of the current milling head position are adopted and the material area has been adjusted.
- 6. Click on [Continue].
- The material settings have been configured.

6



#### 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.

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 that 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:

Alignmer	nt		×
Check align center of t	nment result. Click 'Acc the fiducial.	ept Current Position' if the result is OK. Click 'Find and center' to find and then move to the	
Fiducials:	Phase "DrillFiducial"	2	
	DrillFiducial		
- Alignmen	nt parameters —	O Lite Educials of acianal aminet and	Find and Center
Fiducial	ls for step-and-repeat	Use outer fiducials of entire multiplied project	Accept Current Position
	Diameter Tolerance	0,3 mm	Retry
	Search Area Length	10 mm	Abort
Mir	inimum Fiducial Quality	90 %	Abort
Distance f	for Direct Acceptance	0,1 mm	
	Show Fiducial Time	0 s	
Maximum N	No. of centering Tries	3	
Manual Ce	entering If Not Found	Yes ON0	
	Ask for Confirmation	O Yes	





6

- 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 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 flex-rigid PCB is finished successfully.





# 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



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
<ul> <li>Executing the Process planning wizard</li> <li>Click the icon "Process planning Wizard" on the toolbar:</li> </ul>
Or 1. Click on Wizards > Process planning wizard ➡ The process planning wizard starts:
Process planning wizard CICUIDPO Please select the type of the process. Overview Process type Process PCBs Process 2.5D elements Next Carcel

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

1





Fig. 293:

Selecting the substrate

Fig. 292: Process planning wizard	Process planning wizard	
	How many layers is the bo	oard go
	Overview Process type	
	Number of Layers	
	Substrate Through-hole plating	



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



Select which substrate you are going to use. 6.



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. 294: Through hole plating equipment

Fig. 295: Surface finishing



Sire	uitPro		Laser & Electronics
Overview	What through-h	ole plating equipment	are you going to use?
Process type Number of Layers Substrate Through-hole plating Surface finishing			
Summary	LPKF ProConduct	LPKF MiniContac RS	LPKF Contac RS
	LPKF EasyContac		

- 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):



11. Click on [Next].





Fig. 296: Summary

Circu	iitPro	Laser & Electronics
<b>Overview</b> Process type Number of Layers Substrate	Summary Process type Process PCBs Number of layers	
Through-hole plating Surface finishing Summary	Number of Layers Double-sided Substrate FR4/FR5 Through-hole plating LPKF ProConduct Surface finishing	
	LPKF ProMask	

- 12. Verify the configuration and click on [Done].
- The Process planning wizard is finished.



## **1.2 Importing the Gerber files**



Fig. 297: Icon import

Or

- 1. Click on File > Import...
- ➡ The following dialog is displayed:



- 2. Select all required data according to the table below.
- 3. Click on [Open].

1


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

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

Fig. 299: Assigned layers

porce File Marrie	Format	Aperture/Tool List		Layer/Template		Size/Format	ОК
Tutor.BOA	GerberX 💌	Tutor.BOA	•		•	43,6 x 78,05 mm	
Tutor.BOT	GerberX 🔻	Tutor.BOT	•	BottomLayer	•	40,89 x 69,47 mm	Cancel
Tutor.SMB	GerberX 💌	Tutor.SMB	•	SolderMaskBottom	•	22,38 x 66,73 mm	Add Eile
Tutor.SMT	GerberX 🔻	Tutor.SMT	•	SolderMaskTop	-	38,9 x 72,16 mm	Add File
Tutor.SPT	GerberX 💌	Tutor.SPT	-	SolderPasteTop	-	38,63 x 68,98 mm	Remove
Tutor.TOP	GerberX 💌	Tutor.TOP	-	TopLayer	-	40,84 x 75,76 mm	
<u>202</u> 201 <u>2</u> 01 <u>9</u> 02 <u>2</u> 180	0.00	21.90		Size 4 Unit 1 Values 2 Decimal 0 Digits m.n 2	3,6 > Millin Absc	r78,05 mm meters ↓ lute ↓ leading zeros ↓ () () () () () () () () () () () () ()	



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

Note

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.



- 6. Click on [OK].
- ➡ The data are displayed in the CAM view.
- The Gerber files were imported.



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

Note



import

dialog

#### Importing the drill file 1.3



- Or
- 1. Click on File > Import....
- ➡ The following dialog is displayed:

Fig. 301: Import Import Import | File Name Format | Aperture/Tool List Layer/Template Size/Format Cancel Add File ? 🗙 Open Remove 💌 🕜 🗊 📼 • Look in: Case\_BasicCAMOpe Tutor.BOA 3 Tutor.BOT Tutor.DRL Tutor.SMB Tutor.SMT My Recent Documents 2D View Apertur B Tutor.SPT Desktop My Documents My Computer File name Tutor.DRL ~ Open ~ Files of type All files (".") My Ne Cancel 1 Ready

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

Fig. 302: Changing the layer column





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.

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

mport	File Name	Format		Aperture/Tool List		Layer/Template		Size/Format
	Tutor.DRL	Excellon	v	Tutor.DRL	×	Tutor.DRL	~	7,11 x 10,57 mm

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 where 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.

- $\rightarrow$  If the display is correct, continue with step 6.
- $\rightarrow$  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.

1





For details about the settings, please refer to the tutorial "Processing Gerber and Excellon files".

Fig. 303: Display correct

view



- 6. Click on [OK].
- ➡ The design is now displayed on screen:



The drill file was imported. ٠



### 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...:



➡ The following dialog is displayed:



- 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].

Fig. 306: Create

rectangle



The rubout area is created on your board: ⇒





If you wish to create a rectangle by a specific size, you may use the coordinate system in the rectangle window.

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.



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

Note



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:





The following dialog is displayed: ⇒



- Move the "Create circle" dialog off to the side. 2.
- 3. Left click where you want to place each fiducial hole:

Fig. 310: Example for placing fiducials





Best results are just off each corner of the board.

Note

- 4. Click on [Close].
- The fiducials were inserted.

1



### **1.6 Creating toolpaths**

The Technology Dialog is used for creating toolpaths in CircuitPro.



There are various options how to create toolpaths etc. For more detailed information please refer to the chapter "Technology Dialog" in the compendium.

1

You will pass through following steps:

- i. Selecting the material type
- ii. Selecting the insulation type
- iii. Selecting the contour routing type
- iv. Creating the toolpaths

-

#### 1. Click the icon "Technology Dialog" on the toolbar:

Va-

Fig. 311: Icon Technology Dialog

### Or

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

Clabel	ST DIGIOS		
- Global process	settings		
Material type	FR4	<ul> <li>RF application</li> </ul>	
Insulate			
		Insulation Method	
		Racin	
	$\Rightarrow$	0010	
		Description	
		Texterior with a single feasibility share of	
		Shortest processing time.	
_	1/4		
Pro Pro	ocess		
Show	Details		
- Contour Routi	ng		
		Contour Routing Method	
		Corner gap	
		3-4	
		Description	
		Contour Routing with one can in each corner	
		control houring with one gap in each content	
-	5/6		
Pro	cess		
Show	Details		
- Convert to Too	olpath		
Drills	Show Details		
Fiducials	Show Details		
_		Start	
Pockets	Show Details		
		Close	•

Fig. 312: Technology Dialog



- Selecting the material type
- 1. Select your material type in the corresponding drop down list.
- 2. If your project is a radio frequency application, activate the corresponding check box <RF application>.
- The material type has been selected.
- 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

 $\rightarrow$  The method "Basic insulation" is isolating the traced pads from the copper only.

