ProtoLaser U4/S4/R4

How-to guides



Order code: 10073032 Version: 4.0

Publisher	LPKF Laser & Electronics d.o.o. Polica 33 4202 Naklo Slovenia	
		+386 592 08 800 +386 592 08 820
	Email:	info@lpkf.com
Date of issue	22.12.2021	
Copyright	© 2021 LPKF AG	
	This document and its contents in whole and in parts are subject to copyright. The reproduction, translation or duplication of the contents as photocopy or any digital form requires written permission of LPKF Laser & Electronics AG.	
	English original document	

General information

This document contains all information for the intended use of the system/product delivered. This document is intended for persons with basic knowledge of installation and operation of software-controlled systems. General knowledge of operational safety as well as basic knowledge of using PCs running Microsoft Windows[®] and basic knowledge of your LPKF system software are required.



When processing the how-to examples, carefully note the safety instructions from the applicable user manual of your system!

Validity

This document corresponds to the technical state at the time of publication. LPKF Laser & Electronics AG (abbreviated to **LPKF** in the following) reserves the right to make changes in respect to the content of this document. The figures in this document serve as basic understanding and can differ from the actual state of the system.

Structure of warning messages and safety notes

The safety notes and warning messages in this document identify hazards and risks and they are created in accordance with ANSI Z535.6-2011 and the standards series ISO 3864.

The warning messages are structured as follows:

- Warning sign (only for injuries)
- Signal word indicating the hazard class
- Type and source of the hazard
- Consequences of non-observance
- Measures to avoid the hazard

+ SIGNAL WORD

Type and source of the hazard!

Consequences of non-observance.

- Measures to avoid the hazard.
- Further measure(s) to avoid the hazard.

Warning messages can also be embedded in the format of the surrounding text in order to avoid a *visual disruption* in a sequence. In this case, they are distinguished as follows:

Type and source of the hazard!

Consequences of non-observance.

Measure(s) to avoid the hazard.

Warning messages are classified in hazard classes represented by the signal word. In the following, the warning messages are described in accordance to their hazard classes:

Type and source of the hazard!

This warning message indicates a hazard of high risk that causes death or serious injury if not avoided.

• Measures to avoid the hazard.

Type and source of the hazard!

This warning message indicates a hazard of medium risk that can cause death or serious injury if not avoided.

Measures to avoid the hazard.

Type and source of the hazard!

This warning message indicates a hazard of low risk that can cause minor or moderate injury if not avoided.

Measures to avoid the hazard.

NOTICE

Type and source of the hazard!

This warning message indicates a hazard that can lead to possible property damage.

Measures to avoid the hazard.

Text styles

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

Attribute	Function	
italic	highlights elements of the user interface and of control elements of the system	
bold	highlights important information and keyboard input	
Courier New	highlights file paths	
[]	highlights elements of buttons on software user interfaces	
key	highlights keys of the keyboard	

Tasks or procedures that are described in steps are compiled to sequences in this document. A sequence consists of at least three components: objective, step, and result.

Component	Description	
	Indication of an objective. The sequence starts here.	
1. 2. 3.	Indication of a sorted list of steps. The specified order must be observed.	
	Indication of an intermediate result that is followed by further steps or the result.	
	Indication of the result. The sequence is finished.	
	Indication of a single step.	

Additional information

The following symbols are used to indicate additional information:

This note indicates especially useful information.

Advanced information

This advanced information indicates special knowledge.

Registered Trademarks

Product and brand names are trademarks of LPKF Laser & Electronics AG, registered among others at the US Patent and Trademark Office: LPKF® and the company logo, # 2,385,062 and # 2,374,780; SolarQuipment®, # 3,494,986; ProConduct®, # 3,219,251; Allegro®, # 3,514,950.

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.

Issued patents, if applicable, are listed in the appendix.

Customer service

For technical information contact our LPKF Service:

Address	LPKF Laser & Electronics AG Service & Support Rapid Prototyping Osteriede 7 30827 Garbsen Germany	
Phone	+ 49 5131 7095-1333	
Fax	+ 49 5131 7095-90	
Email	support.rp@lpkf.com	
Internet	http://www.lpkf.com/en	

In our continuous effort to improve our documentation we are asking you to give us your feedback if you notice any discrepancy when working with the system/product, or if you have any comments or suggestions for improvement.

At the moment of packaging, the system/product has been equipped with the latest software version and with the software and hardware documentation currently valid. By now, new versions of the documentation as well as new software versions might be available.

For all the latest news and updates visit the support area of our homepage: www.lpkf.com/en/support-services.

Contents

Intr	ntroduction8				
1	Basics				
	1.1 1.2 1.3	Project placement	5		
		1.3.1Material thickness measurement settings	0 2 3		
		1.3.4Measuring the material thickness manually (at a single point) 251.3.5Referencing the material thickness measurement sensor			
	1.4 1.5 1.6 1.7	Vorking with mixed PCB layouts in CircuitPro PL	9 8		
		1.7.1Delamination method521.7.2Hatching method54			
	1.8	Design guidelines for the production of multi-layer PCBs	6		
		1.8.1 Structures of multi-layer PCBs 56 1.8.2 Required files/layers 57			
		1.8.3 Through holes and via sizes before plating			
		1.8.4 Annular rings			
		1.8.5 Inner layers fill 59 1.8.6 Staggered vias 60			
		1.8.7 Stacked vias			
2	Prod	cing double-sided PCBs63	3		
	2.1 2.2	How to produce a double-sided PCB without through-hole plating			
	2.3	blating			
	2.4				
3	Prod	Producing multi-layer PCBs			
	3.1	low to produce a multi-layer PCB with galvanic through-hole plating by using a laser system only			
	3.2 3.3	How to produce a multi-layer PCB with galvanic through-hole plating 140 How to produce a multi-layer PCB with non-galvanic through-hole Dating	0		
	3.4	low to produce a multi-layer PCB with blind vias and buried vias 165	5		
4					
	4.1 4.2 4.3 4.4	Producing a polyimide stencil 188 Producing a flex-rigid PCB 193 Structuring the solder mask with the laser system 224 Producing an RF PCB 230	3 4		
5	Proc	ssing files	8		
	5.1 5.2	Processing DXF files in CircuitPro PL			
6	Арр	1dix	6		
	6.1 6.2	ist of figures			

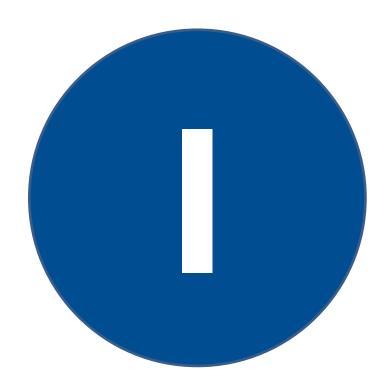
Introduction

This document describes the most common applications for the production of doublesided and multi-layer PCBs with the LPKF ProtoLaser U4/S4/R4. The document also provides numerous useful tips that facilitate your work with the system and the applicable system software.

The first how-to examples are explained in detail to facilitate your start. Further on in this document, recurrent actions are described only briefly. The document contains corresponding references to precedent chapters so that you have access to more detailed information at any time.

Only certain features may be available after the software installation depending on whether you use the license level Basic or the license level Advanced. All how-to guides require the license level Advanced; the user level *Advanced user* has to be selected in the options.





1 Basics

This chapter describes some basic steps in CircuitPro PL. You learn how to set processing data correctly in the software after having placed the processing material in the system. Furthermore, the fiducial recognition is described in detail. You perform these steps in almost each how-to example. Each step contains corresponding references to these chapters.

This chapter also describes working with mixed PCB layouts, measuring the material thickness and testing the tool settings. Moreover, this chapter describes the multi-layer process, as well as specific design guidelines for the production of multi-layer PCBs with blind vias and buried vias. Observe these information, since they provide a base for a successful production of multi-layer PCBs.

1.1 **Project placement**

This chapter describes how to place your project easily in CircuitPro PL. The project can be placed in two different ways:

- Matching the fiducial positions.
- Determining the processing area.

Both procedures are described in the following. Use one of them.

Ensure that the following prerequisites are fulfilled before performing the described tasks:

Prerequisites

- The user guidance step *Processing* is active.
- The dialog *Placement* is displayed.

The following table provides tips for positioning and fiducial recognition:

Тір	Description	Figure
Pilot laser	The pilot laser is used for positioning. The position of the pilot laser can only be changed by moving the processing table. Move the processing table so the position of the pilot laser and the fiducial on the material match. Click on \star to switch the pilot laser on/off.	
Zooming in	Zoom in the processing area by using the mouse wheel for more precise positioning.	

Тір	Description	Figure
Illumination settings	Depending on the material surface you may need to adjust the image until a proper contrast and lighting is achieved. Thus a bright surface and a dark fiducial have to be set. Move the sliders in the tab <i>Image</i> <i>settings</i> . (<i>Image settings</i> are not available for the user level <i>Operator</i> .)	Navigation Image settings Red light 75.0 % Backlight 0.0 % White light 0.0 % Brightness 26.0 % Exposure 12.0 % Gain(master) 18.0 %
Movement control	 Control the movement as follows: Use the arrow buttons in the tab <i>Navigation</i> for movement control. If necessary, adjust the <i>Step size</i> for more precise control. Double-click on the camera image. 	Navigation Image settings Step size 0.500 mm Step size 0.500 mm x 165.000 mm y 310.000 mm z 5.464 mm
Autofocus	If the camera image is not sharp, click on A ^F in the tab <i>Navigation</i> to start the autofocus procedure. The message <i>Executing autofocus</i> is displayed.	Executing autofocus – 🗆 🗙
Removing drilling debris	If a fiducial recognition problem occurs, check the fiducial holes for drilling debris.	
Polishing fiducial area	If the surface surrounding the fiducial is of very poor quality, use a fine sanding paper to polish the fiducial area.	

Table 1:	Tips for positioning and fiducial recognition
----------	---

Matching the fiducial position

By performing this procedure before processing starts you **save time**, since the *Spiral search* will be avoided.

i

Icon explanation in the user guidance step Processing:

- Red crosshairs (-----): is the current position of the processing head
- Purple circle (): displays a fiducial
- 1. Move the dialog *Placement* to get a better overview.
- 2. Click on the tab Processing data.
- 3. Double-click on the processing area.
- □ The processing table starts moving and the red crosshairs are displayed at the point of your double click. The pilot laser is switched on automatically.

i

П

- The position of the pilot laser beam on the material represents the position of the processing head.
- 4. Use the arrow buttons for movement control.
- 5. If necessary, adjust the Step size for more precise control.

Use the tips for positioning that are described in Table 1.

6. Move the processing table so the beam of the pilot laser matches a fiducial.

The pilot laser is switched off. One of the fiducials is displayed:

7. Switch to the view Camera.

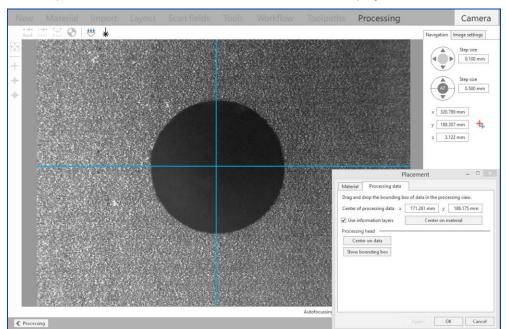


Fig. 1: Fiducial in the view Camera

The processing head in the system is now positioned directly above one of the fiducials on the material.



The real image in the view *Camera* needs to be reflected in the user guidance step *Processing* as well.

8. Switch to the user guidance step Processing.

The position of the layout is displayed as follows:

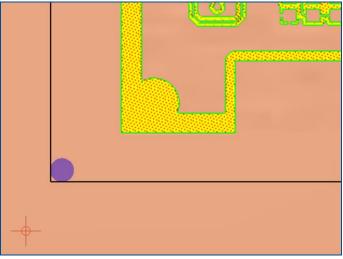
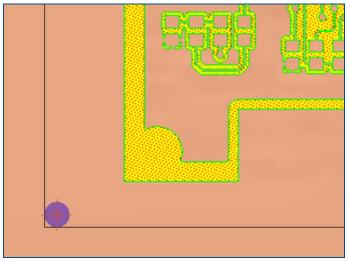


Fig. 2: Inaccurate position of the layout (example image)

The **position** of the **red crosshairs** represents not only the position of the processing head but also the **actual position of the fiducial on the material**. The entire layout needs to be moved so that the fiducial (\bigcirc) and the red crosshairs (\diamond) overlap.

- 9. Move the mouse cursor over the layout.
- \Box The mouse cursor turns into the hand symbol (\bigcirc).
- 10. Move the layout with Drag & Drop to match the location of the fiducial and the red crosshairs:





- □ The red crosshairs are now aligned to the fiducial.
- 11. Click on [OK] in the dialog Placement.
- □ The fiducial recognition is initiated (see chapter 1.2).
- The fiducial position has been matched.

Determining the processing area

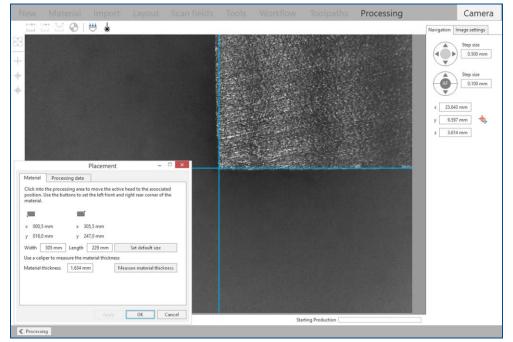
This procedure is an **alternative** to the step *Matching the fiducial position* described above. The aim of this procedure is to precisely match the base material position and base material size to the processing area used by the software CircuitPro PL.

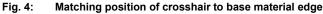
This method is most suitable for working with large designs. This is the case when producing multi-layer PCBs.

- 1. Move the dialog *Placement* to get a better overview.
- 2. Double-click on the processing area.
- □ The processing table starts moving and the red crosshairs are displayed at the point of your double click. The pilot laser is switched on automatically.
- 3. Use the arrow buttons for movement control.
- 4. If necessary, adjust the Step size for more precise control.

Use the tips for positioning that are described in Table 1.

- 5. Switch to the view Camera.
- □ The pilot laser is switched off.
- 6. Move the processing table until the left front corner of the material is displayed.





- 7. Confirm the position by clicking on _____ in the dialog Placement.
- 8. Switch to the user guidance step *Processing*.
- 9. Repeat the steps 2 to 6 for the right rear corner.
- 10. Confirm the position by clicking on **m** in the dialog *Placement*.
- 11. Click on the tab *Processing data* in the dialog *Placement*.
- 12. Click on [Center on material].

- The processing data are now precisely aligned and in the center of the base material.
- 13. Click on [OK].
- □ The crosshairs position to the base material edge has been matched and the fiducial recognition is initiated (see chapter 1.2).
- The processing area has been determined.

1.2 Recognizing fiducials

This chapter describes how to recognize fiducials in CircuitPro PL, especially such with low quality. This procedure follows the project placement procedures described in the previous chapter.

Recognizing fiducials

The camera moves to the position of the first fiducial and determines its exact position. The recognized fiducial and the following dialog are displayed:

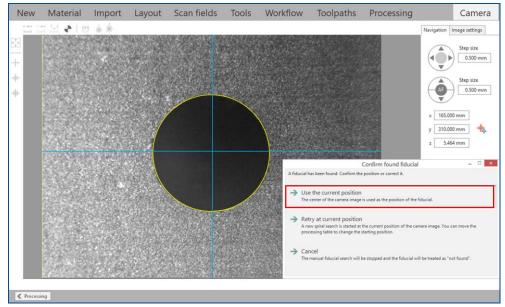


Fig. 5: Recognized fiducial

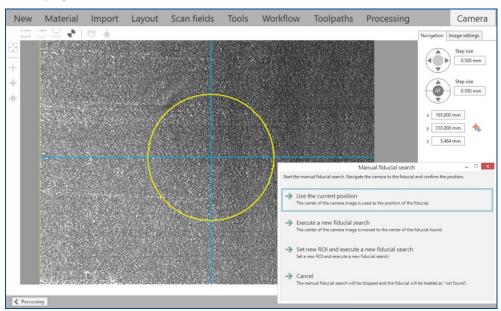
- 1. Click on Use the current position.
- The position of the fiducial has been confirmed. The camera proceeds to read the other fiducials.

If the fiducial is not displayed in the view *Camera*, a spiral search is performed to locate the fiducial and the following message is displayed:

	Spiral search
0	The spiral search is running. Click on the button to stop the spiral search.

Fig. 6: Message Spiral search active

To determine the size of the spiral search area, click on *Processing > Alignment settings…* and set the value of the *Spiral search area*.