Fig. 313: Basic insulation

	5	Basic				
	<u> </u>	Description				
		Insulation with Shortest proce	a single insulation cha ssing time.	nnel.		
Hide de Source Primary	<wiring> layers M Universal Cutter 0,2 mm</wiring>	Insulation width Pads insulation	0,2 mm 0,05 mm	¢ 0	🚖 channels	
Available tools	✓ Universal Cutter 0,2 mm     ✓ Micro Cutter 0,1 mm     End Mill (RF) 0,15 mm     End Mill (RF) 0,25 mm     End Mill (RF) 0,4 mm     End Mill 0,8 mm	Rubout Tolerance Generate optim Force insulation	<no rubout=""> 0,002 mm nized rubout C Remov n nsulation</no>	ve spikes	ncentric	×

#### b) Basic insulation, pads double

 $\rightarrow$  The method "Basic insulation, double pads" isolates the traces and double isolates the pads.

			Basic, pads do	uble		
			Description	a single insulation chann	el.	
2	/4		Short processin	n channel for pads. Ig time.		
Proce	ess					
Hide det	tails					
Source	<wiring> layers</wiring>	~	Insulation width	0,2 mm		
			Dada inculation	0.35 mm		2 A channels
Primary	Universal Cutter 0,2 mm		Pads Insulation	0,00 mm	¥.	
Primary Available tools	Universal Cutter 0,2 mm		Rubout	<no rubout=""></no>	~	Concentric
Primary Available tools	Universal Cutter 0,2 mm ✓ Universal Cutter 0,2 mm ✓ Micro Cutter 0,1 mm □ End Mill (BE) 0.15 mm	•	Pads Insulation Rubout Tolerance	<no rubout=""></no>	× ×	Concentric
Primary Available tools	Universal Cutter 0,2 mm Universal Cutter 0,2 mm Micro Cutter 0,1 mm End Mill (RF) 0,15 mm End Mill (RF) 0,25 mm		Rubout Tolerance	<no rubout=""> 0,002 mm ized rubout</no>	v v spikes	Concentric V
Primary Available tools	Vinversal Cutter 0,2 mm     Vinversal Cutter 0,2 mm     Micro Cutter 0,1 mm     End Mill (RF) 0,15 mm     End Mill (RF) 0,25 mm     End Mill (RF) 0,4 mm     End Mill 0.8 mm		Rubout Tolerance Generate optim	o,00 mm O,002 mm ized rubout	spikes	Concentric
Primary Available tools	Universal Cutter 0,2 mm Universal Cutter 0,2 mm Micro Cutter 0,1 mm End Mill (RF) 0,15 mm End Mill (RF) 0,25 mm End Mill (RF) 0,4 mm End Mill 1 mm		Rubout Tolerance Generate optim Force insulation Perform inner in	(No rubout> 0,002 mm ized rubout Remove nsulation	spikes	Concentric V





#### 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. 315: Partial	Insulate
rubout	Insulation Method
	Partial rubout
	Description
	Insulation with a single insulation channel. Removal of copper in defined areas.
	Process
	Hide details
	Source   Wiring> layers Insulation width 0,2 mm
	Primary Universal Cutter 0,2 mm 💌 Pads insulation 0,35 mm 😨 2 😨 channels
	Available tools 🗹 Universel Cutter 0,2 mm 🛆 Rubout < Rubout> layers 🗸 Concentric 🗸
	End Mill (RF) 0,15 mm Tolerance 0,002 mm
	End Mill (RF) 0,25 mm
	End Mill (GF) 0,4 mm
	V End Will 1 mm
	☑ End Mill 2 mm    ☑ Replace existing toolpath

#### d) Complete rubout

 $\rightarrow$  The method "Complete rubout" isolates the traces and pads along with removing the excess copper from the entire board.

		Insulation Metho	bd			
		Complete rubo	ut			
		Description				
		Entire removal Most precise in	of copper. sulation method.			
Proce	155					
Hide det	ails					
Source	<wiring> layers</wiring>	Insulation width	0,2 mm			
Primary	Universal Cutter 0,2 mm	Pads insulation	0,05 mm	; 0 🔹 d	hannels	
Available tools	🗹 Universal Cutter 0,2 mm 📥	Rubout	<rubout> layers</rubout>	Concentric		~
	Micro Cutter 0,1 mm End Mill (RF) 0.15 mm	Tolerance	0,002 mm			
	End Mill (RF) 0,25 mm	🗖 Generate optin	nized rubout 🛛 🔽 Remove spike	S		
	End Mill (RF) 0,25 mm	Generate optin Generate optin	nized rubout 🛛 Remove spike n	S		
	□ End Mill (RF) 0,25 mm □ End Mill (RF) 0,4 mm ☑ End Mill 0,8 mm ☑ End Mill 1 mm	Generate optin Force insulation	nized rubout 🛛 Remove spike n nsulation	5		

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.



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.

Fig. 316: Complete rubout



- 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

 $\rightarrow$  The method "Basic" routes along the outside of your board without gaps.

Fig. 317: Basic routing



#### b) Horizontal gaps

 $\rightarrow$  The method "Horizontal gaps" routes along the outside of your board with gaps on top and bottom side.



#### c) Vertical gaps

 $\rightarrow$  The method "Vertical gaps" routes along the outside of your board with gaps on the left and right side.

Fig. 318: Horizontal gaps routing





### d) Edge gaps

 $\rightarrow$  The method "Edge gaps" routes along the outside of your board leaving tabs in the center of each edge of the board.





#### e) Corner gap

 $\rightarrow$  The method "Corner gap" routes along the outside of your board leaving tabs in the corners.





#### f) Equidistant gaps

 $\rightarrow$  The method "Equidistant gaps" routes along the outside of your board leaving tabs at equal distance that you specify.

Fig. 322: Equidis- tant gaps routing	Contour 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. 323: Tool	Contour Routing
drop-down menu	O Inside Source <mechanical> layers ✓</mechanical>
	Outside Tool Contour Router 1 mm
	Tabs position All sides 🖌
	Gap width 1 mm
	Distance 50 mm 🚭
	Tolerance 0,002 mm
	Z Replace existing toolpath
	🕑 Generate start drills
	Contour Routing Method
	Edge gaps
	Description
	Contour Routing with one gap on each edge.
	4/6
	2 Process
	Hide details

4. Change the 2.0 mm contour router to the 1.0 mm contour router.



If your design is larger than 2x3 inches, the 2.0 mm Contour Router is recommended.

Note

If your design is smaller than 2x3 inches, the 1.0 mm Contour Router is recommended.



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.



- 2. Click on [Close], to close the report window.
- The toolpaths have been created.



Now your design is ready to be produced as a PCB. Continue with the steps described in the tutorial "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



CircuitPro must be running and the machine must be connected.

Note



## 2.1 Switching to Machining view

To start the board production, you have to switch to machining view first.Click on the tab "Machining view":





• You will see your board on your screen:







### 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. 327: Icon Tool magazine



Or

- 1. Click on Edit > Tool magazine...
- ➡ The following dialog is displayed:

Fig. 328: Tool magazine

<ul> <li>V Universal Cutter 0,2 mm</li> <li>Spiral Drill 0,4 mm</li> <li>Spiral Drill 0,6 mm</li> <li>Spiral Drill 1 mm</li> </ul>	The machine clam Click O to pick Click O to put t	o is empty at the moment. up the tool with the machine hea he tool to the corresponding mag	ad. gazine spot.		
Spiral Drill 2 mm	Holder     I       1     O       2     O       3     O       4     O       5     O       6     O       7     O	Tool		Tool life spent	
	Please use tool hol	der check-boxes to make these	functions available.	Drop tool	

Note

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:



- 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. 330: 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. ⇒

Baquired tools Jniversal Cutter 0,2 mm Spiral Drill 0,4 mm Spiral Drill 0,6 mm Spiral Drill 1 mm	Machine tools The machine clamp is empty at the moment. Click $\bigcirc$ to pick up the tool with the machine head. Click $\bigoplus$ to put the tool to the corresponding magazine spot.								
Spiral Orill 2 mm	Holder       Tool       Tool life spent         1       O       Universal Cutter 0,2 mm (0,00%)       ✓         2       O       Spiral Drill 0,4 mm (0,00%)       ✓       ✓         3       O       Spiral Drill 0,6 mm (0,00%)       ✓       ✓         4       O       Spiral Drill 1 mm (0,00%)       ✓       ✓         5       O       Spiral Drill 2 mm (0,00%)       ✓       ✓         6       O       Contour Router 2 mm (0,00%)       ✓       ✓         7       NONE       ✓       ✓       ✓         Please use tool holder check-boxes to make these functions available.       Check miling width.       Discard tool       Drop tool	0,00% 0,00% 0,00% 0,00% 0,00% 0,00%							



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:



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:

V - 🕀 -

Fig. 333: Icon "Process Planning wizard"



1. Click on Machining > Process all.



Process all
Start processing all or single toolpath objects.