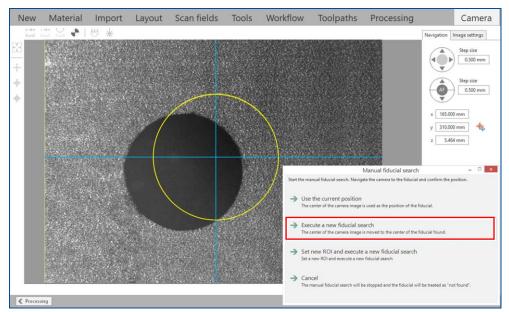


□ If the fiducial is not recognized inside the spiral search area, the view *Camera* is displayed as follows:

Fig. 7: Manual fiducial search

- 2. Move the processing table until the fiducial is visible by performing one of the following steps:
 - Use the arrow buttons in the pane Navigation.
 - Use double- clicks in the camera image.
- 3. If necessary, adjust the Step size for more precise movement control.

Use the tips for positioning that are described in Table 1.



□ The fiducial approximately matches the yellow circle and is displayed as follows:

Fig. 8: Visible fiducial

4. Click on Execute a new fiducial search.

i

□ The camera moves to the position of the fiducial and determines its exact position. The recognized fiducial is displayed:

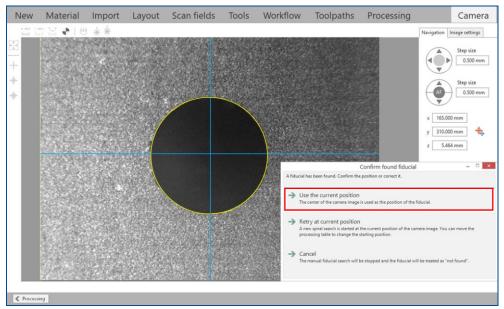


Fig. 9: Recognized fiducial

5. Click on Use the current position.

- □ The position of the fiducial has been confirmed. The camera proceeds to read the other fiducials. The fiducial recognition is finished after all fiducials have been read successfully.
- The fiducials have been recognized.

Alignment settings	×
ROI configuration	
Width (mm) 6.4 mm Height (mm) 4.8 mm	
Offset X (mm) 0 mm Offset Y (mm) 0 mm	
]
Change ROI by mouse Change ROI border color	
]
Fiducial criteria	
Diameter 1.5 mm Tolerance 20 %	
Alignment settings	
✓ Enable spiral search Spiral search area	
✓ Always center fiducials for detection Settle time 200 ms	

In some cases, the fiducial quality is too poor for the camera to find the fiducial.

Recognizing a fiducial of poor quality

This is an example of an unidentified fiducial due to poor quality:

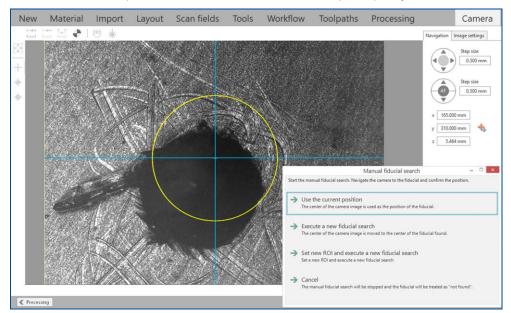


Fig. 10: Fiducial of poor quality

1. Manually move to the **exact** fiducial position (yellow circle **exactly** matches the fiducial position).

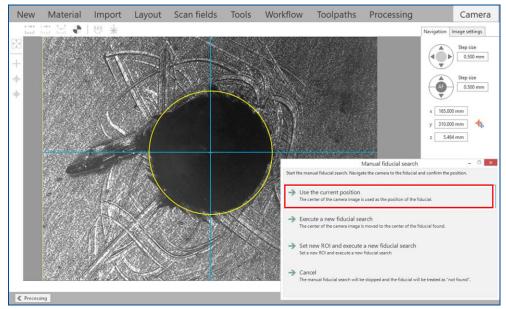


Fig. 11: Matched position of poor quality fiducial

- 2. Click on Use the current position.
- □ The system saves the fiducial position.
- A fiducial of poor quality has been recognized.

1.3 Measuring the material thickness

This chapter describes the material thickness measurement in CircuitPro PL.

1.3.1 Material thickness measurement settings

The dialog *Material thickness measurement settings* contains functions for measuring the material thickness and for referencing the material thickness measurement sensor.

To open the dialog, click on Processing > Material thickness measurement setting.

N	laterial thicknes	s measurement se	ettings – 🗆 💌
Manual measuren	nent can be started v	ia the "Placement" men	u.
Perform thickn	ess measurement du	ring process	
Material thickness	tolerance 100 µ	m	
Min hole distance	2 m	m	
Measurement stra	tegy Measur	e three points 🛛 👻	
Referencing —			
Reference points	х	Y	Z
Left rear	30.50 mm	217.00 mm	
Right rear	275.50 mm	217.00 mm	
Center	153.00 mm	132.50 mm	
Front left	30.50 mm	48.00 mm	
Front right	275.50 mm	48.00 mm	
Material thickness	0 mm		
	Old value	New value	Difference
Reference	8.04 mm		
	Old value 8.04 mm	New value rm referencing	e interente
			OK Cancel

Fig. 12: Dialog Material thickness measurement settings (default settings)

Button/Element	Description	
Perform thickness measurement during process	Activates the material thickness measurement during processing.	
Material thickness tolerance	Specifies the permissible measurement deviation from the specified material thickness. If the measured material thickness exceeds the specified tolerance, a dialog for further actions is displayed (see chapter 1.3.3).	
Min hole distance	Specifies the minimum distance from the hole to the point of measurement. This ensures that the material thickness measurement is not executed in a hole or a pocket on the material. The toolpaths for the holes must be calculated in the layout.	
Measurement strategy	Sets the measurement strategy (see chapter 1.3.2).	
Referencing	Displays the coordinates of the reference points.	
Material thickness	Specifies the material thickness that is used for the referencing process (see chapter 1.3.5).	
Reference	Displays the average referencing values (<i>Old value</i> and <i>New value</i>). The <i>Difference</i> is displayed for orientation.	
Perform referencing	Starts the referencing of the material thickness measurement sensor. Referencing should always be carried out after cleaning or replacing the sinter plate and after transport to ensure the highest possible precision of the system.	

Table 2: Functions in the dialog Material thickness measurement settings

1.3.2 Measurement strategies

If the check box *Perform thickness measurement during process* is activated, the system will perform the material thickness measurement automatically during processing. The system uses three different measurement strategies that can be selected from the drop-down list:

Ma	aterial th	ickness m	neasurement setting	gs –	×
Manual measureme	nt can be s	tarted via th	e "Placement" menu.		
Perform thicknes	s measurer	ment during	process		
Material thickness to	olerance	100 µm			
Min hole distance		2 mm			
Measurement strate	gy	Measure thr	ee points 💙		
Referencing		Measure thr Measure five			
Reference points	1	Measure on	e point per scan field	Z	
Left rear	30.50	mm	217.00 mm		
Right rear	275 50	mm	217.00 mm		

Fig. 13: Drop-down list Measurement strategy

Measurement strategy	Description
Measure three points	The measurement is performed at three points on the layout: • At the right rear corner
	At the left rear corner
	• At the front in the middle
	The average value is considered.
Measure five points	The measurement is performed at five points on the layout: • At the right rear corner
	At the left rear corner
	At the right front corner
	At the left front corner
	At the center
	The average value is considered.
Measure one point per scan field	The measurement is performed at one point in the center of each scan field. The measured value is considered for each scan field separately.



In case of a small layout with only one scan field, the system will always use the strategy *Measure one point per scan field*.

1.3.3 Measuring the material thickness automatically

This chapter describes how to perform material thickness measurement automatically during processing.

Ensure that the following prerequisites are fulfilled before performing the described tasks:

Prerequisites

- The material thickness measurement during processing is activated (see page 20).
- Processing is started and the dialog *Placement* is displayed.

Measuring the material thickness automatically

- 1. Place the processing data.
- 2. Click on [OK].

		Plac	ement	t	_ □
Material	Proces	sing data			
		ssing area to mov ttons to set the le			
		i i			
x 000,5 r	mm	x 305,5 m	m		
y 018,0 r	mm	y 247,0 m	im		
Width	305 mm	Length 229	mm	Set defau	lt size
Use a calip	er to mea	sure the material	thickne	ss	
Material th	ickness	1.634 mm	[Measure mater	ial thickness

Fig. 14: Dialog Placement

□ The laser system starts the material thickness measurement, following the measurement strategy you selected. The following message is displayed:

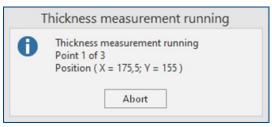


Fig. 15: Message Thickness measurement running

When the material thickness measurement is finished, processing continues.

If the deviation from the *Material thickness* (displayed in the dialog *Placement*) is higher than the *Material thickness tolerance* (see page 20), the following dialog is displayed:

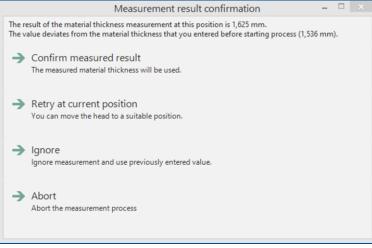


Fig. 16: Dialog Measurement result confirmation

- 3. If necessary, move the processing table to a more suitable position using double-clicks in the processing area or the arrow buttons in the pane *Navigation*.
- 4. Click on Retry at current position.
- □ The laser system starts the material thickness measurement, and the message *Thickness measurement running* is displayed.

When the material thickness measurement is finished, processing continues.

The material thickness has been measured automatically.

1.3.4 Measuring the material thickness manually (at a single point)

This chapter describes how to perform material thickness measurement manually at a single point on the base material before processing starts.

Ensure that the following prerequisites are fulfilled before performing the described tasks:

Prerequisites

- The user guidance step *Processing* is active.
- The processing head is located above the base material.

Measuring the material thickness manually

- 1. Right-click on the processing area.
- □ The following context menu is displayed:

ġ	Placement
• •• !!	Measure

Fig. 17: Context menu

- 2. Select the menu item Placement ...
- □ The following dialog is displayed:

	Placement	-		
Material	Processing data			
	he processing area to move the active head to the as se the buttons to set the left front and right rear corr			
x 000,5 r	nm x 305,5 mm			
y 018,0 r	nm y 247,0 mm			
Width 3	305 mm Length 229 mm Set default	size		
Use a calip	er to measure the material thickness			_
Material th	ickness 1.536 mm Measure material	thick	ness	
				-
	Apply OK		Cance	:1

Fig. 18: Dialog Placement



The input field *Material thickness* displays the material thickness defined in the selected material.

3. Click on [Measure material thickness].

□ The laser system starts the material thickness measurement at the position of the processing head. The following message is displayed:

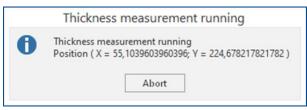
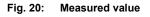


Fig. 19: Message Thickness measurement running

The measured value is displayed in the dialog *Placement*.

		Placemen	t	- 0	x
Material	Processing dat	a			
			ctive head to the as and right rear corn		
		•			
x 000,5 r	mm x	305,5 mm			
y 018,0 r	mm y	247,0 mm			
Width	305 mm Lengt	h 229 mm	Set default	size	
Use a calip	er to measure the	material thickne	ess		
Material th	ickness 1.634	mm	Measure material	thickness	
			OK	Cane	:el



- 4. Click on [OK].
- The material thickness has been measured manually.

1.3.5 Referencing the material thickness measurement sensor

This chapter describes how to perform referencing the material thickness measurement sensor. The referencing should be performed in the following situations:

- After replacing or cleaning the sinter plate.
- After transporting the system.
- In case of unexpected results during the material thickness measurement.

Ensure that the following prerequisites are fulfilled before performing the described tasks:

Prerequisites

Auxiliaries

- The user guidance step *Processing* is active.
- Base material of a uniform thickness, 229 mm × 305 mm (~9 in × ~12 in) (e.g., metal sheet for stencil production)

Referencing the material thickness measurement sensor

1. Mount the base material onto the processing table.

It is extremely important that you use a base material of uniform thickness. The flatness of the base material has a direct influence on the measurement result.

- 2. Click on Processing > Material thickness measurement setting.
- □ The dialog Material thickness measurement settings is displayed.
- 3. Enter the exact material thickness into the input field *Material thickness* (in this example *0.15 mm*).

	Material t	hickness r	neasurement s	ettings – 🗆 🛛		
Manual measu	rement can be	started via t	ne "Placement" mei	nu.		
✓ Perform thic	kness measur	ement during	process			
Material thickn	ess tolerance	100 µm				
Min hole distance		2 mm	2 mm			
Measurement s	trategy	Measure three points				
Referencing —						
Reference poin	its	X	Y	Z		
Left rear	30.50	mm	217.00 mm			
Right rear	275.5	0 mm	217.00 mm			
Center	153.0	0 mm	132.50 mm			
Front left	30.50) mm	48.00 mm			
Front right	275.5	0 mm	48.00 mm			
Material thickn	ess 0.15	mm				
	Old value		New value	Difference		
Reference	8.04 mm					
		Perform	referencing	OK Cancel		

Fig. 21: Entering Material thickness

4. Click on [Perform referencing].

□ The system measures the material thickness at five reference points. The measured values are displayed in the column *Z*. The average value is displayed in the column *New value*.

	laterial thickne	ss measurement se	ettings – 🗆 🗅			
Manual measurem	ent can be started	via the "Placement" mer	ıu.			
Perform thickne	ess measurement d	uring process				
Material thickness	tolerance 100	μm				
Min hole distance	2 n	nm				
Measurement strat	tegy Measu	Measure three points				
Referencing						
Reference points	X	Y	Z			
Left rear	30.50 mm	217.00 mm	8.07 mm			
Right rear	275.50 mm	217.00 mm	8.04 mm			
Center	153.00 mm	132.50 mm	8.05 mm			
Front left	30.50 mm	48.00 mm	8.12 mm			
Front right	275.50 mm	48.00 mm	8.04 mm			
Material thickness	0.15 mm					
	Old value	New value	Difference			
Reference	8.04 mm	8.05 mm	0.00 mm			

Fig. 22: Measured values and average value of the reference points

If the deviation between the minimum measured value and the maximum measured value exceeds 0.15 mm, repeat the referencing and/or use a base material that has a more uniform thickness.

5. Click on [OK].

□ The system saves the measured values of the reference points.



The value in the field *New value* is used for calculating the offset between the tip of the material thickness measurement sensor and the laser focus.

The material thickness measurement sensor has been referenced.

1.4 Working with mixed PCB layouts in CircuitPro PL

-	-

Working with mixed PCB layouts is only available in the license level Advanced of CircuitPro PL. If you have any questions contact the LPKF sales department.

This chapter describes how to import and process mixed PCB layouts at the same time in CircuitPro PL.

The following steps are performed in this tutorial:

- Preparing the individual PCB layouts
- Importing the individual PCB layouts

Preparing the individual PCB layouts

1. In the user guidance step *New*, select the template that suits the type of project you have.

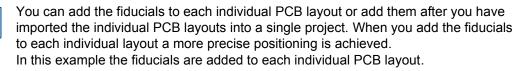
In this example the template *Double-sided*, *ProtoLaser S4*, *galvanic through-hole plating* is used.

- 2. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 3. Select a suitable material. In this example the material *FR4, Double-sided, 1.5 mm, 5/5 μm* is used.

Fig. 23: Select material

- 4. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 5. Click on 📑.

- 6. Select the files you want to import.
- 7. Click on [Open].
- □ The data are automatically assigned to the correct layers and the user guidance step *Import* is displayed.
- 8. Click on [Import].
- □ The imported data are displayed in the user guidance step *Layout*.



- 9. Click on Insert > Automatic fiducial creation or click on 🔩.
- □ The dialog Automatic fiducial creation is displayed.
- 10. Adapt the settings for the fiducials.
- 11. Click on [OK].
- □ The fiducials are automatically added to the layout.

You can compute the toolpaths of each individual PCB layout or compute them after you have imported the individual PCB layouts into a single project. In this example the toolpaths of each individual PCB layout are computed.

- □ The dialog *Computing toolpaths* is displayed.
- 13. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 14. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 15. Click on [Close].
- 16. Click on *File* > Save As... or on \square .
- 17. Select a suitable folder, name the file and click on [Save].
- □ The file is saved in the .cp2d file format.
- 18. Repeat all the steps in this procedure to prepare all the other individual PCB layouts.



Make sure you use **the same template and same material** for each imported PCB layout!

The individual PCB layouts have been prepared.

Importing the individual PCB layouts



Note that this project is an example. The layout of your project will be different.

- In the user guidance step *New*, select the template that you have used for the preparation of your individual PCB layouts.
 In this example the template *Double-sided*, *ProtoLaser S4*, *galvanic through-hole plating* is used.
- 2. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- Select the material that you have used for the preparation of your individual PCB layouts.