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



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. 335: Material settings	Material Settings Application PCB Front panel/Engraving	(2.5D)			Continue
	Properties Material Type Copper Thickness [µm] Material thickness Underlay plate thickness	FR4 18,0 10 mm 2 mm			Abort
	Location Click into the machine ar milling head to the asso Use the buttons to set th rear comer of the materi Please make sure that th limiter of the machine he the tape used to fix the r	ea to move the clated position. e front left and right al. e milling depth ead does not touch material.	X: Y: Z:	Current head position 0 mm 0 mm	
	Material width Material length Surface level [mm] Material Corners [mm]	80,22 mm 107,55 mm <undefined> (153,06 / 72,41) : (23</undefined>	33,28	/ 179,96)	

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



PCB is selected as default.

Note

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 rear corner of your material:

Fig. 336: Right rear corner	Material Settings   Application   Properties  Material Type  FR4  Copper Thickness [um]  I8,0  Material thickness  I0 mm  Underlay plate thickness  Zmm  Click into the machine area to move the milling head to the associated position. Use the buttons to set the front left and right			Continue Close Abort	80000戸A中A出 20000 2000 2000 1 0 0 0 0 0 0 0 0 0 0 0		
	Imiter of the machine he the tape used to fix the r Material width Material length Surface level [mm] Material Corners [mm]	ad does not touch naterial. 217,27 mm 159,27 mm cundefined> (6,50 / 20,50) : (223	Y: Z:	223,77 mm 189,27 mm 0 mm 89,27)			2

/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":

PCB     Pcont panel/Engraving (2.50)     Properties     Material Type     F4     Coper Thickness [um]     16,0	Appleadon			Continue	
● Frent panel/Engraving (2:50)       Aboxt         ● Popperties       Aboxt         Material Type       FR4         Coper Thickness [um]       18,0.0         Material thickness       10 mm         Underlay plate thickness       2 mm         Location       It is the machine area to move the milling death to the associated position.         Use the buttons to set the front left and right rear comer of the material.       Image: Current head position.         Please makes using that the milling depth the associated position.       X:         Please makes using that the milling depth the ape used to fix the material.       Y:         12:       0 mm         Material width       217,27 mm         Surface level [mm]       curred fields         Material length       159,27 rm         Surface level [mm]       curred fields         Material Comer [mm]       (cold (20,50): (223,777 / 189,27)	PCB			Close	
Abort Properties Abort Properties Coper Thickness [µm] 15,0 0 0 Material thickness 10 mm 0 15,0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Front panel/Engraving	(2.5D)		Close	O O S Carve -
Material Type       FR4         Copper Thickness [µm]       18,0         Material Type       Image: Comper Thickness         Underlay plate thickness       10 mm         Underlay plate thickness       2 mm         Location       Image: Comper Thickness         Click into the material.       Image: Comper Thickness         Verify the buttons to set the front left and right rear comer of the material.       Image: Comper Thickness         Please make sure that the milling depth limiter of the material.       X:       223,77 mm         Y:       189,27 mm       Image: Comper Thickness         Z:       0 mm       Image: Comper Thickness         Material width       217,27 mm       Image: Comper Thickness         Surface level (mm)       condefined>         Material longth       159,277 mm         Surface level (mm)       condefined>         Material width       217,277 mm         Surface level (mm)       condefined>	Properties			Abort	1 4 3 - X 30 and -
Copper Thickness [µm] 18,0 Material thickness 10 mm Underlay plate thickness 2 mm Location	Material Type	FR4	~		
Material width 217,27 mm Material width 217,27 mm Material width 217,27 mm Surface level [mm] (c,50/20,50) : (223,77 / 189,27)	Copper Thickness [µm]	18,0	¢		
Underlay plate thickness 2 mm   Location  Location  Click into the machine area to move the milling head to the associated position. Use the buttons to set the front left and right milter of the matchine head does not touch Y: 189,27 mm Y: 189,27 mm X: 223,77 mm X: 189,27 mm X: 223,77 mm X: 189,27 mm X: 223,77 mm X: 189,27 mm X:	Material thickness	10 mm	<b>A</b>		
Location Click into the machine area to move the milling head to the associated position. Use the buttons to set the front left and right rear comer of the material. Please make sure that the milling depth Imiter of the material. Please make sure that the milling depth Imiter of the material. Y: 189,27 mm 2: 0 mm Material width 217,27 mm Surface level [mm] (c50/ 20,50) : (223,77 / 189,27)	Underlay plate thickness	2 mm	<b>*</b>		
Material width         217,27 mm           Material length         159,27 mm           Surface level [mm] <undefined>           Material Comers [mm]         (6,50 / 20,50) : (223,77 / 189,27)</undefined>	Use the buttons to set the rear corner of the materi	he front left and right ial.	Current head position		•
Material length         159,27 mm           Surface level [mm] <undefined>           Material Corners [mm]         (6,50 / 20,50) : (223,77 / 189,27)</undefined>	Use the buttons to set th rear corner of the materi Please make sure that th limiter of the machine he the tape used to fix the r	he front left and right lal. he milling depth ead does not touch material.	Current head position X: 223,77 mm Y: 189,27 mm Z: 0 mm		
Surface level [mm] <undefined> Material Comers [mm] (6,50 / 20,50) : (223,77 / 189,27)</undefined>	Use the buttons to set th rear corner of the materi Please make sure that th limiter of the machine h the tape used to fix the n Material width	ne front left and right ial. he milling depth ead does not touch material. 217,27 mm	Current head position X: 223,77 mm Y: 189,27 mm Z: 0 mm Current head position		+ -
Material Corners [mm] (6,50 / 20,50) : (223,77 / 189,27)	Use the buttons to set the rear conner of the matteri Please make sure that the limiter of the machine h the tape used to fix the r Material width Material length	e front left and right ial. he milling depth ead does not touch material. 217,27 mm 159,27 mm	Current head position X: 223,77 mm Y: 189,27 mm Z: 0 mm Current head position		· · · · · · · · · · · · · · · · · · ·
	Use the buttons to set th rear comer of the materi Please make sure that th limiter of the machine h the tape used to fix the r Material width Material length Surface level [mm]	e front left and right ial. he milling depth ead does not touch material. 217,27 mm 159,27 mm < undefined>	Current head position X: 223,77 mm Y: 189,27 mm Z: 0 mm Control of the second s		• •

The coordinates of the current head position are saved. ⇒

operations

Fig. 337: Click on button

2



Fig. 338: Front

left corner

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



➡ The machine head moves to this position.



- The coordinates of the current head position are saved and the processing area has been fit to the material.
- 4. Click on [Continue].
- The material settings were entered.