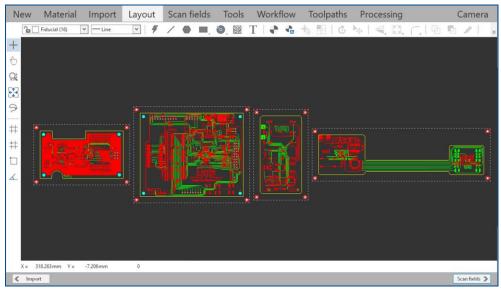
In this example the material FR4, Double-sided, 1.5 mm, 5/5 μ m is used.

- 4. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 5. Click on T.
- 6. Navigate to the folder that contains the .cp2d files you saved in the previous procedure.
- 7. Press and hold the Ctrl key and select the files you want to import.
- 8. Click on [Open].
- □ The user guidance step *Import* is displayed.

Nev	w N	/laterial	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
đ	P	CB layout 1.cp2d	I.	CP2	2D document	~	PCB layout 1.cp2d	~	^	
		Source		Targ	et		Size/Format			
_		✓ Fiducial		Fidu	ucial	~	88.855 x 53.943 mm			
зI		✓ BoardOutlin	ne	Boa	rdOutline	~	80.632 x 45.72 mm			
-		✓ DrillPlated		Dril	Plated	~	76.543 x 42.57 mm			
_		✓ TopLayer		Тор	Layer	~	76.403 x 40.526 mm			
_		✓ SilkScreenTe	op	Silk	ScreenTop	~	88.855 x 53.943 mm			
_		✓ SolderMask	Тор	Solo	der Mask Top	~	91.855 x 56.943 mm			
_		BottomLay	er	Bot	tomLayer	~	66.42 x 34.003 mm			
_		✓ SilkScreenB	ottom	Silk	ScreenBottom	~	88.855 x 53.943 mm			
_		✓ SolderMask	Bottom	Solo	derMaskBottom	~	91.855 x 56.943 mm			
_				Gal	vanic THP Drill Fiducials	~	88.855 x 53.943 mm			
_		Drill Fiducia	ils_Drills	Dril	s	~	88.855 x 53.943 mm			
		_		Gah	vanic THP Drill Plated Th	rough-Hole: 🗸	1		~	
	<u>58</u> ,63 					55.63	•			
<	Material			-5.61	25.01	Discard	Import			Layout 🕽

Fig. 24: User guidance step Import

9. Click on [Import].



□ The imported PCB layouts are displayed in the user guidance step *Layout*.

Fig. 25: Imported PCB layouts

i

In this example, one of the PCB layouts is multiplied (by using instances).

10. Select one PCB layout by clicking on the dashed line surrounding it.

□ The layout is selected and changes its color.

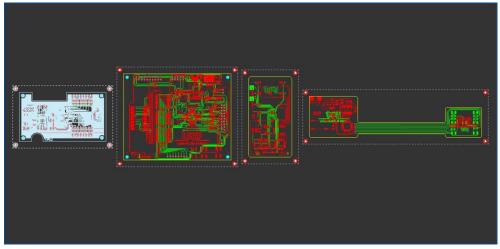


Fig. 26: One PCB layout selected

11. Right-click on the black background and select *Edit instance type...* from the context menu.

	Edit instance type					
	Invert selection					
Dissolve instance type						
▶	Move object Ctrl+M					
Ċ	Rotate					
	horizontally					
	vertically					
	Assign to layer					
	Select by layer					
-	Export Ctrl+E					
· · ·	Measure					
X	Delete					

Fig. 27: Menu Edit instance type

□ The following dialog is displayed:

Edit instance type				
Count	x 🚺	y 1		
Distance	x 171.423 mm	y 49.674 mm		
	OK	Cancel		

Fig. 28: Dialog Edit instance type

- 12. Enter the desired number of instances in the input fields *Count*. In this example, enter **3** counts for the y axis.
- 13. Increase the entered value under *Distance* in the input field *y* by **10** mm.
- □ The dialog *Edit instance type* changes as follows:

	E	dit instance	type	e 🔸
Count	×	1	у	3
Distance	x	91.855 mm	у	66.943 mm
		OK		Cancel

Fig. 29:Dialog Edit instance type | Values14. Click on [OK].

□ The selected layout is multiplied:

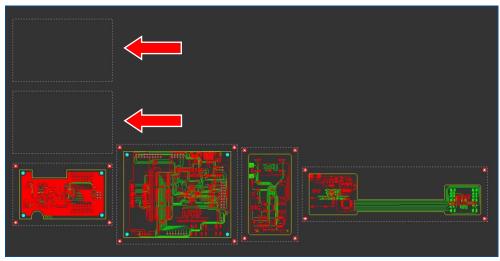


Fig. 30: Layout multiplied

15. Compute toolpaths, if you have not computed them in the individual PCB layouts yet.

In this example, the toolpaths have already been computed in the individual PCB layouts.

- 16. Switch to the user guidance step *Processing*.
- □ The PCB layouts are displayed in the center of the base material:

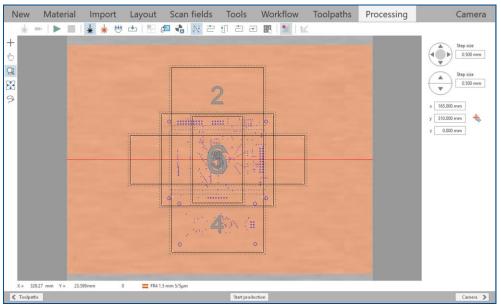
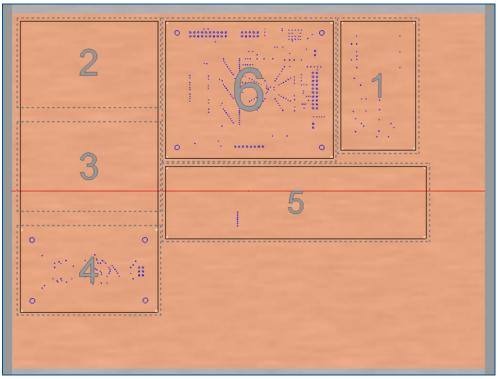
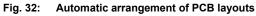


Fig. 31: User guidance step Processing



17. Click on 🚈 to arrange the PCB layouts automatically.



- 18. Right-click on the background.
- 19. Select the menu item *Placement...* from the menu.

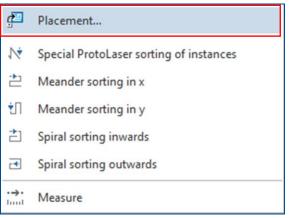


Fig. 33: Menu item Placement

□ The dialog *Placement* is displayed.

- 20. Click on the tab Processing data.
- 21. Enter **15** mm in the input field *Job distance*.

	Placement – 🗆 🗙					
Material Processing data						
Drag and drop the bounding box of data in the processing view.						
Project	PCB layout 1.cp2d					
Center of processing data x 74.645 mm y 108.871 mm						
✓ Use information layers	Center on material					
Automatic arrangement						
Job distance 15 mm Arrange all						
Processing head						
Center on data						
Show bounding box						
	Apply OK Cancel					

Fig. 34: Dialog Placement | Processing data

- 22. Click on [Arrange all].
- 23. Click on [OK].
- □ The distance between individual PCB layouts is increased.

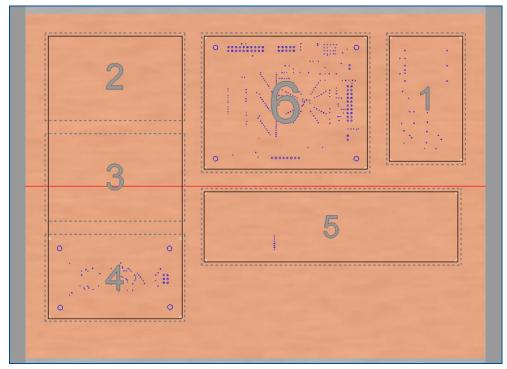


Fig. 35: Distance between PCB layouts increased



Alternatively, arrange the PCB layouts manually. Open the dialog *Placement*, click on the tab *Processing data*, and perform one of the following steps:

- Drag & drop each individual PCB layout to the desired location.
- Select the individual PCB layout in the drop-down list *Project* and enter the values in the input fields x and y for *Center of processing data*.

	Placement – 🗆 🗙
Material Processing data	
Drag and drop the bounding b	ox of data in the processing view.
Project	PCB layout 1.cp2d
Center of processing data x	153 mm y 132.5 mm
✓ Use information layers	Center on material
Automatic arrangement Job distance 5 mm Processing head Center on data Show bounding box	Arrange all
	Apply OK Cancel

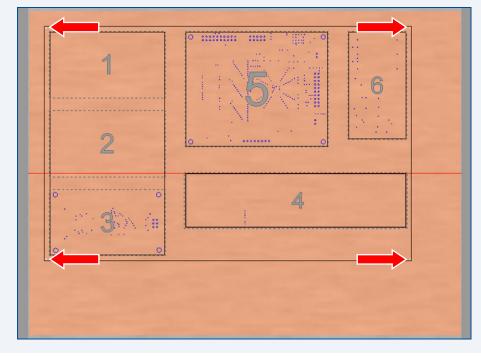
- 24. Save your project.
- The individual PCB layouts have been imported.

The individual PCB layouts have successfully been prepared and imported. You can now start processing the mixed PCB layouts.

Creating fiducials after importing the individual PCB layouts

If you have not created fiducials in the individual PCB layouts, you can create them after importing the .cp2d files into a single project.

- 1. Multiply the individual layouts in the user guidance step *Layout*, if desired.
- 2. Calculate the toolpaths, if necessary.
- 3. Switch to the user guidance step *Processing*.
- 4. Arrange the layouts on the base material.
- 5. Click on 🔩.
- □ The dialog *Automatic fiducial creation* is displayed.
- 6. Adapt the settings for the fiducials, if necessary.
- 7. Click on [OK].
- □ The fiducials are created automatically with calculated scan fields and toolpaths.



1.5 Testing and tuning the tool settings

This chapter describes testing and tuning the tool settings in the software CircuitPro PL. The procedure follows the project placement, the fiducial recognition and the material thickness measurement described in the previous chapters.

The aim of this procedure is to check the processing quality on a small test sample on your material before you start to process your project. You can modify the tool settings to improve the processing quality.



The procedure is especially useful for checking the processing quality of the galvanically through-hole plated material.

Ensure that the following prerequisites are fulfilled before performing the described tasks:

Prerequisites

Spare parts and auxiliaries

- The user guidance step *Processing* is active.
- Portable hand-held microscope
- The dialog *Test tool settings* is displayed.



The dialog *Test tool settings* is automatically displayed during processing before structuring starts.

You can also display the dialog by performing the following steps:

- 1. Switch to the user guidance step *Processing*.
- 2. Expand the group *Processing* in the pane *Workflow setup*.
- 3. Expand a desired phase.
- 4. Right-click on the desired work package Structure or work package Drill.
- 5. Select Check tool settings in the context menu.

The following steps are described:

- Placing and processing the test sample
- Checking the processed test sample
- Tuning the tool settings
- Saving the tool settings globally
- Resuming the processing procedure
- Disabling the test procedure of the tool settings (optional)



- 1. Move the dialog *Test tool settings* to get a better overview.
- 2. Click on Place and process "Small test sample 7x7 mm".

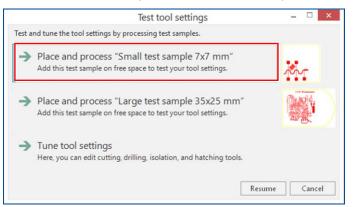


Fig. 36: Place and process "Small test sample 7x7 mm"



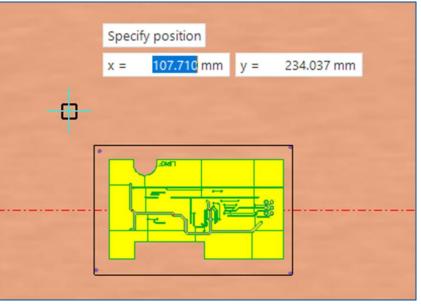


Fig. 37: Specifying the position



The best position of the test sample is just outside of the outer contour.

- 3. Perform one of the following steps:
 - Click on a desired point in the processing area to specify the position of the test sample.
 - Enter the values in the input fields for *x* and *y*.

□ The test sample is displayed in the processing area. The following dialog is displayed:

Test tool settings
Process test sample now?
Process Cancel

Fig. 38: Dialog Test tool settings

- 4. Click on [Process].
- Processing the test sample is started.
 When processing the test sample is finished, the following dialog is displayed:

	Check test tool finished	
0	Check test tool finished	

Fig. 39: Dialog Check test tool finished

- 5. Click on [Close].
- □ The dialog *Test tool settings* is displayed.
- The test sample has been placed and processed.

Checking the processed test sample

- 1. Check the processed test sample using a portable hand-held microscope.
- 2. Identify possible faults in the processing quality.
- 3. If the processing quality of the test sample is acceptable, proceed with the step *Resuming the processing procedure* (see page 45).
- 4. If the processing quality of the test sample is not acceptable, proceed with the step *Tuning the tool settings* (see page 42).
- The processed test sample has been checked.



For detailed information on optimizing the processing quality, refer to the TechNote **ProtoLaser: Optimizing the processing quality**.

Tuning the tool settings

1. Click on *Tune tool settings*.

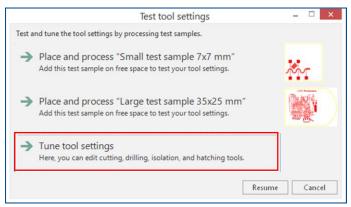
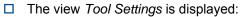


Fig. 40: Tune tool settings



Workflow setup 🔍	Processing < Tool Settings	
 Material composition 		
▷ FR4 1.5 mm 18/18µm	🚍 FR4 1.5 mm 18/18μm 18 μm Isolate 💙 🐔	9
D Layout	Tool type Isolation	General Advanced
Scan fields	트루 Tool type Isolation	Laser parameters
▲ Tools	Tool name Isolate	
4 FR4 1.5 mm 18/18μm	_	Frequency 60 kHz Pulse energy 217 µJ
🖻 🌏 Isolate	Task 1	Power 13 W
D 😔 PreCut	0 mm, 13 W, 60 kHz, 270 mm/s, RT 1	Mark speed 270 mm/s
Divide Strip		
D 😔 Hatch		Process parameters
▷ 🌏 Heat ▷ 🌏 Short Heat		
 Workflow 		Focus offset 0 mm
 Vorknow Toolpaths 		Repetitions 1 Delay 0 ms
 Processing 		✓ Enable air flow
Processing		Chapte air now
		Processing
	Processing	Discard Apply

Fig. 41: View Tool Settings

i

2. In the pane *Workflow setup* select a specific tool, whose settings you wish to modify (in this example the tool *Heat* is selected).

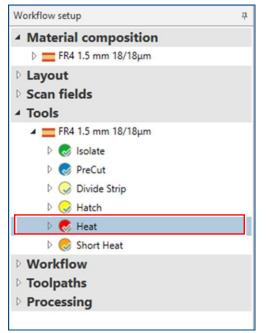
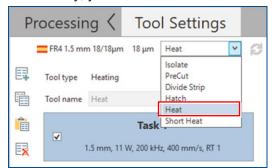


Fig. 42: Tool selected

Alternatively, you can select a tool from the drop-down list:



3. Modify the tool settings in the tab *General* and/or tab *Advanced*.

Processing 🔇 Tool Settings	
🚃 FR4 1.5 mm 18/18μm 18 μm Heat 💙 💭	
Tool type Heating Tool name Heat	General Advanced Laser parameters Frequency 200 kHz Pulse energy 55 μJ
Task 1 I.5 mm, 11 W, 200 KHz, 400 mm/s, RT 1	Power 11 W Mark speed 400 mm/s Process parameters
	Enable air flow Processing
Processing	Discard Apply

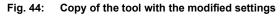
Fig. 43: Modifying the tool settings



For detailed information on modifying the tool settings (i.e., parameters), refer to the TechNote **ProtoLaser: Optimizing the processing quality**.

- 4. Click on [Apply].
- □ A copy of the tool with the modified settings is created. The green check mark (✓) next to the tool name indicates that this tool will be used for processing in this project.

Workflow setup	Ą
 Material composition 	
▷ FR4 1.5 mm 18/18µm	
Layout	
Scan fields	
4 Tools	
🖌 🚃 FR4 1.5 mm 18/18μm	
🖻 🌏 Isolate	
🖻 🎯 PreCut	
👂 😡 Divide Strip	
👂 🥪 Hatch	
🕨 🛑 Heat	
👂 💭 🌏 Heat Copy	
👂 🌏 Short Heat	
Workflow	
Toolpaths	
Processing	



- 5. Perform steps 2 to 4, if you wish to modify the settings of another tool.
- The tool settings have been tuned.

Saving the tool settings globally

The new tools with the modified settings will be used for processing only in your current project. Save the new tools to the global catalog, if you wish to use them for your other projects in CircuitPro PL.

1. Click on \mathfrak{S} next to the tool name with the modified settings.

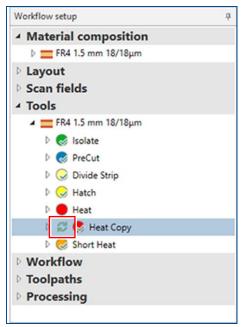


Fig. 45: Saving the tool

□ The tool is saved to the global catalog and is displayed in the pane *Workflow setup* as follows:

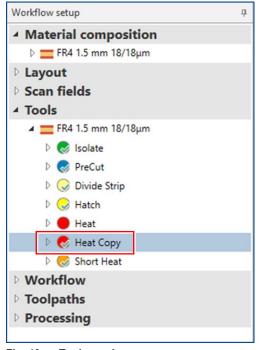


Fig. 46: Tool saved

2. Click on the user guidance step Processing to leave the view Tool Settings.

The tool settings have been saved globally.

In order to check if the processing quality is optimal now, perform the sequences *Placing and processing the test sample* (see page 40) and *Checking the processed test sample* (see page 41) again.

Resuming the processing procedure

In the dialog Test tool settings click on [Resume].

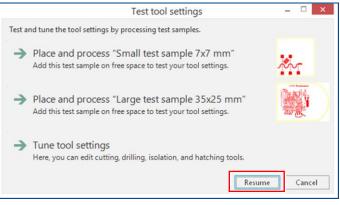


Fig. 47: Resuming the processing procedure

The processing procedure has been resumed.

Disabling the test procedure of the tool settings (optional)

If desired, you can disable the test procedure of the tool settings for your project.

- 1. Switch to the user guidance step *Workflow*.
- 2. Select a phase that includes a work package *Structure* (or more of them) from the drop-down list.

New	Material	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
Bottom Si Bottom Si	TUD] [6] Iŭ							
Mount Mater	ial Bottom	Check Copper T	'nickn →	Material Placement		d Fiducials Bottom	Structure Bottom	→ Top Side	
Work packa V Activat Name Type Instruction	Mount Material I	~							
Message	Mount the mater Bottom facing up processing area.	ial with the side							
< Tools				Discar	i Apply	Compute toolpaths			Toolpaths 义

Fig. 48: Phase Bottom Side selected

3. Select the work package Structure.

New	Material	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
Bottom	Side 💌 📮 🗌	101							
V Mount Mat	terial Bottom	Check Copper	ſhickn →	Material Placement	→ Read	Fiducials Bottom	Structure Bottom	→ Top Side	
Activ	kage options								
Name	Structure Bott]						
Туре	Structuring	~	-						
Scan fie			-						
Sorting	policy ProtoLaser st	andard V							
Paramet	ers								
< Tools				Discare	i Apply	Compute toolpaths]		Toolpaths 义

Fig. 49: Work package Structure Bottom selected

4. Deactivate the check box *Test tool settings*.

Test tool set	Structure Bottom		
Туре	Structuring	~	
Scan field set	Bottom Side scan field set	*	
Sorting policy	ProtoLaser standard	~	
E	RR		

Fig. 50: Test tool settings disabled

- 5. Repeat the steps 2 to 4 for other phases that include a work package *Structure*.
- 6. Click on [Apply].
- The test procedure of the tool settings has been disabled.

1.6 Processing multi layers

This chapter describes the multi-layer process using the MultiPress S. The processing steps are explained briefly and some useful production tips are given.

For detailed information on the multi-layer process refer to the user manual of **MultiPress S**.

The following steps are described:

- Starting the preheating stage
- Preparing the materials
- Assembling the multi-layer stack
- Pressing and curing the multi-layer stack

Starting the preheating stage

- 1. Switch on the MultiPress S.
- 2. Make sure that the profile *LPKF* Set is shown in the display.

If the *LPKF Set* profile is not shown in the display, refer to the user manual of MultiPress S.

- 3. In the main menu, select the entry *Start* and press the button *ENT*.
- 4. Select the entry Preheating and press the button ENT.
- □ The system will heat up to the preset temperature. The display shows the current data of the preheating stage.

Assemble the multi-layer stack during the preheating stage (described in the following two procedures).

The preheating stage has been started.

Preparing the materials

i

Keep the materials at **ambient temperature** for **24 hours** before use. To avoid skin fat deposit and dust on the materials, work with lint-free gloves.

- 1. Clean the press molds and press sheets with isopropyl alcohol or acetone. Scrape off any resin residues from previous cycles.
- 2. Clean the core materials and laminate materials with isopropyl alcohol or acetone.
- Heat-treat the core materials and laminate materials before assembly at 100 °C (~212 °F) for 30 minutes (to reduce moisture content).

Do not heat-treat prepreg materials!

- 4. Cool the materials for 10 minutes at ambient temperature.
- The materials have been prepared.



After cooling, immediately proceed with assembling the multi-layer stack! For detailed information on assembling the multi-layer stack refer to the user manual of MultiPress S.

Assembling the multi-layer stack

i

Dowel pins in the press mold align the individual layers. Rings between the bonded layers protect the register holes and the fiducials from being covered with prepreg resin.

1. Assemble the press mold and the materials (**starting** with the **lower press mold**) according to the following figure:

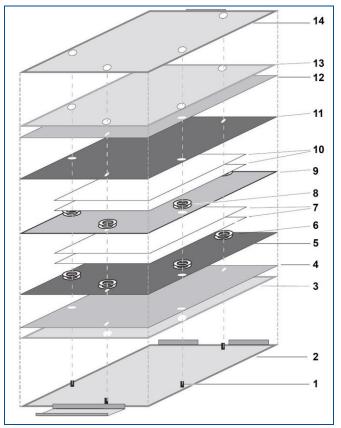


Fig. 51: Assembly of the press mold and the materials

- 1 Dowel pins
- 2 Aluminum press mold (lower part)
- 3 Press cushion
- 4 Steel press sheet
- 5 Bottom laminate (copper side facing downwards)
- 6 Sealing rings (1 or 2 rings)
- 7 Prepreg (1 or 2 sheets)

- 8 Sealing rings (1 or 2 rings)
- 9 Core material
- 10 Prepreg (1 or 2 sheets)
- **11** Top laminate (copper side facing upwards)
- 12 Steel press sheet
- 13 Press cushion
- 14 Aluminum press mold (upper part)



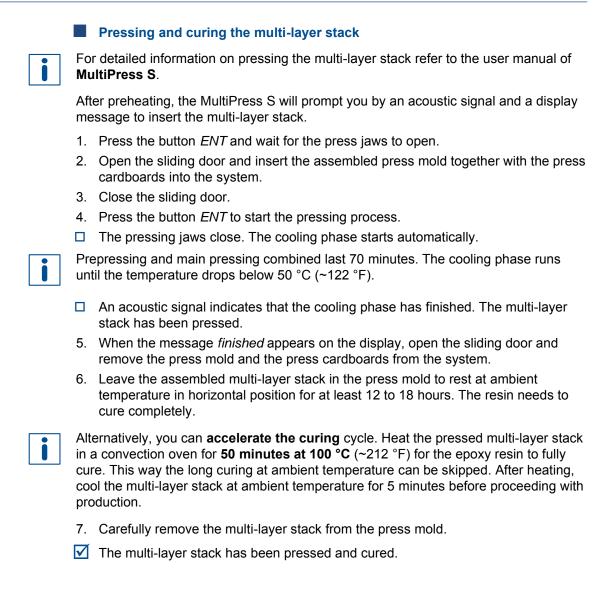
When using one sheet of prepreg, one ring should be used on each dowel pin. When using two sheets of prepreg, two rings should be used on each dowel pin.

2. Place the press mold containing the materials to be pressed between two blue press cardboard sheets.



Use original LPKF press cardboard sheets, since press cardboards that are not heat-resistant can ignite during operation and set the system on fire.

The multi-layer stack has been assembled.



1.7 Processing methods of the ProtoLaser

This chapter describes the processing methods of the ProtoLaser systems. There are two methods of material removal:

- Delamination method
- Hatching method

The delamination method is only used for processing laminated PCB materials. The hatching method is used for processing laminated PCB materials and non-laminated PCB materials.

Laminated PCB materials usually consist of a conductive layer (e.g. copper) that is laminated with a bonding layer onto a non-conductive substrate (e.g. FR4):

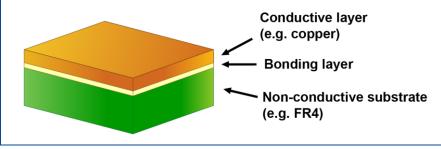


Fig. 52: Example of a laminated PCB material

Non-laminated PCB materials usually consist of a conductive layer (e.g. gold, copper) that is electroplated/electrodeposited onto a non-conductive substrate (e.g. Aluminum oxide $- Al_2O_3$):

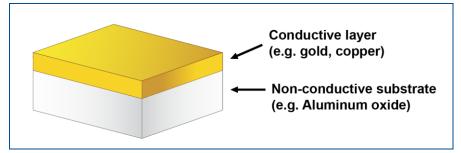
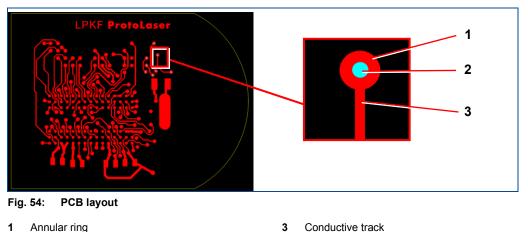


Fig. 53: Example of a non-laminated PCB material

The conductive layer to be removed is called the rubout area. Both terms are used in this document.

1.7.1 Delamination method

The following figure shows an example layout of a PCB in the user guidance step *Toolpaths*. It is used for explaining the delamination method:



The delamination method consists of the following stages:

- 1. Creating the isolation channels
- 2. Creating strips

2

Hole

3. Delaminating strips

Creating the isolation channels

Processing starts by creating isolation channels (i.e. "contours") around the objects (conductive tracks, pads, etc.). The isolation channels separate the rubout area from the PCB layout on the conductive layer. The following figure shows the isolation channel around the conductive track in the user guidance step *Toolpaths*:

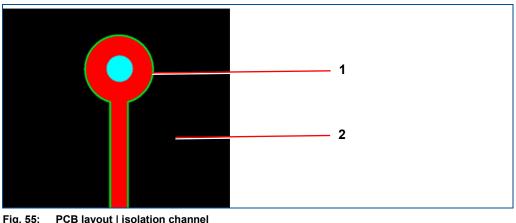
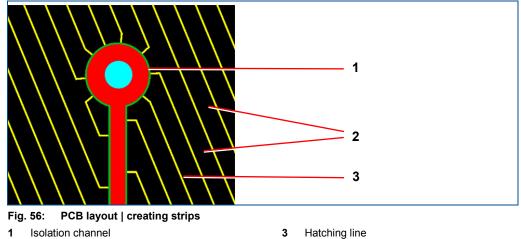


Fig. 55: PCB layout | isolation channel 1 Isolation channel

² Rubout area

Creating strips

The process is continued by creating strips on the rubout area. The following figure shows the strips with the hatching lines in the user guidance step Toolpaths:

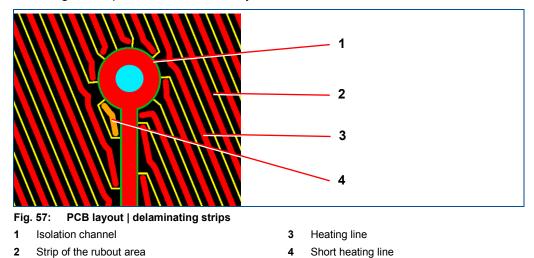


1 Isolation channel

2 Strips of the rubout area

Delaminating strips

The final stage is the delamination of strips. Each strip of the rubout area is being heated up until the conductive layer is removed from the substrate. The following figure shows heating lines in the user guidance step Toolpaths that are used for removing the strips of the conductive layer:



1.7.2 Hatching method

The following figure shows an example layout of an RF PCB in the user guidance step *Toolpaths.*

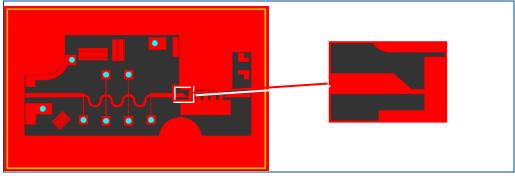


Fig. 58: RF PCB layout | RF geometry

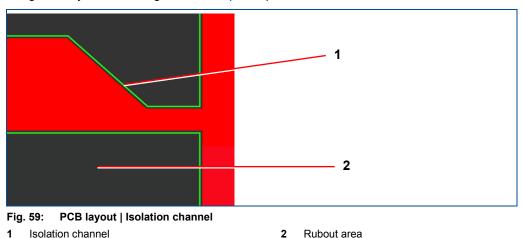
The hatching method consists of the following stages:

- 1. Creating the isolation channels
- 2. Hatching

In comparison to the delamination method, the conductive layer is removed by ablation and not by delamination.

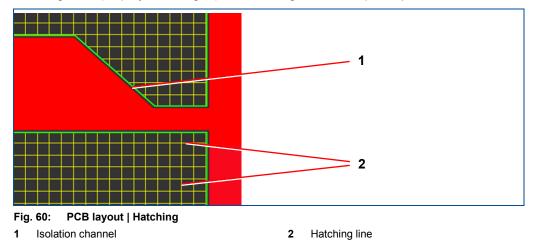
Creating the isolation channels

Processing starts by creating isolation channels (i.e. contours) around the objects (e.g. RF geometries). The isolation channels separate the rubout area from the PCB layout on the conductive layer. The following figure shows the isolation channel around the RF geometry in the user guidance step *Toolpaths*:



Hatching

The process is continued. The rubout area is cut into lines that are very close to each other (vertically, horizontally or in both directions). The following figure shows the hatching lines (displayed as a grid) in the user guidance step *Toolpaths*:



The laser beam is wider than the hatching lines displayed in figure 10, so that the structured hatching lines overlap during processing. This way all the material is removed from the rubout area.

1.8 Design guidelines for the production of multi-layer PCBs

This chapter describes basic design guidelines that should be observed when designing a layout for a multi-layer PCB or a multi-layer PCB with blind vias and buried vias. These design guidelines apply to the production of multi-layer PCBs with LPKF systems and with materials approved by LPKF.

1.8.1 Structures of multi-layer PCBs

This section explains the structures of multi-layer PCBs and multi-layer PCBs with blind vias and buried vias.

The following figures show the following information:

- The sequence of the materials used;
- Identification of layers;
- Illustration of different hole types;
- The materials used in a specific procedure.

The drill layer names indicate which two layers are connected by a hole. For example: The drill layer *Blind via (Top-L2)* connects the Top layer and Layer 2.

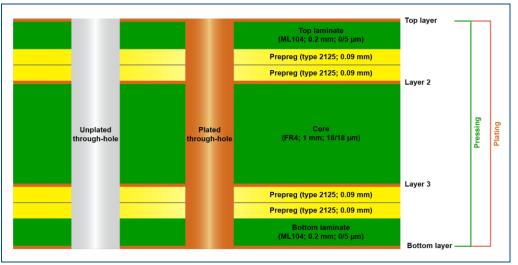


Fig. 61: A 4-layer PCB structure

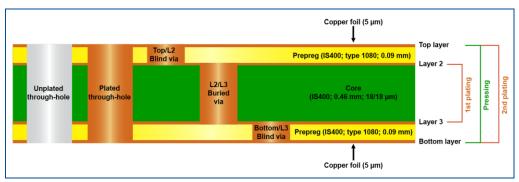


Fig. 62: A 4-layer PCB structure with blind vias and buried vias

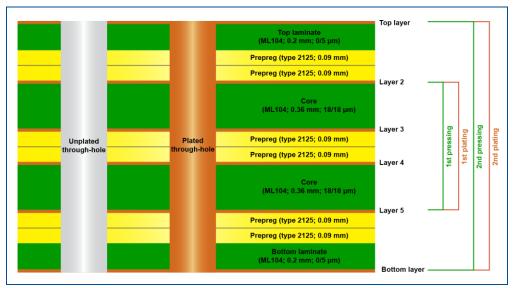


Fig. 63: A 6-layer PCB structure

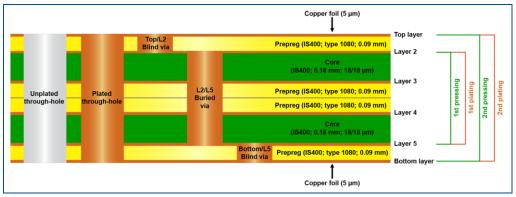


Fig. 64: A 6-layer PCB structure with blind vias and buried vias

i

To produce an 8-layer PCB structure use one additional core material and two additional prepreg materials (compared to the 6-layer PCB structure).

1.8.2 **Required files/layers**

The following table contains the required files for designing 4-layer PCBs:

Structuring	Drilling	Other
TopLayer	DrillPlated (plated through hole)	BoardOutline
Layer 2	DrillUnplated (unplated through hole)	SolderMaskTop
Layer 3	Blind via (Top-L2)*	SolderMaskBottom
BottomLayer	Buried via (L2-L3)*	
	Blind via (Bottom-L3)*	

Table 2: Required files/layers

* Applicable for 4-layer PCBs with blind vias and buried vias.