Fig. 340: Placement

#### Phase "Placement"

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

Following dialog is displayed:

Cat contar	Continue
Set terner	Close
Rotation	
Angle 0 ° 😒 🔿 📿	
Sten and Reneat	
Number of copies	
X: 1 Y: 1	
Spacing between copies [mm]	
X: 0 mm 🗢 Y: 0 mm 🗢	
Recet	

- 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).



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.

#### 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.

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

?	Confirmation of focus height					
	The focus search has finished and the machine has now moved into the calculated focus position. Please check whether the image is sharp. Click 'Yes' to accept the position, click 'No' to retry focusing or click 'Cancel' to abort the operation.					
	Yes No	Cancel				

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:

Alignmen	nt		×
Check align center of t	ment result. Click 'Aco he fiducial.	ept Current Position' if the result is OK. Click 'Find and center' to find and then move to the	
Fiducials:	Phase "DrillFiducial"		
	DrillFiducial		
- Alignmen	it parameters	O Lites fiducials of original emission only	Find and Center
Fiduciał	ls for step-and-repeat	Use outer fiducials of entire multiplied project	Accept Current Position
	Diameter Tolerance	0,3 mm 🖨	Retry
	Search Area Length	10 mm	abort
Mir	nimum Fiducial Quality	90 %	Abort
Distance f	for Direct Acceptance	0,1 mm	
	Show Fiducial Time	0 s	
Maximum N	No. of centering Tries	3	
Manual Ce			
	Ask for Confirmation	O Yes   No	J

- 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.

Fig. 341: Alignment



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.



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".



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

Tip

processes. Furthermore LPKF systems for applying solder paste, for mounting components and for soldering circuit boards are available. Do not hesitate to ask us.





# 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



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.

Tip



## 3.1 Importing the DXF file

	CircuitPro must be running.
	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. 342: Icon import	🔣 🌪 🧭 - 🚸 - 🕅 👫 💽
	Or
	1. Click on File > Import
	The following dialog is displayed:
Fig. 343: Import dialog	Import       C         Import       Format         Import       C         Import       Fee         Import       Fee         Import       C         Import       C     <

- 2. Select the file "Tutor\_dxf.dxf".
- 3. Click on [Open].

3



- ➡ The document's information is shown in the table.
- 4. Change the layer/template column to "TopLayer":

Fig. 344: Change column	Import					
	Import	File Name	Format	Aperture/Tool List	Layer/Template	Size/Format
		Tutor_dxf.dxf	Dxf format 💌	Tutor_dxf.dxf 🗾 💌	TopLayer 🔽	200 x 62 mm

5. Select in the tab "General" the increment that the DXF file was originally drawn in:

Fig. 345: Selecting the increment	General
	~~



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:



• The file was imported.



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



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. 347: Board outline selected



- 2. Press the right mouse button.
- 3. Select "Assign objects to layer".

objects to layer

4. Click on "BoardOutline":



- 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.



You can also single click on the traces. Hold the control key down to select multiple items.

Fig. 349: Highlighting the traces



- 2. Press the right mouse button.
- 3. Select "Assign objects to layer".
- 4. Click on "TopLayer":







- ➡ 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.



Fig. 351: Pane "Layers"

- Converting the objects to polygons
- 1. Single left click on the "TopLayer" in the pane "Layers":

🗋 🗙 🔼	1	٠	[]]	Hide empty								
Name	Vis	Sel	Colors	Mode		Tech		Inv	Phase	_	[Z]	^
Fiducial (4)				True Width	~	Fiducials	~		DrillFiducial	~	1	
DrillPlated (0)		☑		True Width	~	Drilling	~		DrillingPlated	~	2	
DrillUnplated (0)	⊻	V		True Width	~	Drilling	~		DrillingUnplated	~	3	
SilkScreenTop (0)	◄			True Width	~	Silk Screen	~		Undefined	~	4	
SolderPasteTop (0)				True Width	~	Solder Paste	~		Dispense	~	5	
SolderMaskTop (0)				True Width	×	Solder Mask	×		Undefined	×	6	
TopLayer (42)				True Width	~	Wiring	~		MillingTop	~	7	
TextTop (0)	◄	⊻		True Width	~	Wiring	~		MillingTextTop	~	8	
RuboutTop (0)	V			Outline	~	Rubout	~		MillingTop	~	9	
PocketTop (0)	⊻			Thin Line	~	2.5D milling top	~		MillingPocketTop	~	10	
Layer2 (0)				True Width	~	Wiring	~		MillingLayer2	~	11	
RuboutLayer2 (0)				Outline	~	Rubout	~		MillingLayer2	~	12	
BoardOutline (0)	◄			True Width	~	Mechanical	~		ContourRouting	~	13	
RuboutLayer3 (0)	V			Outline	~	Rubout	~		MillingLayer3	~	14	
Layer3 (0)	⊻			True Width	~	Wiring	~		MillingLayer3	~	15	
PocketBottom (0)	⊻	V		Thin Line	~	2.5D milling bottom	~		MillingPocketBottom	~	16	
RuboutBottom (0)	◄			Outline	~	Rubout	~		MillingBottom	~	17	
TextBottom (0)				True Width	~	Wiring	~		MillingTextBottom	~	19	
BottomLayer (0)				True Width	~	Wiring	~		MillingBottom	~	20	
SolderMaskBottom (0)	✓	◄		True Width	~	Solder Mask	~		Undefined	~	21	
SolderPasteBottom (0)				True Width	~	Solder Paste	~		Dispense	~	22	~

2. Click on the icon "Select objects on layer" in the pane "Layers":

: 🗋 🗙 🔽	T	+		-1	Hide empty	_		_	_			_	
Name	Vis	Sel	Co	lors	Mode	- Invi	Tech		Inv	Phase	_	[Z]	^
Fiducial (4)			L		Select objects of	n lay	ucials	×		DrillFiducial	~	1	
DrillPlated (0)					True Width	~	Drilling	*		DrillingPlated	~	2	
DrillUnplated (0)	☑				True Width	~	Drilling	~		DrillingUnplated	~	3	
SilkScreenTop (0)					True Width	~	Silk Screen	~		Undefined	~	4	
SolderPasteTop (0)					True Width	~	Solder Paste	~		Dispense	~	5	
SolderMaskTop (0)					True Width	~	Solder Mask	~		Undefined	~	6	
TopLayer (42)					True Width	~	Wiring	~		MillingTop	~	7	
TextTop (0)	V				True Width	~	Wiring	~		MillingTextTop	~	8	
RuboutTop (0)	•				Outline	~	Rubout	~		MillingTop	~	9	
PocketTop (0)					Thin Line	~	2.5D milling top	~		MillingPocketTop	~	10	
Layer2 (0)					True Width	~	Wiring	~		MillingLayer2	~	11	
RuboutLayer2 (0)					Outline	¥	Rubout	~		MillingLayer2	~	12	
BoardOutline (0)					True Width	~	Mechanical	~		ContourRouting	~	13	
RuboutLayer3 (0)					Outline	~	Rubout	~		MillingLayer3	~	14	
Layer3 (0)					True Width	~	Wiring	~		MillingLayer3	~	15	
PocketBottom (0)					Thin Line	~	2.5D milling bottom	~		MillingPocketBottom	~	16	
RuboutBottom (0)					Outline	~	Rubout	~		MillingBottom	~	17	-
TextBottom (0)					True Width	~	Wiring	~		MillingTextBottom	~	19	
BottomLayer (0)					True Width	~	Wiring	~		MillingBottom	~	20	
SolderMaskBottom (0)	•				True Width	~	Solder Mask	~		Undefined	~	21	
SolderPasteBottom (0)					True Width	~	Solder Paste	~		Dispense	~	22	

Fig. 352: Select objects on layer





to polygon

- 3. Press the right mouse button.
- In the context menu, select "Convert to polygon": 4.





This will combine all segments and fill the object.

The objects were converted to polygons. ٠



Fig. 354: Highlighting the drill holes

- Converting the drill holes
- 1. Highlight all drill holes:





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.