1.8.3 Through holes and via sizes before plating

When designing the layout of a multi-layer PCB, it is important to observe the size range of the holes.

Hole type	Minimum size	Maximum size
Through hole	200 µm	1
Buried via	200 µm	400 µm
Blind via	100 µm	300 µm

The following table specifies the size range for a certain hole type:

Table 3: Size range according to hole type

A multi-layer PCB with hole sizes different from these will not function properly.

1.8.4 Annular rings

An annular ring is a copper ring around a plated hole and its width is an important design and manufacturing consideration. If a wide annular ring area is provided in the design, it ensures that in the manufactured printed circuit board a good electrical connectivity between pad and hole is retained.

The following figure shows an examplary annular ring:

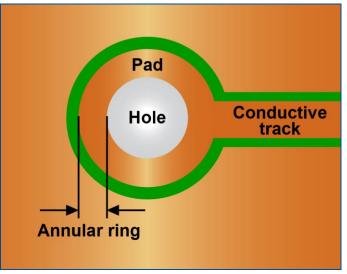


Fig. 65: Annular ring

The minimum width of the annular ring is the **minimum amount of copper** between the **edge of the hole and the edge of the pad** after plating of the finished hole. The PCB production process using LPKF systems complies with the IPC 2221-B standard. According to this standard the minimum annular ring for Class 3 shall **not be less than 150 \mum**.

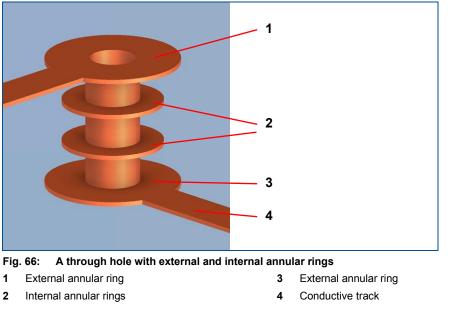
External annular ring

An external annular ring is an annular ring on the external layers of a multi-layer PCB.

Internal annular ring

An internal annular ring is an annular ring on the internal layers of a multi-layer PCB. A through hole should always have an annular ring on every passing layer, regardless of its electrical connectivity.

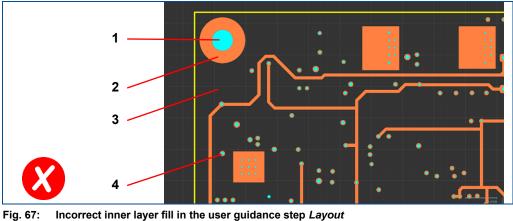
The following figure shows a through hole crossing four layers, as well as its internal and external annular rings:



Always use a minimum width of at least **100 µm** for internal annular rings.

1.8.5 Inner layers fill

All inner layers should be filled with copper as much as possible. In case of larger rubout areas, the epoxy content of the prepreg may not be sufficient to fill all gaps. Consequently, air filled voids between layers can expand during the reflow process and cause a blister effect on a multi-layer PCB.

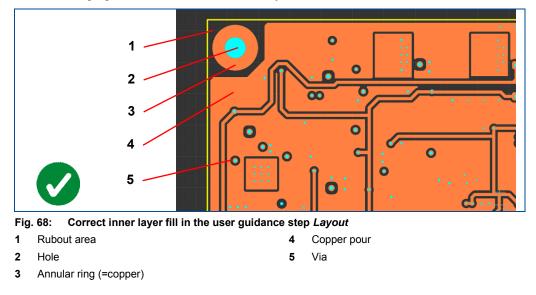


The following figure shows an incorrect inner layer fill:

Hole	3	Rubout area
Annular ring	4	Via

1

2



The following figure shows a correct inner layer fill:

If the inner layers are less than 75 % filled with copper (a rough estimation), two prepreg sheets must be used.

1.8.6 Staggered vias

Staggered vias are vias on neighboring layers that are located close to each other, but do not overlap. Staggered vias only apply to multi-layer PCBs with blind vias and buried vias.

Before plating, a **minimum offset of 150 µm** between vias on neighboring layers must be observed during the design of the PCB layout.

The following figure shows staggered vias and the minimum distance between them:

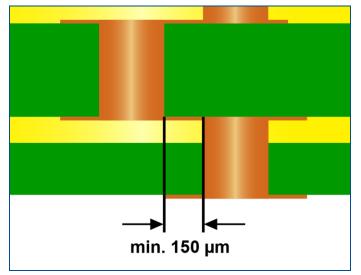


Fig. 69: Staggered vias and minimum distance

1.8.7 Stacked vias

Stacked vias are vias on neighboring layers that overlap. Stacked vias only apply to multi-layer PCBs with blind vias and buried vias. The PCB production process with LPKF systems and software does not support stacking of vias. **Use staggered vias** instead.

The following figure shows stacked vias:

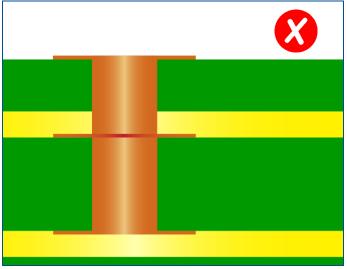
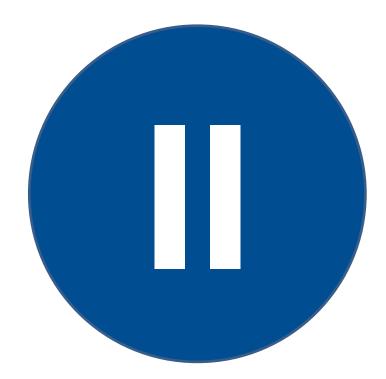


Fig. 70: Stacked vias

Double-sided PCBs



2 Producing double-sided PCBs

This chapter describes the production of four different types of double-sided PCBs:

- 1. Producing a double-sided PCB without through-hole plating.
- 2. Producing a double-sided PCB with galvanic through-hole plating.
- 3. Producing a double-sided PCB with non-galvanic through-hole plating.
- 4. Producing a double-sided flexible PCB with galvanic through-hole plating.

The following LPKF systems are required for the procedures:

Procedure	LPKF system
1	ProtoLaser U4/S4/R4
2	ProtoLaser U4/S4/R4, ProtoMat S or E, Contac S4
3	ProtoLaser U4/S4/R4, ProtoMat S or E, ProConduct
4	ProtoLaser R4, Contac S4

 Table 4:
 Required LPKF systems

2.1 How to produce a double-sided PCB without through-hole plating

This chapter describes how to create a double-sided PCB without through-hole plating using a laser system only.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

Auxiliaries

LPKF Cleaner

System

- ProtoLaser U4/S4/R4
- (order code: 115891)Oil-free compressed air
 - Oil-free compressed ai Brush
- Tap water

The following steps are performed in this tutorial:

• Switching on the system

Base material FR4,

229 mm × 305 mm ×

1.5 mm, 18/18 µm

(order code: 115967)

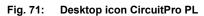
copper-plated

- Selecting the material
- Importing the data
- Multiplying the layout (optional)
- Creating fiducials
- Computing toolpaths and scan fields automatically
- Processing the PCB
- Cleaning the system
- Cleaning the PCB

Switching on the system

- 1. Press the on/off button at the system front.
- The system is started up. The PC boots automatically. The on/off button is lit.
- 2. Double-click on the desktop icon of LPKF CircuitPro PL.





□ The following message is displayed:

	Sy	stem co	nnection) 		×
Connecting t	ne system	- Checkin	g status light	s - Updat	te - 50	%
	-					

Fig. 72: Message System connection

□ The system software recognizes the system automatically and establishes the connection. The following dialog is displayed:

	Signal light check
A	Check whether all signal lights are switched on. Are the red, yellow, and green signal lights lit?
	Yes No

Fig. 73: Dialog Signal light check

- 3. If all lamps of the stack light are lit, click on [Yes].
- □ The user guidance step *New* is displayed:

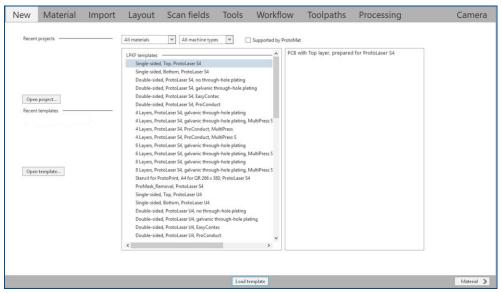
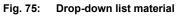


Fig. 74: User guidance step New

4. Select *Double-sided* from the drop-down list.

All materials	~
All materials	
Single-sided	
Double-sided	
4-layer-Multilayer	
6-layer-Multilayer	
8-layer-Multilayer	



5. Select your laser system from the drop-down list (in this example PL U4).

All machine types	<
PL S4	
PL U4	
PL ST	
PL R4	
PL U3	
PL R	
PL S	
All machine types	

Fig. 76: Drop-down list system

□ A list of templates for double-sided materials is displayed:

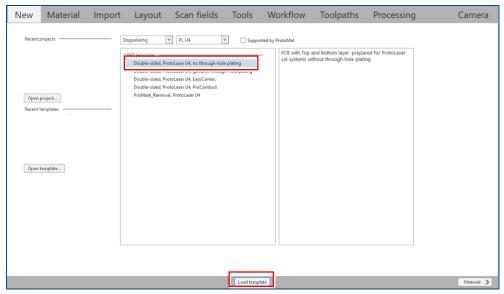


Fig. 77: List of templates

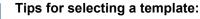
- 6. Select the template Double-sided, ProtoLaser U4, no through-hole plating.
- 7. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- The system has been switched on.

Connecting the system manually

If automatic connection of the system fails, you can connect the system manually with the system software: Click on *Processing > Connect to machine...*, select your system in the drop-down list and click on [Connect].

The system requires a warm-up time of up to **20** minutes for the laser source to attain a constant diode temperature. The warm-up phase starts automatically when processing the first job.

Alternatively, you can start the warm-up phase manually. In the user guidance step *Processing*, click on $\frac{1}{4}$. You can continue to work in the user guidance step *Layout* during the warm-up phase.



- Select the template according to the number of layers.
- Select the template according to the metallization type.
- Select the template according to the type of multi-layer press.

Selecting the material

1. In the user guidance step *Material* select the material *FR4 Double-sided*, *1.5 mm*, *18/18 μm*.

Double-sided 🗸			<u>_</u>				
FR4 Double-sided, 1 mm, 18/18µm FR4 Double-sided, 1.5 mm, 18/18µm	6 ^	Material n Total thick Treat as no		i mm			
rn+ Double-sided, 1,5 mm, 35/35µm	â	Service					
FR4 Double-sided, 1.5 mm, 5/5µm	•	Copper	18 µm		Top Lay	er 🗸	
FR4 Nanya Double-sided, 1,5 mm, 35/35µm	6	Core	1.5 mm				
IS400 Double-sided, 0.18 mm, 18/18µm	•	Copper	18 µm		Bottom	Layer	
IS400 Double-sided, 0.46 mm, 18/18μm	â						
ML104 Double-sided, 0.36 mm, 18/18µm	ô						
PyraluxTK Double-sided, 0.12 mm, 12/12µm	ô						
RO3003C Double-sided, 0,51 mm, 18/18µm	ô						
RO3003C Double-sided, 0,51 mm, 35/35µm	ô						
RO3006 Double-sided, 0,64 mm, 35/35µm	ô						
000000	~						

Fig. 78: User guidance step Material

- 2. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- The material has been selected.

Selecting the material type

When selecting the *Material* you are also selecting the tools with laser parameters for a specific material.

Example of the material name composition:

FR4 Double-sided, 1 mm, 18/18µm	6	^
FR4 Double-sided, 1.5 mm, 18/18µm	6	
FR4 Double-sided, 1,5 mm, 35/35µm	â	
FR4 Double-sided, 1,5 mm, 5/5µm	6	
FR4 Nanya Double-sided, 1.5 mm, 35/35µm	A	
IS400 Double-sided, 0,18 mm, 18/18μm	â	
16400		-

FR4 Double-sided – material type

- 1.55 mm material thickness
- 18/18 μm copper thickness

Importing the data

- 1. In the user guidance step *Import* click on [1].
- □ The following dialog is displayed:

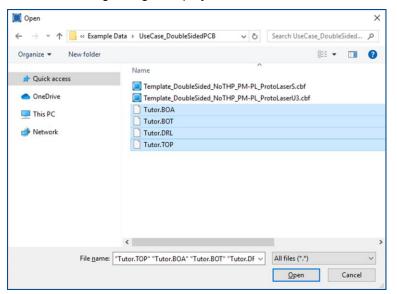


Fig. 79: Dialog Open

2. Navigate to the folder that contains the data you wish to import. The example data used for this tutorial are located in the folder:

C:\Program Files\LPKF Laser & Electronics AG\ LPKF CircuitPro PL\Example Data\UseCase DoubleSidedPCB.

- 3. Select the desired files for import (in this example *Tutor.BOA, Tutor.BOT, Tutor.DRL* and *Tutor.TOP*).
- 4. Click on [Open].
- □ The data are automatically assigned to the correct layers and the user guidance step *Import* is displayed:

New	Material In	nport Lay	/out	Scan fields	Тос	ols Workflow	Toolpath	s Pr	ocessing	Camera
C .	Tutor.BOA		GerberX		~	BoardApe	~	File	Format	
Ū	Source		Target			Size/Format		Unit	Millimeters 🖂	
4	✓ BoardOutline		BoardOutli	ne		80.632 x 45.72 mm		Values	Absolute V	
æ					_					
/C _	Tutor.BOT		GerberX		~	BotApe	~	Decimal	Omit leading zeros	
	Source		Target			Size/Format		Digits m.r	n 2 3	
	GerberX data		BottomLay	er	~	66.42 x 34.003 mm				
	Tutor.TOP		GerberX		~	ТорАре	~			
	Source		Target			Size/Format				
	GerberX data		TopLayer		~	76.403 x 40.526 mm				
	Tutor.DRL		Excellon		~	Tutor.DRL	*			
	Source		Target			Size/Format				
	✓ Tutor.DRL		Tutor.DRL		~	76.543 x 42.57 mm				
20 53 - 30 - 7.9 rem	16	lo.00,								
K Materi	al				Dis	card Import				Layout 🔉

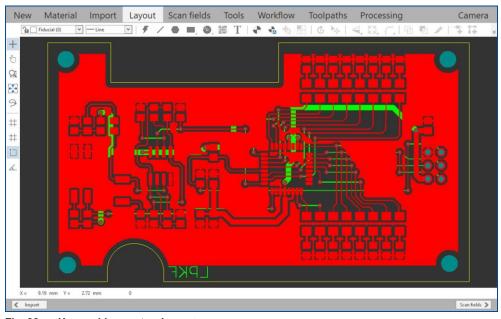
Fig. 80: User guidance step Import

5. Assign the holes to the layer DrillUnplated.

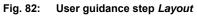
Tutor.BOA	GerberX	~	BoardApe	~	File Fi	ormat
Source	Target		Size/Format		Unit	Millimeters 🔗
✓ BoardOutline	BoardOutline	~	80.632 x 45.72 mm		Values	Absolute V
Tutor.BOT	GerberX	×	BotApe	~	Decimal	Omit leading zeros
Source	Target		Size/Format		Digits m.n	2 3
✓ GerberX data	BottomLayer	~	66.42 x 34.003 mm		orgio min	2 5
Tutor.TOP	GerberX	*	ТорАре	~		
Source	Target		Size/Format			
GerberX data	TopLayer	~	76.403 x 40.526 mm			
Tutor.DRL	Excellon	~	Tutor.DRL	~		
Source	Target		Size/Format			
✓ Tutor.DRL	DrillUnplated	~	76.543 x 42.57 mm			
Apertures/Tools Text	Tutor.DRL BoardOutline Cutinside DrillUnplated	Ŷ				
•	TextTop SilkScreenTop	÷	•			
<u>8</u> 9 😐						
1.89	27.41 52.92		78.44			

Fig. 81: Assigning holes to layer

- 6. Check in the preview of the layout whether the graphics and size (in the *Size/Format* column) are correct. If not, you can adjust the settings in the sub-tab *File*.
- 7. Click on [Import].



□ The user guidance step *Layout* is displayed:



The data have been imported.

Assigning the layers manually

If the data have not been assigned to the correct layers automatically, they can be assigned manually. In the drop-down list in the column *Target*, select which layer is to be assigned to the imported source data.

Tutor.BOA	GerberX	~	BoardApe	
Source	Target		iize/Format	
✓ BoardOutline	Tutor.BOA	×	0.632 x 45.72 mm	
Tutor.TOP	Tutor.BOA BoardOutline	^	ТорАре	[
Source	CutInside		iize/Format	
✓ GerberX data	DrillUnplated TopLayer TextTop SilkScreenTop	~	'6.403 x 40.526 mm	

Multiplying the layout (optional)

If desired, you can multiply the layout. In this example multiplies of the layout are not necessary.