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".





# 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



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\ UseCase_BasicCAMOperations". Tip
	1. Click the icon "Import" on the toolbar:
Fig. 355: Icon import	i 💽 🕐 - 🚱 - 🖍 🐏 💽
	Or 1. Click on File > Import ◆ The following dialog is displayed:
Fig. 356: Import dialog	Import       File Name       Format       Aperture/Tool List       Layer/Template       Size/Format       OK         Cancel       Add File       Add File       Add File       Remove         Look in:       UseCase_BasicCAMOperations       Import       Remove         Unitor.BOT       Tutor.BOT       Tutor.SNT         Tutor.SNT       Tutor.SNT       Tutor.SNT         Deskop       My Documents       Tutor.TOP
	Image: Project and the provided in the provided

2. Select all files in the folder.





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. 357: Imported files

Import	File Name	Format		Aperture/Tool Li	st	Layer/Template		Size/Format	OK
		GerberX	•					43,6 x 78,05 mm	
V	Tutor.BOT	GerberX	-	Tutor.BOT	-	Tutor.BOT		40,89 x 69,47 mm	Cancel
V	Tutor.SMB	GerberX		Tutor.SMB		Tutor.SMB	-	22,38 x 66,73 mm	Add File
V	Tutor.SMT	GerberX	-	Tutor.SMT	-	Tutor.SMT	-	38,9 x 72,16 mm	Aug File
V	Tutor.SPT	GerberX	•	Tutor.SPT	-	Tutor.SPT	-	38,63 x 68,98 mm	Remove
V	Tutor.TOP	GerberX		Tutor.TOP	•	Tutor.TOP	-	40,84 x 75,76 mm	
	Tutor.DRL	Excellon		Tutor.DRL		Tutor.DRL	-	42,25 x 76,86 mm	
					-	_			
2D View	Apertures/Tools	Text View	N	vlessage View		General 0	ptio	ns	
						Size	43,6	78,05 mm	
39.02						Unit	Milli	meters v	
						Values	ADSC	sute 👻	
-						Decimal	Omit	t leading zeros 👻 👻	
- 13.01						Digits m.n	2	4 3 A	
3									
-									
-									
-									
mm	-21.80	0.00		21.80					
		and the second se							

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").
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. 358:
Unrecognised
files

Import	File Name	Format		Aperture/Tool L	st	Layer/Template		Size/Format
~	Tutor.BOA	GerberX	~	Tutor.BOA	~	Tutor.BOA	~	43,6 x 78,05 mm
~	Tutor.BOT	GerberX	~	Tutor.BOT	~	Tutor.BOT	~	40,89 x 69,47 mm
~	Tutor.SMB	GerberX	~	Tutor.SMB	~	Tutor.SMB	~	22,38 x 66,39 mm
~	Tutor.SMT	GerberX	~	Tutor.SMT	~	Tutor.SMT	~	38,9 x 71,82 mm
~	Tutor.SPT	GerberX	~	Tutor.SPT	~	Tutor.SPT	~	38,63 x 68,98 mm
~	Tutor.DRL	Excellon	~	Tutor.DRL	~	Tutor.DRL	~	42,25 x 76,86 mm
	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:

 Import	File Name	Format		Aperture/Tool List		Layer/Template		Size/Format
<b>V</b>	Tutor.BOA	GerberX	~	Tutor.BOA	~	Tutor.BOA	~	43,6 x 78,05 mm
~	Tutor.BOT	GerberX	~	Tutor.BOT	~	Tutor.BOT	~	40,89 x 69,47 mm
~	Tutor.SMB	GerberX	~	Tutor.SMB	×	Tutor.SMB	~	22,38 x 66,39 mm
~	Tutor.SMT	GerberX	~	Tutor.SMT	~	Tutor.SMT	~	38,9 x 71,82 mm
~	Tutor.SPT	GerberX	~	Tutor.SPT	~	Tutor.SPT	~	38,63 x 68,98 mm
~	Tutor.DRL	Excellon	~	Tutor.DRL	~	Tutor.DRL	~	42,25 x 76,86 mm
	Tutor.TOP	Undefined	~		~		~	
2D View	Apertures/Tools	Gerber GerberX Excellon S&M Dxf format HP-GL data LPKF_MilL [ Directif CAM	forn Drill F	nat File (LMD)		Î		

2. Click on the corresponding file format ("GerberX" in this case).

GerberX 🔽 Tutor.TOP

The information in the columns "Aperture/Tool List", "Layer/Template", and "Size/Format" is displayed automatically if you have selected the correct file format.

✓ TopLayer

Fig. 360: File details

➡ The checkmark in the "Import" column is set automatically by CircuitPro.

• The file format is selected.

Tutor.TOP

✓ 40,84 x 75,76 mm



#### Selecting the desired target layer 4.3

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. 361: "Layer/Template" column

mport	File Name	Format		Aperture/Tool List		Layer/Template	
~	Tutor.BOA	GerberX	~	Tutor.BOA	~	Tutor.BOA	Y
~	Tutor.BOT	GerberX	~	Tutor.BOT	~	BottomLayer	~
	Tutor.TOP	GerberX	~	Tutor.TOP	~	Layer2	1

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:

🛅 🗙 🔼	1	₽	[]]	Hide empty								
Name	Vis	Sel	Colors	Mode		Tech		Inv	Phase		[Z]	1
Fiducial (0)				True Width	~	Fiducials	~		DrillFiducial	~	1	
DrillPlated (0)				True Width	~	Drilling	~		DrillingPlated	~	2	
DrillUnplated (0)		✓		True Width	~	Drilling	~		DrillingUnplated	~	3	
SilkScreenTop (0)	⊻	⊻		True Width	~	Silk Screen	~		Undefined	~	4	
SolderPasteTop (0)		⊻		True Width	~	Solder Paste	~		Dispense	~	5	
SolderMaskTop (0)		☑		True Width	~	Solder Mask	~		Undefined	~	6	
TopLayer (0)		☑		True Width	~	Wiring	~		MillingTop	~	7	
TextTop (0)				True Width	~	Wiring	~		MillingTextTop	~	8	
RuboutTop (0)				Outline	~	Rubout	~		MillingTop	~	9	
PocketTop (0)		☑	Ц	Thin Line	~	2.5D milling top	~		MillingPocketTop	~	10	
BoardOutline (0)		✓		True Width	~	Mechanical	~		ContourRouting	~	12	
PocketBottom (0)			ĻĻ	Thin Line	~	2.5D milling bottom	<b>~</b>		MillingPocketBottom	×	13	
RuboutBottom (0)				Outline	~	Rubout	×		MillingBottom	~	17	
TextBottom (0)				True Width	~	Wiring	~	<u> </u>	MillingTextBottom	~	18	
SoldorMaskRotters (0)				True Width	×	Wiring Saldar Mark	×		MillingBottom	×	19	· ·
SolderMaskbottom (0)				True Width	×	Solder Mask		<u> </u>	Disease		20	
SilkScreenBottom (0)				True Width		Silk Screen			Undefined		22	
Tutor BOA (1)				True Width	•	Liokoowo			Undefined	÷	23	
Laver2 (2204)				True Width		Liekeewe			Undefined	•	24	





# 4.4 Setting/correcting size and format

Processing Gerber and

Excellon files

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.



If the imported files are GerberX files, the measurement unit is preset by default and cannot be modified.