The functions *Step and Repeat* and *Create instance type* are only available in the license level Advanced of CircuitPro PL. If you have any questions contact the LPKF sales department.

- 1. In the user guidance step *Layout* select the entire layout by pressing Ctrl + A.
- □ The layout is selected.
- 2. Right-click on the layout.
- □ The following context menu is displayed:

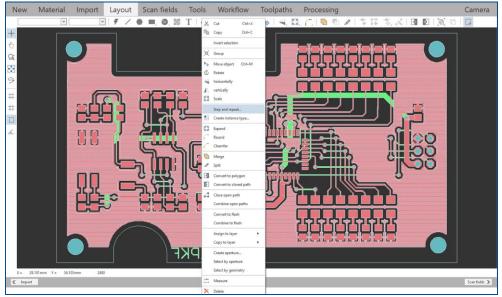


Fig. 83: Context menu Step and repeat

3. Click on Step and repeat...

Count	t x 1 y	1
• Ga	ap distance	
x	0.000 mm	v#
у	0.000 mm	_ ⊳ _× ∢_
0 0	ffset	~
x	80.632 mm	*
у	45.720 mm	y Maxel
		Preview
Co	mbine to flash l	ist
	OK	Cancel

The following dialog is displayed:

Fig. 84: Dialog Step and repeat

Enter the desired number of repetitions along each axis in the fields *Count*.
 For this example, four multiples of the layout have been made. Enter 2 counts for the x axis and 2 counts for the y axis.

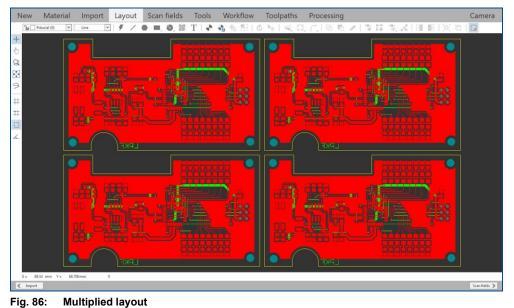
Some space for cutout is required between the copies, so the *Gap distance* has to be specified for the x and y direction.

- 5. Enter 2 mm in the fields Gap distance.
- After entering all the values, the dialog is displayed as follows:

Step and repeat
Count x 2 y 2
 Gap distance
x 2.000 mm y
y 2.000 mm
○ Offset
x 82.632 mm
y 47.720 mm
Preview
Combine to flash list
OK Cancel

Fig. 85: Dialog Step and repeat after entering values

- 6. Click on [OK].
- □ The multiplied layouts are created.
- 7. In order to zoom out and get an overview of the multiplied layouts, perform one of the following steps:
 - Scroll the mouse wheel.
 - Press the Home key.
 - Click on 🔂.
- 8. Press Esc or click anywhere on the black background to deselect the highlighted layout.
- □ The user guidance step *Layout* changes as follows:



The layout has been multiplied.

Creating an instance type

As an alternative to multiplying the layout data, you can use instances. The advantages are:

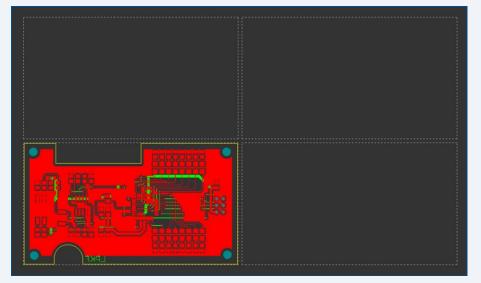
- Less memory is needed.
- The calculation of the toolpaths is speeded up.

You can create an instance type by performing the following steps:

- 1. Select the entire layout by pressing Ctrl + A.
- 2. Click on Insert > Create instance type or on E.
- □ The following dialog is displayed:

	Cr	eate instan	ce typ	e ×
Count	x	1	у [1
Distance	x	80.632 mm	у [45.72 mm
		OF	<	Cancel

- Enter the desired number of instances in the input fields *Count*.
 For this example, enter 2 counts for the x axis and 2 counts for the y axis.
- 4. Increase the entered values in the input fields *Distance* by 2 mm.
- 5. Click on [OK].
- The instance type has been created:

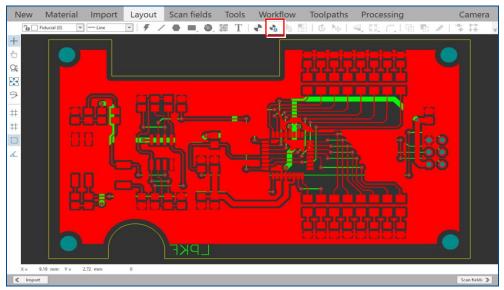


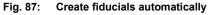
Creating fiducials

i

The function *Automatic fiducial creation* is only available in the license level Advanced of CircuitPro PL. If you have any questions contact the LPKF sales department.

 In the user guidance step Layout, click on Insert > Automatic fiducial creation or click on ♣a.





□ The following dialog is displayed:

creation.	to be created for the automati	c fiduciai
1 — 3 —	2	
Position method	Highest accuracy	~
Distance	4 mm	
Source layer	BoardOutline	~
Delete fiducials	None	~

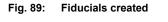
Fig. 88: Dialog Automatic fiducial creation

For optimum alignment results it is recommended to create four fiducials and to place them outside the outer contour.

You can prevent a faulty alignment of the base material with three fiducials. At least two fiducials are required for correct operation of the process; arrange them diagonally.

- Adapt the settings for the fiducials.
 In this example four fiducials with a distance of 4 mm from the *BoardOutline* are created.
- 3. Click on [OK].

- □ The fiducials are automatically added to the layout:



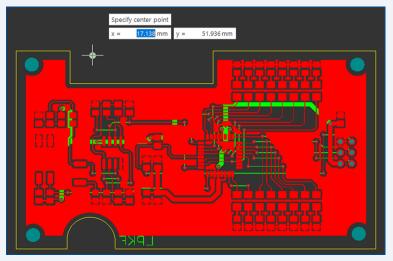
The fiducials have been created.



Creating fiducials manually

You can add fiducials to the layout manually by performing the following steps:

- 1. Click on *Insert* > *Fiducial* or click on \clubsuit .
- □ The input fields for *Specify center point* are displayed:



- 2. To insert the fiducial, perform one of the following steps:
 - Click on a desired point in the layout.
 - Enter the values for *x* and *y* in the input fields.
- 3. Repeat step 2 for all other fiducials.
- 4. Press Esc to close the function.

Computing toolpaths and scan fields automatically

- 1. Switch to the user guidance step Scan fields.
- Click on [Compute scan fields] or on

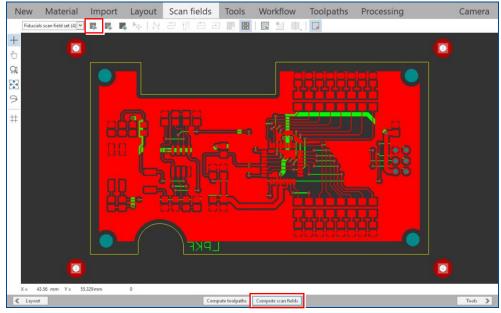


Fig. 90: Compute scan fields

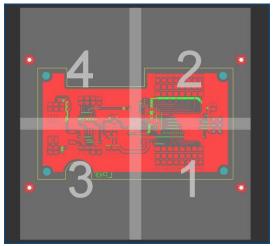
□ The following dialog is displayed:

Com	npu	te scan fields 🛛 🛛 🗙
Scan fields set		< <all field="" scan="" sets="">></all>
Size	x	50 mm y 50 mm
Safe margin	x	5 mm y 5 mm
Without overlap		
Overlap	x	5 mm y 5 mm
✔ Without top to bot	tom	scan field offset
Top to bottom offset	х	0 mm
		Compute Close

Fig. 91: Dialog Compute scan fields

- 3. Adapt the settings for the scan fields, if necessary. In this example, the default settings are used.
- 4. Click on [Compute].

□ The scan fields are computed automatically.





5. Click on [Compute toolpaths] or on

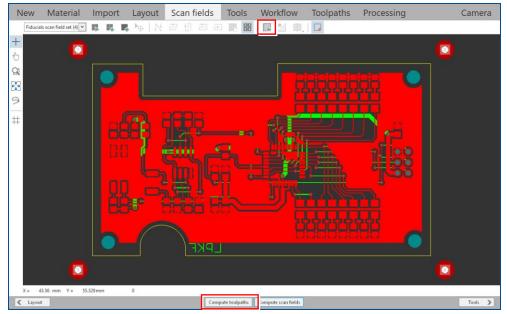


Fig. 93: Computing toolpaths

□ The following message is displayed:

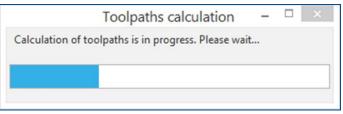


Fig. 94: Message Toolpaths calculation

□ The toolpaths are computed automatically with default settings. The following message is displayed:

Computation results	-		×
▲ () Used tools			
Rubout tools:			
Isolate (Laser 4566.9 mm, jumps 1043.1 mm)			
PreCut (Laser 42.7 mm, jumps 148.7 mm)			
Hatch (Laser 36829.4 mm, jumps 73280.4 mm)			
Divide Strip (Laser 19.2 mm, jumps 770.2 mm)			
Heat (Laser 35675.3 mm, jumps 69727.9 mm)			
Short Heat (Laser 1352.4 mm, jumps 17674.9 mm)			
Cutting tools:			
Cut Material (Laser 1662.4 mm, jumps 1.9 mm)			
Drilling tools:			
Drill Fiducials (4 pulses, jumps 0 mm)			
Drill Material (64 pulses, jumps 360 mm)			
		Clos	e

Fig. 95: Message Computation results

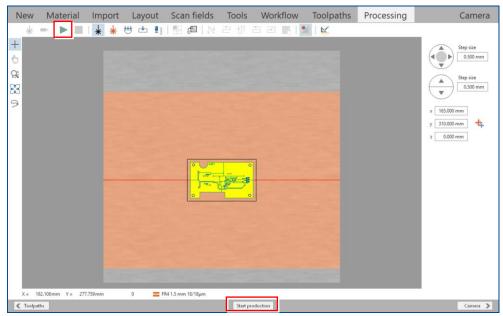
- 6. Check the computation results for any possible warnings or errors and make corrections, if required.
- 7. Click on [Close].

The toolpaths and scan fields have been computed.

Processing the PCB

Clean the material surface with LPKF Cleaner, if a discoloration (oxide layer) is discernible. Thus, a surface is achieved that always has the same characteristics.

1. Switch to the user guidance step Processing.



2. Click on b or click on [Start production].

□ The following message is displayed:

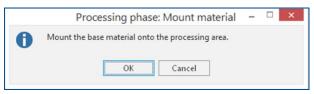


Fig. 97: Dialog Processing phase: Mount material

- 3. Open the cover.
- 4. Place the base material onto the processing table.
- 5. Click on 🐫.
- □ The base material is fastened onto the processing table by vacuum.

If the base material is bent too much and the vacuum does not allow to fasten it correctly, fasten it with adhesive tape.

- 6. Close the cover.
- 7. Click on [OK].
- The following dialog is displayed:

Material Processing data Click into the processing area to move the active head to the associated position. Use the buttons to set the left front and right rear corner of the material. x 000,5 mm x 000,5 mm y 018,0 mm y 018,0 mm y 247,0 mm Width 305 mm Length 229 mm Set default size Use a caliper to measure the material thickness Material thickness 1,536 mm	
position. Use the buttons to set the left front and right rear corner of the material. x 000,5 mm x 305,5 mm y 018,0 mm y 247,0 mm Width 305 mm Length 229 mm Set default size Use a caliper to measure the material thickness	
x 000,5 mm x 305,5 mm y 018,0 mm y 247,0 mm Width 305 mm Length 229 mm Set default size Use a caliper to measure the material thickness	
y 018,0 mm y 247,0 mm Width 305 mm Length 229 mm Set default size Use a caliper to measure the material thickness	
Width 305 mm Length 229 mm Set default size Use a caliper to measure the material thickness	
Use a caliper to measure the material thickness	
Material thickness 1536 mm Measure material thickness	
Material difectiess 1.550 mm	5
Apply OK Can	cel

Fig. 98: Dialog Placement

- 8. Move the dialog *Placement* to get a better overview.
- 9. Determine the processing area. The base material position and base material size must match the processing area used by the CircuitPro PL software.



For detailed information on project placement by determining the processing area, refer to chapter 1.1.

10. Click on the tab Processing data.

		Placement	-	×
Material	Processing data			
Drag and o	Irop the bounding	box of data in the proce	essing view.	
Center of p	processing data x	153 mm y	132.5 mm	
✓ Use info	rmation layers	Center on n	naterial	
Processing	head			
Cente	r on data			
Show bo	ounding box			

Fig. 99: Dialog Placement | Processing data

- 11. Place the processing data by performing one of the following steps:
 - Use drag & drop.
 - Enter the values in the fields *x* and *y*.
 - Click on [Center on material] to place the processing data on the center of the base material.
- 12. Click on [OK].
- The laser system starts the material thickness measurement. The following message is displayed:

Т	hickness measurement running
0	Thickness measurement running Point 1 of 3 Position (X = 175,5; Y = 155)
	Abort

Fig. 100: Message Thickness measurement running



For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the following message is displayed:

	Laser warm-up	phase in progress	s –		×
9:29 remai	ing in the current warm-	up phase			
		Skip o	ne warm	-up ph	nase

Fig. 101: Message Laser warm-up phase in progress

After warm-up, the fiducials and the through-holes are drilled and the following dialog is displayed:

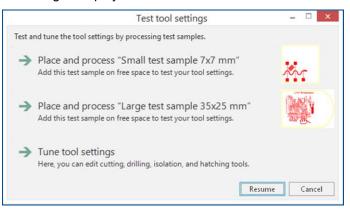


Fig. 102: Dialog Test tool settings

In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 13. Click on [Resume].
- □ The Bottom side (*BottomLayer*) is structured and the following message is displayed:

	Processing phase: Flip material 🛛 – 🗆 🗙
0	Turn the base material over around the machine's symmetry axis.
	Cancer

Fig. 103: Message Processing phase: Flip material

- 14. Turn the base material over around the symmetry axis of the system.
- 15. Click on [OK].
- □ The dialog *Placement* is displayed.
- 16. Place the processing data. The location of the layout **must match** the location of the PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 17. When project placement is complete, click on [OK].
- □ The laser system reads the fiducials on the Top side (*TopLayer*).



After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed. When finished, the dialog *Test tool settings* is displayed.

18. Click on [Resume].

□ The Top side (*TopLayer*) is structured and the following message is displayed:

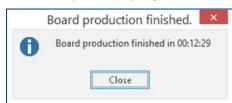


Fig. 104: Message Board production finished

- 19. Click on [Close].
- 20. Open the cover.
- 21. Remove the PCB from the system.
- The PCB has been processed.

Cleaning the system

The processing area has to be cleaned if heavily soiled.

- Use a vacuum cleaner to remove chips and residues from the processing area.
- The system has been cleaned.

Cleaning the PCB

- 1. Check for any remaining copper strips on the PCB that have not been removed by the laser.
- 2. Spray the PCB with LPKF Cleaner and use a brush to clean it.
- 3. Rinse the PCB with tap water and dry it with compressed air.



If the PCB is still not free of unwanted copper strips, apply a piece of adhesive tape that does not leave glue residues on the PCB and pull it off. Any remaining copper strips should be attached to the adhesive tape.

The PCB has been cleaned.

Residual copper strips

If despite all cleaning any copper strips still remain on the PCB, check the material and the tools with laser parameters for a specific material.

Modify the parameters of the tools so that there are no more copper strips on the PCB after processing. For detailed information on adjusting the tools, refer to chapter 1.5.

The PCB production is finished.

2.2 How to produce a double-sided PCB with galvanic through-hole plating

This chapter describes how to create a double-sided PCB using a laser system, a circuit board plotter and a galvanic through-hole plating system.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

Auxiliaries

System

- 1 set of tools for ProtoMat ProtoLaser U4/S4/R4
- LPKF Cleaner
 - ProtoMat S or E Contac S4

copper-plated with protection foil (order code:

SET-10-1053)

1.5 mm, 5/5 µm

Base material FR4,

229 mm × 305 mm ×

- Brush
- Oil-free compressed air

(order code: 115891)

Tap water

The following steps are performed in this tutorial:

- Switching on the ProtoMat
- Importing the data in CircuitPro PM
- Assigning holes to the layer DrillUnplated in CircuitPro PM
- Multiplying the layout in CircuitPro PM
- Creating fiducials in CircuitPro PM
- Saving the file in CircuitPro PM
- Generating toolpaths in CircuitPro PM
- Processing a PCB (with ProtoMat)
- Galvanic through-hole plating the PCB (with Contac S4)
- Preparing the data in CircuitPro PL
- Structuring the PCB (with ProtoLaser)
- Drilling unplated through holes and cutting out the PCB (with ProtoMat)
- Cleaning the PCB
- Cleaning the ProtoMat

Switching on the ProtoMat

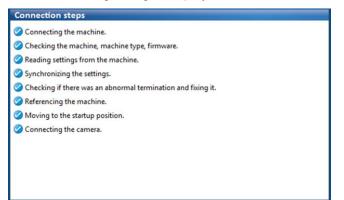
- 1. Press the on/off switch.
- □ The system is switched on.
- 2. Turn on the PC that is connected to the system.
- 3. Double-click on the desktop icon of CircuitPro PM.

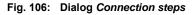


Fig. 105: Desktop icon CircuitPro PM

4. Wait for the system to connect and initialize.

□ The following dialog is displayed:





□ The following dialog is displayed:

CircuitPro	
Projects Templates	
itencil_size A4 for QR 266×380 frame.cbf	Stencil for original ProtoPrint frame.
M_PL_SingleSided_Top.cbf	PCB with one predefined layer on top side for use with ProtoMat and ProtoLaser system.
M_PL_SingleSided_Bottom.cbf	PCB with one predefined layer on bottom side for use with ProtoMat and ProtoLaser system.
M.D. DoubleSided NoTHD obf	BCB with production and bottom laws without through hole plating for use with ProtoMat and Protol associatem.
M_PL_DoubleSided_GalvanicTHP.cbf	PCB with predefined top and bottom layer prepared for galvanic through-hole plating for use with ProtoMat and ProtoLaser system.
M.P. Dubblided Proceeding of	
M_PL_DoubleSided_EasyContac.cbf	PCB with predefined top and bottom layer prepared for EasyContac through-hole plating for use with ProtoMat and ProtoLaser system.
M_PL_4Layer_GalvanicTHP_MultiPress.cbf	PCB with four predefined layers prepared for galvanic through-hole plating for use with ProtoMat and ProtoLaser system.
M_PL_4Layer_GalvanicTHP_MultiPressS.cbf	PCB with four predefined layers prepared for galvanic through-hole plating with MultiPress S for use with ProtoMat and ProtoLaser syste
M_PL_4Layer_GalvanicTHP_MultiPressS_Blind-Buried vias (ML104).cbf	PCB with four predefined layers prepared for galvanic through-hole plating with MultiPress S with blind and buried vias for use with Prot
M_PL_4Layer_ProConduct_MultiPress.cbf	PCB with four predefined layers prepared for ProConduct through-hole plating for use with ProtoMat and ProtoLaser system.
M_PL_4Layer_ProConduct_MultiPressS.cbf	PCB with four predefined layers prepared for ProConduct through-hole plating with MultipPress S for use with ProtoMat and ProtoLaser
M_PL_6Layer_GalvanicTHP_MultiPress.cbf	PCB with six predefined layers prepared for galvanic through-hole plating for use with ProtoMat and ProtoLaser system.
M_PL_6Layer_GalvanicTHP_MultiPressS.cbf	PCB with six predefined layers prepared for ProConduct through-hole plating with MultipPress S for use with ProtoMat and ProtoLaser s
M_PL_8Layer_GalvanicTHP_MultiPress.cbf	PCB with eight predefined layers prepared for galvanic through-hole plating for use with ProtoMat and ProtoLaser system.
M_PL_8Layer_GalvanicTHP_MultiPressS.cbf	PCB with eight predefined layers prepared for ProConduct through-hole plating with MultipPress S for use with ProtoMat and ProtoLase
44_PL_4Layer_GalvanicTHP_MultiPressS.cbf	PCB with four predefined layers prepared for galvanic through-hole plating with MultiPress S for use with ProtoMat E44 and ProtoLasers
AA DL AL suar DroConduct MultiDrareS chf	DCB with four oradefined Isver oreosed for DroConduct through hole olstion with MultiDreer S for use with on DrotoMat EM and Droto >>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>
Load custom template	
Set as default	Close

Fig. 107: Dialog New document

5. In the tab *Templates* select the template: *PM_PL_DoubleSided_GalvanicTHP.cbf*.

The ProtoMat has been switched on.

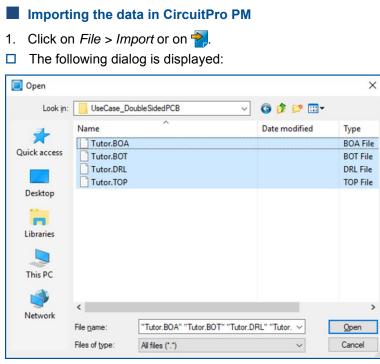


Fig. 108: Dialog Open

Navigate to the folder that contains the data you wish to import. For the example data used for this tutorial refer to the folder:

C:\Program Files (x86)\LPKF Laser & Electronics\ LPKF CircuitPro PM\Example Data\UseCase_DoubleSidedPCB.

- 3. Select the files you wish to import (in this example *Tutor.BOA, Tutor.BOT, Tutor.DRL*, and *Tutor.TOP*).
- 4. Click on [Open].
- □ The data should automatically be assigned to the correct layers and the following dialog is displayed:

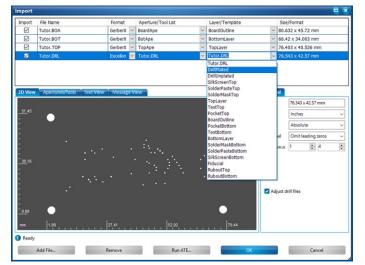


Fig. 109: Dialog Import | Assigned layers

5. Assign the holes to the layers *DrillPlated* or *DrillUnplated*, depending whether the holes are to be plated or not. In this example the option *DrillPlated* is used.



If there are no separate files for plated and unplated holes, they can be manually reassigned to the proper layer later (refer to step Assigning holes to the layer *DrillUnplated* in CircuitPro PM on page 87).

- 6. Look at the preview and check whether the graphics and size (in the *Size/Format* column) of the drilling file are correct. If not, you can adapt the settings in the sub-tab *General*.
- 7. Click on [OK].
- □ The CAM view changes as follows:

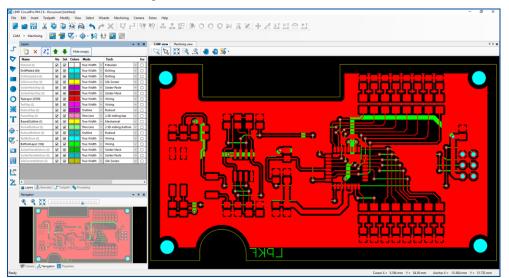


Fig. 110: Imported data in the CAM view

The data have been imported in CircuitPro PM.



Assigning the layers manually

If the data have not been assigned to the correct layers automatically, they can be assigned manually. In the drop-down list in the column *Layer/Template*, select which layer is to be assigned to the imported source data.

Assigning holes to the layer *DrillUnplated* in CircuitPro PM

The holes that do not have a copper ring (i.e. they are not directly surrounded by copper), as well as the holes that should not be galvanically plated, must be assigned to the *DrillUnplated* layer.

- 1. Press and hold the <u>Ctrl</u> key and click on all holes that are to be assigned to the layer **DrillUnplated**.
- □ The holes are highlighted in gray:

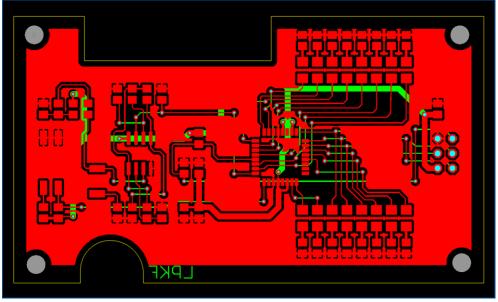


Fig. 111: CAM view after selecting holes

2. Right-click on the layout and select *DrillUnplated* in the context menu under *Assign objects to layer.*

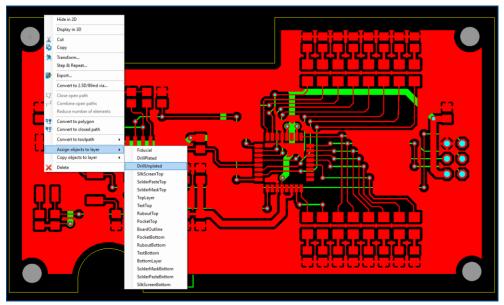


Fig. 112: Context menu Assign objects to DrillUnplated layer

□ An additional layer is created automatically.

The holes have been assigned to the layer *DrillUnplated* in CircuitPro PM.

Multiplying the layout in CircuitPro PM

The layout can be copied and placed freely for panel production depending on the size of the layout and the size of the base material.

- 1. Select the entire layout by pressing Ctrl + A.
- □ The layout is highlighted.
- 2. Right-click on the highlighted layout.
- □ The following context menu is displayed:

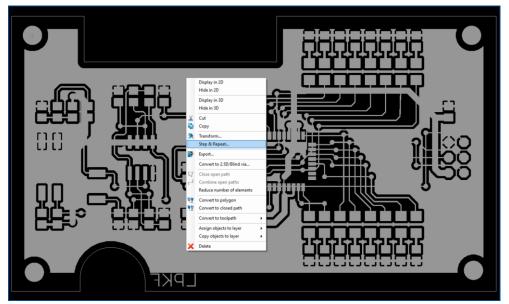


Fig. 113: Context menu Step & Repeat

- 3. Click on Step & Repeat...
- □ The following dialog is displayed:

Step & R	epe	at				- 🙁
Repetition	х	1	•	γ 1		•
Distance	х	80.632 mm	•	γ 45.72 mm	n	•
		Combine to flash, list				
		Арр	у		Close	

Fig. 114: Dialog Step & Repeat

4. Enter the desired repetitions along each axis in the *Repetition* fields. In this example the layer is multiplied by 4 exemplars. Enter **2** repetitions for the x axis and **2** repetitions for the y axis.



Some space for cutout is required between copies, so the **distance** should be **increased** in both x and y direction.

5. Increase the values in the *Distance* fields by 5 mm, if you are using a contour routing tool with 2 mm.

□ After entering all values, the dialog *Step & Repeat* changes as follows:

Repetition	Х	2	•	Y	2		\$
Distance	х	85.632 mm	▲ ▼	Y	50.72 mm		÷
		Com	bine to flash, list:				
		_	Analy			Close	

Fig. 115: Dialog Step & Repeat after entering values

- 6. Click on [Apply].
- 7. Click on [Close].
- 8. In order to zoom out and get an overview of the multiplied projects, perform one of the following steps:
 - Scroll the mouse wheel.
 - Press the key Home.
 - Click on 🔀.

□ The CAM view changes as follows:

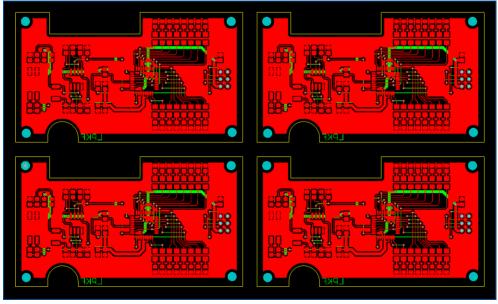


Fig. 116: CAM view of the multiplied layout

The layout has been multiplied in CircuitPro PM.

Creating fiducials in CircuitPro PM

- 1. Click on Insert > Fiducial > Fiducial... or on 🕀 and Fiducial...
- □ The following dialog is displayed:

Layer:	Fic	lucial			~		
	0	Absolute OR	elative to anchor	point			
Center:	X:	0 mm	₽ Υ:	0 mm	ø:	1.5 mm	÷

Fig. 117: Dialog Create fiducial

- 2. Perform one of the following steps:
 - Click on the positions in the layout where you want to place the fiducials.
 - Create the fiducials by entering the x and y position in the dialog.



For optimum alignment results it is recommended to create four fiducials and to place them outside the outer contour.

Using three fiducials is a good way to avoid wrong orientation of the base material. At least two fiducials are required for correct operation of the process; arrange them diagonally.

- 3. Click on [Close] or press [Esc].
- □ The CAM view changes as follows:

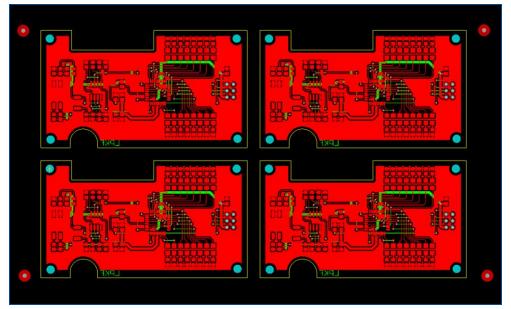


Fig. 118: CAM view after creating fiducials

The fiducials have been created in CircuitPro PM.

Saving the file in CircuitPro PM

- 1. Click on File > Save As...
- 2. Select a suitable folder, name the file and click on [Save].
- The file has been saved in CircuitPro PM.

Generating toolpaths in CircuitPro PM

- 1. Click on *Toolpath* > *Technology Dialog…* or on \mathbb{N} .
- □ The dialog *Technology Dialog* is displayed.

Technolog	y Dialog							×
Global process se	ettings							
Material type	FR4		~ Copper	layer thickness	18 µm	\$	RF application	
Isolate Contor	or routing Drills Fiducia	lls	Pockets Blin Isolation Me Basic Description	d vias				
Prov	<wiring> layers</wiring>	~	Shortest p	0.2 mm	\$		channels	
Primary Available tools	Universal Cutter 0,2 mm	~	Pads isolation	<no rubout?<="" td=""><td></td><td></td><td>centric</td><td>~</td></no>			centric	~
Avanable tools	ProtoVed Laser Custom	~	Tolerance Generate opt Force isolatic Perform inne	0.002 mm timized rubout	Replace existi Remove spike Design rule cl	ing too		~
							Start Close	

Fig. 119: Tab Isolate

2. In the tab *Contour routing* use the arrow buttons to select the entry *Edge gaps*. This is usually the preferred method of contour routing.

Technology Dialog			×
Global process settings			_
Material type FR4	~ Co	opper layer thickness 18 µm 💼 🔲 RF application	
Isolate Contour routing Drills Fiducia		Blind Vias	
		e gaps]
	Con	iption tour Routing with one gap on each edge.	1
4/6			
			-
O Inside	Source	<mechanical> layers ~</mechanical>	
Outside	Tool	Contour Router 2 mm v	
	Tabs position	All sides ~	
	Gap width	1 mm 🔹	
	Distance	50 mm	
	Tolerance	0.002 mm	
		Replace existing toolpath	
		Cenerate start drills	
		Start Close	

Fig. 120: Tab Contour routing

3. Click on the tab *Pockets* and deactivate the check box *Process*, since there are no pockets in this project.

1 Technolo	gy Dialog						8
Global process	settings —						
Material type	FR4	~	Copper layer thickness	18 µm	•	RF application	
Isolate Cont	our routing	Drills Fiducials Pocket	Blind vias				
Process	Tolerance	□ End Mill (RF) 0,15 mm □ End Mill (RF) 0,25 mm ⊡ End Mill (RF) 0,4 mm ☑ End Mill 0,8 mm 0.002 mm existing toolpath					
					Start	Close	

Fig. 121: Tab Pockets

- 4. Click on [Start].
- The software generates all toolpaths and identifies all required tools. A report of the required tools is displayed:

A Warnings	Class
 Drilling 	Close
Attention, no marking drills on layers:	
DrillUnplated	Save
Check availability of the corresponding marking	Print
drill phase and its processing order.	· ····
Required Tools	
Drilling Tools:	
1 x Spiral Drill 2 mm(44 strokes)	
1 x Spiral Drill 0,4 mm(371 strokes)	
1 x Spiral Drill 0,6 mm(7 strokes)	
1 x Spiral Drill 1 mm(42 strokes)	
1 x Spiral Drill 1,5 mm(4 strokes)	
Contour Router:	
1 x Contour Router 2 mm (1190.8 mm)	
Conical Tools:	
1 x Universal Cutter 0,2 mm (420.0 mm)	
Show more	

Fig. 122: Message Computation Results

- 5. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 6. Click on [Close].
- 7. Click on *File* > *Save* or on **=** to save the changes.
- The toolpaths have been generated in CircuitPro PM.



For detailed information on contour routing types and settings refer to the compendium of CircuitPro PM.

Processing a PCB (with ProtoMat)

Milling phases will be skipped, as this will be done by the ProtoLaser.

- 1. Load the tool magazine and assign the tools to positions.
- 2. Click on *Machining* > *Process all* or in the pane *Processing* on .
- 3. Perform the following phases:
 - MountMaterial
 - MaterialSettings
 - Placement
- □ The phases *DrillFiducial*, *MarkingDrills* and *DrillingPlated* are performed.
- 4. Remove the PCB from the system.
- 5. Rinse the PCB with tap water and dry it using compressed air.
- 6. When the message *Processing Phase: ThroughHolePlating* is displayed, remove the PCB from the system and click on [Cancel] to temporarily stop the process.
- The PCB has been processed.