Note

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. 363: Abnormal design size

mport								
Import	File Name	Format	1	Aperture/Tool List		Layer/Template		Size/Format
¥	Tutor.BOA	GerberX	~	Tutor.BOA	~	Tutor.BOA	~	43,6 x 78,05 mm
~	Tutor.BOT	GerberX	~	Tutor.BOT	~	Tutor.BOT	~	40,89 x 69,47 mm
<b>V</b>	Tutor.DRL	Excellon	~	Tutor.DRL	~	Tutor.DRL	~	1073,2 x 1952,22 mm

### 1. Click on the tab "General".

Import | File Name Format Aperture/Tool List Layer/Template Size/Format GerberX 👻 Tutor.BOA GerberX 💙 Tutor.BOT Excellon 👻 Tutor.DRL ✓ 43,6 x 78,05 mm Tutor.BOA ✓ Tutor.BOA Tutor.BOT ~ Tutor.BOT 🛩 40,89 x 69,47 mm Cancel Tutor.DRL ✓ Tutor.DRL ✓ 1073,2 × 1952,22 … Add File 2D View 1073,2 × 1952,22 mm inch ~ Unit Ahsolut Values ~ Omit leading zeros ~ Decimal 3 Digits m.n 1 -1 Ready

Fig. 364: 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. 365: Corrected measurement unit

Import	File Name	Format		Aperture/Tool List	t	Layer/Template		Size/Format	OK
	Tutor.BOA	GerberX	~	Tutor.BOA	~	Tutor.BOA	~	43,6 x 78,05 mm	
~	Tutor.BOT	GerberX	~	Tutor.BOT	~	Tutor.BOT	~	40,89 x 69,47 mm	Cancel
	Tutor.DRL	Excellon	~	Tutor.DRL	Y	Tutor.DRL	-	42,25 x 76,86 mm	Add Elo
									Add Hie
								T   1	Remove
20 16-00	AnothernetTeals	Tout Menu	-	Marriage Minus					
20 view	Apertures/100is	Text view		message view	_	General	_		
						Size	42	$,25 \times 76,36 \text{ mm}$	
<u>38.</u> 51						Unit	mn	n 🗸	
						Values	Ab	solute	
						Desimal	07	sit leading toros	
						Decimai	OII		
- <u>12.</u> 89 -						Digits m.n	1	3 3	
<u>12</u> .73									
<u>38</u> .35									
	1 21 12	10.00		121.12					
(INTER	-21.13	0.00							

The measurement unit has been checked and corrected.



- Processing Gerber and Excellon files
  - 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:

ct preview	Import	File Name	Format		Aperture/Tool List		Layer/Template		Size/Format	OK
		Tutor.BOA	GerberX	~	Tutor.BOA	~	Tutor.BOA	~	43,6 x 78,05 mm	
		Tutor.BOT	GerberX	~	Tutor.BOT	~	Tutor.BOT	~	40,89 x 69,47 mm	Cancel
		Tutor.DRL	Excellon	~	Tutor.DRL	~	Tutor.DRL	~	19,29 x 18,89 mm	Add File
										Remove
	2D Viev	Apertures/Tool	s Text View	1	Message View		General	_		
	100000						Size	19,	29 × 18,89 mm	
	9.96						Unit	mm	n 💌	
	-						Values	Abs	solute 🖌	
	-						Decimal	Om	it leading zeros 🖌 🖌	
	- <u>-3.6</u> 7						Digits m.n	1		
	1.1.1									
	-									
	-									
	- 8.93									
	mm	-9.73			14 9.5					

- 1. Click on the tab "General".
- 2. Change the "n" digit count from "4" to "3".
- ➡ The design's preview changes as follows:

unporc	File Name	Format		Aperture/Tool List	t	Layer/Template		Size/Format	
	Tutor.BOA	GerberX	~	Tutor.BOA	~	Tutor.BOA	~	43,6 x 78,05 mm	
~	Tutor.BOT	GerberX	*	Tutor.BOT	~	Tutor.BOT	~	40,89 x 69,47 mm	Cancel
	Tutor.DRL	Excellon	~	Tutor.DRL	~	Tutor.DRL	Y	42,25 x 76,86 mm	Add File
									Remove
-						-			
20 view	Apertures/Tools	Text view		message view	_	General	-		
						Size	42	,25 × 76,86 mm	
<u>38.5</u> 1 -						Unit	mn	n 🖌	
-						Values	Ab	solute	
-						Desimal	0	nit leading soror	
-						Decimal	O		
- 12.89						Digits m.n	1	<u> </u>	
-								~	
-									
2									
<u>12</u> .73									
-									
-									
Ē									
<u>38</u> .35									

• The number of decimal digits has been checked and corrected.

Fig. 367: Correct preview



# 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.



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. 369: 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:



The aperture's properties are modified.

4



Fig. 371: Layer

name in Gerber

file

#### Using the layer name defined in the Gerber file 4.6

The Gerber files contain the name of the layer. You can see the layer name in the "Text View" tab of the Import dialog:





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: ⇒





- Click in the category tree on the left on General > Import/Export > Formats > Gerber.
- The dialog changes as follows:

🖃 General	🖯 Import		OK	
Display	360-degree interpolation as d	ela. False	- Alter	
Measurement	Rotate AM-octagon	False		
Import / Export	Rotate square	True	Cancel	
- Import assignments	Step and repeat to flash	False		
- Export configurations	Use layer name	True	~	
⊟ Formats	Use layer polarity	True		
Gerber		False K		
- Excellon				
- LMD				
- STEP				
- DXF				
- Miscellaneous				
- CAM settings				
Machine				
	Use laver name			
	Use layer name definition from G	erber file.		

3. 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. 374: Import using option "Use layer name"

mport	File Name	Format		Aperture/Tool Lis	:t	Layer/Template		Size/Format
<b>V</b>	Tutor.BOA	GerberX	~	Tutor.BOA	~	BoardOutline	~	43,6 x 78,05 mm
<ul> <li>Image: A start of the start of</li></ul>	Tutor.BOT	GerberX	~	Tutor.BOT	~	BottomLayer	~	40,89 x 69,47 mm
~	Tutor.SMB	GerberX	~	Tutor.SMB	~	SolderMaskBottom	~	22,38 x 66,39 mm
1	Tutor.SMT	GerberX	~	Tutor.SMT	~	SolderMaskTop	~	38,9 x 71,82 mm
4	Tutor.SPT	GerberX	~	Tutor.SPT	~	SolderPasteTop	~	38,63 x 68,98 mm
<ul> <li>Image: A start of the start of</li></ul>	Tutor.TOP	GerberX	~	Tutor.TOP	~	TopLayer	~	40,84 x 75,76 mm
~	Tutor.DRL	Excellon	~	Tutor.DRL	~	Tutor.DRL	V	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. 375: "Options" tab

Import	File Name	Format	Aperture/Tool Lis	st	Layer/Template		Size/Format	OK
<b>V</b>	Tutor.BOA	GerberX	Tutor.BOA	~	Tutor.BOA	~	43,6 x 78,05 mm	
	Tutor.BOT	GerberX	<ul> <li>Tutor.BOT</li> </ul>	~	Tutor.BOT	~	40,89 x 69,47 mm	Cancel
<b>V</b>	Tutor.SMB	GerberX	Tutor.SMB	~	Tutor.SMB	~	22,38 x 66,39 mm	
	Tutor.SMT	GerberX	Tutor.SMT	~	Tutor.SMT	~	38,9 x 71,82 mm	Add Hie
	Tutor.SPT	GerberX	Tutor.SPT	~	Tutor.SPT	~	38,63 x 68,98 mm	Remove
<b>V</b>	Tutor.TOP	GerberX	Tutor.TOP	~	Tutor.TOP	~	40,84 x 75,76 mm	
<b>V</b>	Tutor.DRL	Excellon	Tutor.DRL	~	Tutor.DRL	~	42,25 x 76,86 mm	
- <u>113.01</u>					Ose lay     Rotate 5     Step an     Rotate 4     Set 360     Apply to	er pola Square d Rep AM-Oct interp	nty sat to flash lagon olation as default irber files	
<u>39</u> .02 mm	-21.80	0,00,	<b> </b> 21.80					

- 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.



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.





# List of figures

Fig. 1: Connection steps	10
Fig. 2: New document	
Fig. 3: Import	12
Fig. 4: Assigned lavers	13
	11
Fig. 6: Croate rectangle	14
Fig. 7. Destangle around the node	. 10
Fig. 7. Rectangle around the pads	15
	15
Fig. 9: Step & Repeat	17
Fig. 10: Layout in CAM view	17
Fig. 11:Create circle	18
Fig. 12:Four fiducials	18
Fig. 13: Technology Dialog	19
Fig. 14: Computation Results	20
Fig. 15: Tool magazine	21
Fig. 16: Inserting the tool	21
Fig. 17: Tools in the tool magazine	22
Fig. 18: Machining > Process all	23
Fig. 19: Material settings	24
Fig. 20: Right rear corner	25
Fig. 21: Click on button	25
Fig. 22: Lower left corner	26
Fig. 23: Defined processing area	26
Fig. 24: Placement	27
Fig. 25: Alignment	29
Fig. 26: Connection steps	32
Fig. 27: New document	33
Fig. 28: Import	34
Fig. 29: Assign layer	35
Fig. 30: CAM view	36
Fig. 31: Dispense	37
Fig. 32: Machining view with solder paste paths	
Fig. 33: Technology Dialog	
Fig. 34: Wizards > Dispense preparation wizard	40
Fig. 35: Prenare dispenser	40
Fig. 36: Assembling the dispenser	40 41
Fig. 37: Enter the thickness	– /1
Fig. 38: Desition of auxiliary board	+ 1
Fig. 20: Head touching the paper	42
Fig. 39. Flead touching the paper	43
Fig. 40. Dispense icon	43
Fig. 42: Alianment	44
Fig. 42: Augument	44
Fig. 43: Dispense preparation finished	45
Fig. 44: vvizards > Dispense process wizard	46
Fig. 45: Mount board	46



Fig. 46: Enter the thickness	47
Fig. 47: Select drill hole	48
Fig. 48: Message "Select drill hole"	48
Fig. 49: Alignment	49
Fig. 50: Dispense icon	50
Fig. 51: Process dispensing	51
Fig. 52: Connection steps	54
Fig. 53: New document	55
Fig. 54: Material properties	55
Fig. 55: CAM view	56
Fig. 56: Create rectangle	57
Fig. 57: Rectangle in CAM view	57
Fig. 58: Second rectangle	58
Fig. 59: Create circle	59
Fig. 60: Circle in CAM view	59
Fig. 61: Step Repeat	60
Fig. 62: Multiplying the circle in X direction	60
Fig. 63: Select all objects	62
Fig. 64: Context menu	62
Fig. 65: Convert into 2.5D	63
Fig. 66: Properties of the 2.5D object	63
Fig. 67: Create 2.5D milling	65
Fig. 68: CAM view	66
Fig. 69: 3D view	66
Fig. 70: Tool magazine	67
Fig. 71: Inserting the tool	67
Fig. 72: Assigning the tools in the dialog	68
Fig. 73: Assigned tools	68
Fig. 74: Assigning additional tools	69
Fig. 75: Tools in the tool magazine	69
Fig. 76: Machining > Process all	70
Fig. 77: Material placement	71
Fig. 78: Pane "Processing"	71
Fig. 79: Autofocus	72
Fig. 80: Cross hair of camera	72
Fig. 81: Message for second position	72
Fig. 82: Move to measuring position	73
Fig. 83: Material surface level defined	74
Fig. 84: Dismount the depth limiter	74
Fig. 85: Pneumatic Depth limiter	75
Fig. 86: Fasten depth limiter	75
Fig. 87: Mechanical depth limiter	76
Fig. 88: Connection steps	78
Fig. 89: New document	79
Fig. 90: Material properties	79
Fig. 91: CAM view	80
Fig. 92: Importing the STEP file	81

Fig. 9	93: 3D view	82
Fig. 9	94: Transformation	82
Fig. 9	95: Rotate [auto]	83
Fig. 9	96: 3D view rotatet around Z axis	83
Fig. 9	97: Move in X and Y direction	84
Fig. 9	98: Move object in Z direction	84
Fig. 9	99: Create 2.5D milling	85
Fig.	100: 3D view	86
Fig.	101: Creating circles	87
Fig.	102: CAM view with fiducials	88
Fig.	103: Technology Dialog	89
Fig.	104: Tool magazine	90
Fig.	105: Inserting the tool	90
Fig.	106: Tools in the tool magazine	91
Fig.	107: Machining > Process all	92
Fig.	108: Material placement	93
Fig.	109: Pane "Processing"	93
Fig.	110: Autofocus	94
Fig.	111: Cross hair of camera	94
Fig.	112: Message for second position	95
Fig.	113: Move to measuring position	95
Fig.	114: Material surface level defined	96
Fig.	115: Dismount the depth limiter	96
Fig.	116: Pneumatic Depth limiter	97
Fig.	117: Fasten depth limiter	97
Fig.	118: Mechanical depth limiter	98
Fig.	119: Alignment	99
Fig.	120: Centering the fiducial1	00
Fig.	121: Connection steps1	02
Fig.		03
Fig.	123: CAM view - multi-layer	03
Fig.	124: Import	05
Fig.	125: Assigned layers	06
Fig.	126: CAM view	06
Fig.	127: Create Rectangle1	07
Fig.	128: Rubout Top	07
Fig.	129: Rubout Bottom	08
Fig.	130: Step & Repeat	09
Fig.	131: Design in the CAM view	10
Fig.	132: Technology Dialog1	11
Fig.	133: CAM view	12
Fig.	134: Tool magazine	13
Fig.	135: Inserting a tool	13
Fig.	<b>v</b>	
<u> </u>	136: Loaded tool holder1	14
Fia. '	136: Loaded tool holder	14 15
Fig. <sup>2</sup> Fig. <sup>2</sup>	136: Loaded tool holder	14 15 16
Fig. <sup>2</sup> Fig. <sup>2</sup> Fig. <sup>2</sup>	136: Loaded tool holder	14 15 16 17



Fig. 140: Click on the icon	117
Fig. 141: Defining the front left corner	117
Fig. 142: Defined processing area	118
Fig. 143: Placement	118
Fig. 144: Alignment	120
Fig. 145: Connection steps	128
Fig. 146: New document	129
Fig. 147: Create rectangle	130
Fig. 148: Rectangle	131
Fig. 149: Pane "Layers"	132
Fig. 150: Mode "Thin Line"	132
Fig. 151: Create rectangle	133
Fig. 152: Rectangles in the CAM view	133
Fig. 153: Create circle	135
Fig. 154: Created circle	135
Fig. 155: Multiplied circles	
Fig. 156: Another series of drill holes	
Fig. 157: Drill holes	
Fig. 158: Create Polygon	
Fig. 159: Polygon	
Fig. 160: Circular mark	
Fig. 161: Circular marks	
Fig. 162: Define text	
Fig. 163: Text in CAM view	
Fig. 164: Holes in CAM view	
Fig. 165: Create closed path	
Fig. 166: Open path	
Fig. 167: Cutouts for the fan	
Fig. 168: Pane "Properties"	
Fig. 169:Line width changed	
Fig. 170: Create Circle	
Fig. 171: Created circle	
Fig. 172: Created circles	
Fig. 173: Pane "Layers"	150
Fig. 174: Technology Dialog	
Fig. 175: Created toolpaths	
Fig. 176: Tool magazine	153
Fig. 177: Inserting a tool	153
Fig. 178: Loaded tool holder	
Fig. 179: Machining > Process all	155
Fig. 180: Material settings	156
Fig. 181: Defining the right rear corner	
Fig. 182: Click on the icon	157
Fig. 183: Defining the front left corner	158
Fig. 184: Defined processing area	
Fig. 185: Placement	
Fig. 186: Connection steps	

Fig. 187: New document	163
Fig. 188: Import	164
Fig. 189: Assign layers	165
Fig. 190: CAM view	165
Fig. 191: Identical proportion	166
Fig. 192: Transformation >Scale	167
Fig. 193: Dimensions changed	167
Fig. 194: "Layers" pane	168
Fig. 195: Technology Dialog	168
Fig. 196: Insulation details	169
Fig. 197: Deactivating functions	170
Fig. 198: Tool magazine	172
Fig. 199: Insert tool	172
Fig. 200: Loaded tool holder	173
Fig. 201: Machining > Process all	174
Fig. 202: Material Settings	175
Fig. 203: Right rear corner	176
Fig. 204: Click on button	176
Fig. 205: Front left corner	177
Fig. 206: Click on the button	177
Fig. 207: Placement	178
Fig. 208: Connection steps	182
Fig. 209: New document	183
Fig. 210: CAM view	184
Fig. 211: Phases	185
Fig. 212: Moving fiducials	185
Fig. 213: Import	187
Fig. 214: CAM view	188
Fig. 215: "Toolpath" pane	189
Fig. 216: "Lavers" pane	190
Fig. 217: Phase selection list	190
Fig. 218: Technology Dialog	193
Fig. 219: Insulation details	194
Fig. 220: Fiducials - details	194
Fig. 221: Deactivate functions	195
Fig. 222: Tool magazine	196
Fig. 223: Inserting a tool	196
Fig. 224: Loaded tool holder	197
Fig. 225: Machining > Process all	198
Fig. 226: Material Settings	199
Fig. 227: Defining the right rear corner	200
Fig. 228: Click on the icon	
Fig. 229: Defining the front left corner	
Fig. 200. Defined processing area	200
FIG. Z3U. Deuneo processino area	200
Fig. 230. Defined processing area	200 201 201
Fig. 230: Defined processing area Fig. 231: Placement Fig. 232: Technology Dialog	200 201 201 204
Fig. 230: Defined processing area Fig. 231: Placement Fig. 232: Technology Dialog Fig. 233: Contour routing details	200 201 201 204 204



Fig. 234: Drills details	205
Fig. 235: Deactivate functions	205
Fig. 236: Tool magazine	206
Fig. 237: Inserting a tool	206
Fig. 238: Loaded tool holder	207
Fig. 239: Machining > Process all	208
Fig. 240: Material Settings	209
Fig. 241: Defining the right rear corner	210
Fig. 242: Click on the icon	210
Fig. 243: Defining the front left corner	210
Fig. 244: Defined processing area	211
Fig. 245: Selecting a cutout around a fiducial	213
Fig. 246: Properties of the cutout	213
Fig. 247: Aperture lists	214
Fig. 248: Aperture size	214
Fig. 249: Modifying the aperture size	215
Fig. 250: Technology Dialog	216
Fig. 251: Contour routing details	217
Fig. 252: Drills details	217
Fig. 253: Deactivate functions	218
Fig. 254: Tool magazine	219
Fig. 255: Inserting a tool	219
Fig. 256: Loaded tool holder	220
Fig. 257: Machining > Process all	221
Fig. 258: Mounting the material	221
Fig. 259: Material Settings	222
Fig. 260: Defining the right rear corner	223
Fig. 261: Click on the icon	223
Fig. 262: Defining the front left corner	223
Fig. 263: Defined processing area	224
Fig. 264: Creating a new phase	226
Fig. 265: Deleting toolpaths	228
Fig. 266: Technology Dialog	229
Fig. 267: Contour routing - details	230
Fig. 268: Fiducials - details	230
Fig. 269: Deactivating functions	231
Fig. 270: CAM view	231
Fig. 271: Layer CuttingLaminat	232
Fig. 272: Details contour routing	232
Fig. 273: CAM view	233
Fig. 274: Edit breakout tabs	234
Fig. 275: Breakout tab position zoomed in	234
Fig. 276: Breakout tab position moved	234
Fig. 277: Circle moved	235
Fig. 278: Left breakout tab	235
Fig. 279: New breakout tab positions	235
Fig. 280: Tool magazine	236

Fig.	281:	Inserting a tool	236
Fig.	282:	Loaded tool holder	237
Fig.	283:	Machining > Process all	238
Fig.	284:	Material Settings	239
Fig.	285:	Defining the right rear corner	240
Fig.	286:	Click on the icon	240
Fig.	287:	Defining the front left corner	240
Fig.	288:	Defined material area	241
Fig.	289:	Alignment	242
Fig.	290:	Icon Process Planning Wizard	246
Fig.	291:	Process planning wizard	246
Fig.	292:	Process planning wizard	247
Fig.	293:	Selecting the substrate	247
Fig.	294:	Through hole plating equipment	248
Fig.	295:	Surface finishing	248
Fig.	296:	Summary	249
Fig.	297:	Icon import	250
Fig.	299:	Assigned layers	251
Fig.	300:	Icon import	253
Fig.	301:	Import dialog	253
Fig.	302:	Changing the layer column	254
Fig.	303:	Display correct	255
Fig.	304:	Design displayed in CAM view	255
Fig.	311:	Icon Technology Dialog	260
Fig.	312:	Technology Dialog	260
Fig.	313:	Basic insulation	261
Fig.	314:	Basic insulation, pads double	261
Fig.	315:	Partial rubout	262
Fig.	316:	Complete rubout	262
Fig.	317:	Basic routing	263
Fig.	318:	Horizontal gaps routing	263
Fig.	319:	Vertical gaps routing	264
Fig.	320:	Edge gaps routing	264
Fig.	321:	Corner gap routing	264
Fig.	322:	Equidis-tant gaps routing	265
Fig.	323:	Tool drop-down menu	265
Fig.	324:	Computation Results	266
Fig.	325:	Tab "Machining view"	268
Fig.	326:	Machining view	268
Fig.	327:	Icon Tool magazine	269
Fig.	328:	Tool magazine	269
Fig.	329:	Tool list	270
Fig.	330:	Insert tool into holder	270
Fig.	331:	Tool magazine	271
Fig.	332:	Colors of the tool holders	271
Fig.	333:	Icon "Process Planning wizard"	272
Fig.	334:	Machining > Process all	272
-			



Fig. 335:	Material settings	73
Fig. 336:	Right rear corner	74
Fig. 337:	Click on button	74
Fig. 338:	Front left corner	75
Fig. 339:	Defined processing area	75
Fig. 340:	Placement	76
Fig. 341:	Alignment	78
Fig. 342:	Icon import	82
Fig. 343:	Import dialog28	82
Fig. 344:	Change column	83
Fig. 345:	Selecting the increment	83
Fig. 346:	CAM view	84
Fig. 355:	Icon import29	94
Fig. 356:	Import dialog29	94
Fig. 357:	Imported files	95
Fig. 358:	Unrecognised files	96
Fig. 359:	Selecting the file format manually	96
Fig. 360:	File details	96
Fig. 361:	"Layer/Template" column	97
Fig. 362:	"Layers" pane	97
Fig. 363:	Abnormal design size	98
Fig. 364:	Wrong measurement unit	98
Fig. 365:	Corrected measurement unit	99
Fig. 366:	Incorrect preview	00
Fig. 367:	Correct preview	00
Fig. 368:	Aperture properties	<b>)</b> 1
Fig. 369:	Available aperture shapes	02
Fig. 370:	Modified aperture shape and size	02
Fig. 371:	Layer name in Gerber file	03
Fig. 372:	Extras > Options	03
Fig. 373:	Extras > Options	04
Fig. 374:	Import using option "Use layer name"	04
Fig. 375:	"Options" tab	05