For detailed information on the ProtoMat phases refer to the how-to guides of **ProtoMat**.

Galvanic through-hole plating the PCB (with Contac S4)

- 1. Switch on the system.
- 2. Select a profile.



The recommended total copper thickness after through-hole plating is 30 μm to 35 $\mu m.$

- 3. Start the process.
- 4. Prepare the PCB for through-hole plating.
- 5. Clean the PCB.
- 6. Condition the PCB.
- 7. Activate the PCB.
- 8. Clean the holes with LPKF ViaCleaner.
- 9. Copper-plate the PCB.
- 10. Switch off the system.
- The PCB has been galvanically through-hole plated.



For detailed information on galvanic through-hole plating, refer to the user manual of **Contac S4**.

i

Tips for more efficient galvanic through-hole plating

- In case of oxidation, clean it using a soft brush and LPKF Cleaner.
- Use compressed air to remove drilling debris from the through holes.
- Use compressed air to remove water from the through holes before activation in tank 3.
- Turn the board over at half of plating time in order to achieve equal copper deposition on both sides.

Preparing the data in CircuitPro PL

For detailed information on preparing the data with CircuitPro PL refer to chapter 2.1.

- 1. Switch to the user guidance step New.
- 2. Click on [Open project...].

New	Material	Import	Layout	Scan fields	Tools	Workflo	w Toolpaths	Processing	Camera
Recent	projects		All materials	All machine type	s 💌	Supported by Pr			_
Recent	template		Single-sided Double-side Double-side Double-side Double-side Layers, Pro 4 Layers, Pro 6 Layers, Pro 6 Layers, Pro 8 Layers, Pro 9 Layers, Pro 8 Layers, Pro 9 Layers, Pro	Top, ProtoLasser S4 Bettom, ProtoLasser S4 A protoLasser S4, and hnou A protoLasser S4, anayoni A protoLasser S4, anayoni A protoLasser S4, BroConduct, M toLasser S4, BroConduct, M Bettom, ProtoLasser U4, A ProtoLasser U4, BroConduct, M C ProtoLasser U4, BroConduct, M C ProtoLasser U4, BroConduct, B RotoLasser U4, B RotoLass	hrough-hole plating ac oct gh-hole plating Jh-hole plating Jh-hole plating gh-hole plating	tting MultiPress S MultiPress S MultiPress S 4	PCB with Top layer, prepare	d for ProtoLaser S4	
		_	_		Instead				
					Load t	template			Material 🔰

Fig. 123: Open project

- □ The dialog *Open* is displayed.
- 3. Navigate to the folder that contains the file you previously saved in CircuitPro PM.
- 4. Select CBF document (*.cbf) from the drop-down list.
- 5. Select the appropriate .cbf file and click on [Open].

Upgrade project The file C:\Users\User name\Desktop\Double-sided PCB.cbf contains old format data and requires to be upgraded. Choose a template for the upgrade. A ✓ All machine types ✓ Supported by ProtoMat All materials LPKF templates Single-sided, Top, ProtoLaser S4 Single-sided, Bottom, ProtoLaser S4 Double-sided, ProtoLaser S4, no through-hole plating Double-sided, ProtoLaser S4, galvanic through-hole plating Double-sided, ProtoLaser S4, EasyContac Double-sided, ProtoLaser S4, ProConduct 4 Layers, ProtoLaser S4, galvanic through-hole plating 4 Layers, ProtoLaser S4, galvanic through-hole plating, MultiPress S 4 Layers, ProtoLaser S4, ProConduct, MultiPress 4 Layers, ProtoLaser S4, ProConduct, MultiPress S 6 Layers, ProtoLaser S4, galvanic through-hole plating 6 Layers, ProtoLaser S4, galvanic through-hole plating, MultiPress S 8 Layers, ProtoLaser S4, galvanic through-hole plating 8 Layers, ProtoLaser S4, galvanic through-hole plating, MultiPress S Stencil for ProtoPrint, A4 for QR 266 x 380, ProtoLaser S4 ProMask_Removal, ProtoLaser S4 Single-sided, Top, ProtoLaser U4 Single-sided, Bottom, ProtoLaser U4 Double-sided, ProtoLaser U4, no through-hole plating Double-sided, ProtoLaser U4, galvanic through-hole plating Ignore fiducial and information layers Close Upgrade

□ The following dialog is displayed:

Fig. 124: Dialog Upgrade project

6. Perform the following steps:

- Select Double-sided from the drop-down list.
- Select your laser system from the drop-down list (in this example PL S4).
- Activate the check box Supported by ProtoMat.
- Select the template Double-sided, ProtoLaser S4, ProtoMat, galvanic through-hole plating.

The view changes as follows:

Upgrade project ×
The file C:\Users\User name\Desktop\Double-sided PCB.cbf contains old format data and requires to be upgraded. Choose a template for the upgrade.
Double-sided V PL S4 V Supported by ProtoMat
LPKF templates
Double-sided, ProtoLaser S4, ProtoMat, galvanic through-hole plating Double-sided, ProtoLaser S4, ProtoMat, EasyContac Double-sided, ProtoLaser S4, ProtoMat, ProConduct
Ignore fiducial and information layers Upgrade Close

Fig. 125: Upgrade selected

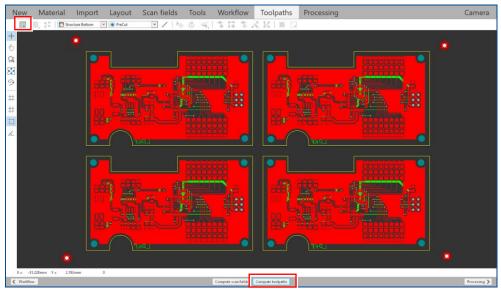
7. Click on [Upgrade] or double click on the template.

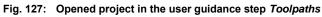
- □ The user guidance step *Material* is displayed.
- 8. Select the material *FR4 Double-sided*, 1.5 mm, 5/5 μm.

FR4 Double-sided, 1 mm, 18/18µm	6	^
FR4 Double-sided, 1.5 mm, 18/18µm	â	
FR4 Double-sided, 1.5 mm, 35/35µm	â	
FR4 Double-sided, 1,5 mm, 5/5µm	6	
FR4 Nanya Double-sided, 1.5 mm, 35/35µm	â	
IS400 Double-sided, 0.18 mm, 18/18µm	â	
IS400 Double-sided, 0.46 mm, 18/18µm	â	
ML104 Double-sided, 0.36 mm, 18/18µm	â	
PyraluxTK Double-sided, 0.12 mm, 12/12µm	â	
RO3003C Double-sided, 0.51 mm, 18/18µm	â	
RO3003C Double-sided, 0.51 mm, 35/35µm	â	
RO3006 Double-sided, 0,64 mm, 35/35µm	6	

Fig. 126: Material selection

- 9. Click on [Select material] or double-click on the material.
- □ The opened project is displayed in the user guidance step *Layout*.
- 10. Switch to the user guidance step *Toolpaths*.





□ The following dialog is displayed:

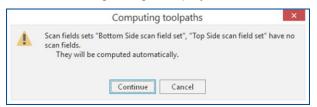


Fig. 128: Dialog Computing toolpaths

- 12. Click on [Continue].
- □ The message *Computation results* is displayed.
- Check the computation results for any possible warnings or errors and make corrections, if needed.
- 14. Click on [Close].
- 15. Click on *File* > Save or on \square .
- 16. Select a suitable folder, name the file and click on [Save].
- □ The file is saved in the .cp2d file format.
- The data have been prepared in CircuitPro PL.

Structuring the PCB (with ProtoLaser)



For detailed information on structuring the PCB with the ProtoLaser refer to chapter 2.1.

- 1. Switch to the user guidance step Processing.
- Click on local or click on [Start production].
- □ The following message is displayed:

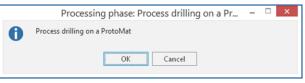


Fig. 129: Message Processing phase: Process drilling on a ProtoMat

- 3. Click on [OK].
- □ The following message is displayed:

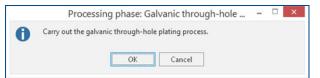


Fig. 130: Message Processing phase: Galvanic through-hole plating

- 4. Click on [OK].
- □ The following message is displayed:



Fig. 131: Message Processing phase: Mount material bottom

5. Open the cover.

- 6. Place the base material with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.
- 7. Click on 👑.
- □ The base material is fastened onto the processing table by vacuum.
- 8. Close the cover.
- 9. Click on [OK].
- □ The following dialog is displayed:

Copper th	nickness	-	□ ×
Enter the copper thickness:	5 µm]	
Г	OK		Cancel

Fig. 132: Dialog Copper thickness

10. Enter the total copper thickness after galvanic through-hole plating in the input field.



You can determine the total copper thickness by performing one of the following steps:

- Calculate the copper thickness from the copper deposition rate in Contac S4 (approx. 0.15 µm/min).
- Measure the copper thickness with the ProtoMat S104.
- 11. Click on [OK].
- □ The following dialog is displayed:

Placement	-		×
Material Processing data			
Click into the processing area to move the active head to the as position. Use the buttons to set the left front and right rear com material.			
x 000.5 mm x 305.5 mm			
y 018.0 mm y 247.0 mm			
Width 305 mm Length 229 mm Set default s	ize		
Use a caliper gauge to measure the material thickness.			
Material thickness 1.56 mm Measure material th	nickn	ess	
Аррју ОК		Can	cel

Fig. 133: Dialog Placement

- 12. Move the dialog *Placement* to get a better overview.
- 13. Click on the tab Processing data.
- 14. Place the processing data. The location of the layout **must match** the location of the PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

15. After project placement is complete click on [OK].

□ The laser system reads the fiducials on the Bottom side (*BottomLayer*).

For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the dialog Test tool settings is displayed.

In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 16. Click on [Resume].
- □ The Bottom side (*BottomLayer*) is structured and the following message is displayed:

	Processing phase: Flip material 🛛 – 🗖 🗙
0	Turn the base material over around the machine's symmetry axis.

Fig. 134: Message Processing phase: Flip material

- 17. Turn the base material over around the symmetry axis of the system.
- 18. Click on [OK].
- □ The dialog *Placement* is displayed.
- 19. Place the processing data **matching** the location of the PCB and fiducials on the processing table.
- 20. When project placement is complete, click on [OK].
- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 21. Click on [Resume].
- □ Structuring the Top side (*TopLayer*) is started.

When the message *Processing phase: Process drilling on a ProtoMat* is displayed, remove the PCB and proceed with drilling and cutting on the ProtoMat.

The PCB has been structured.

Drilling unplated through holes and cutting out the PCB (with ProtoMat)

- 1. In CircuitPro PM click on File > Open or on _____.
- □ The following dialog is displayed:

Open				×
Look in:	UseCase_D	loubleSidedPCB ~	G 🗊 📂 🛄 -	
4	Name	^	Date modified	Туре
Quick access	Double-sid	led PCB.cbf		CircuitPro
Desktop				
Libraries				
This PC				
1	<			>
Network	File <u>n</u> ame:	Double-sided PCB.cbf	~	<u>O</u> pen
	Files of type:	CBF document (*.cbf)	~	Cancel

Fig. 135: Dialog Open

- 2. Navigate to the folder that contains the file you previously saved in CircuitPro PM.
- 3. Select the *.cbf file and click on [Open].
- □ Your project is displayed in the *CAM view*.

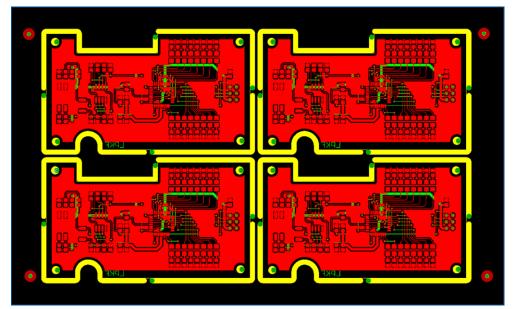
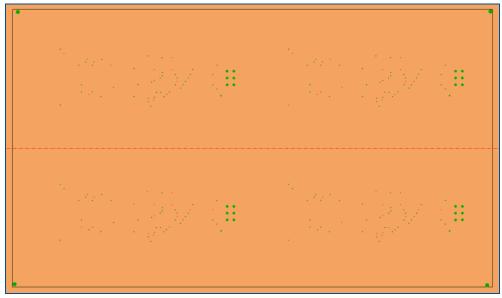


Fig. 136: CAM view of the opened project

4. Switch to the Machining view.



□ Your project is displayed in the *Machining view*:



5. In the pane Processing, select MountMaterialTop in the drop-down list.

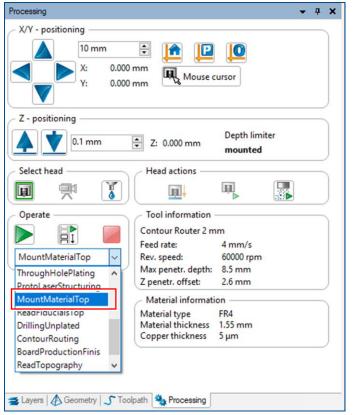


Fig. 138: Drop-down list of the processing phases

□ The *Machining view* displays the drilling and contour-routing data:

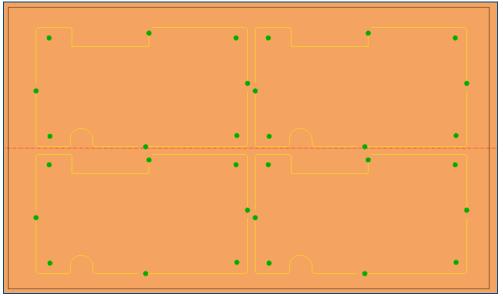


Fig. 139: *Machining view* of drilling and contour-routing data

- 6. Click on $\exists r_1$.
- □ The message *Processing Phase: MountMaterialTop* is displayed.
- 7. Place the board onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** and fasten it using adhesive tape.
- 8. Perform the following phases:
 - MaterialSettings_1
 - Placement_1
- □ The phase *ReadFiducialsTop* is performed.



For detailed information on fiducial recognition refer to the how-to guides of **ProtoMat**.

- □ The phases *DrillingUnplated* and *ContourRouting* are performed.
- The unplated through holes have been drilled and the PCB has been cut out.

Cleaning the PCB

- 1. Remove the PCB from the system.
- 2. Spray the PCB with LPKF Cleaner and use a brush to clean it.
- 3. Rinse the PCB with tap water and dry it with compressed air.
- 4. Break or cut the breakout tabs.
- The PCB has been cleaned.

The PCB production is finished.

Cleaning the ProtoMat

The processing area has to be cleaned if heavily soiled.

- Use a brush and a dry antistatic cloth to remove residues from the processing table.
- Use a vacuum cleaner to remove residues from the interior.
- The ProtoMat has been cleaned.

2.3 How to produce a double-sided PCB with non-galvanic through-hole plating

This chapter describes how to create a double-sided PCB using a circuit board plotter, a laser system and a non-galvanic through-hole plating system.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

1 set of tools for ProtoMat •

Consumables

Auxiliaries

System

- Base material FR4, 229 mm × 305 mm × 1.5 mm, 18/18 µm copper-plated (order code: 115967)
- Convection oven
 (order code: 115877)
- ProtoMat S or E
 ProConduct

ProtoLaser U4/S4/R4

- LPKF Cleaner
 (order code: 115891)
- Brush
- Oil-free compressed air
- Tap water

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PM
- Drilling fiducials (with ProtoMat)
- Preparing the data in CircuitPro PL
- Structuring the PCB (with ProtoLaser)
- Drilling unplated through holes and cutting out the PCB (with ProtoMat)
- Drilling plated through holes (with ProtoMat)
- Plating through holes (with ProConduct)

Preparing the data in CircuitPro PM



For detailed information on preparing the data in CircuitPro PM refer to chapter 2.2.

- 1. In the tab *Templates* of the dialog *New document*, select the template: *PM_PL_DoubleSided_ProConduct.cbf*.
- 2. Click on *File* > *Import* or on 🔁
- 3. Navigate to the folder that contains the data you wish to import. For the example data used for this tutorial refer to the folder: C:\Program Files (x86)\LPKF Laser & Electronics\ LPKF CircuitPro PM\Example Data\UseCase DoubleSidedPCB.
- 4. Select the files you wish to import (in this example *Tutor.BOA, Tutor.BOT, Tutor.DRL* and *Tutor.TOP*).
- 5. Click on [Open].
- □ The data are automatically assigned to the corresponding layers.
- 6. Correct the layer assignments, if necessary.
- 7. Click on [OK].
- 8. You can make multiples of the layout, if desired.
- 9. Add fiducials to the layout.
- 10. Click on *Toolpath* > *Technology Dialog* or on \mathbb{N} .
- □ The dialog *Technology Dialog* is displayed.
- 11. Click on [Start].

- □ The computation results are displayed in a dialog.
- 12. Click on [Close].
- 13. Save the file.
- The data have been prepared in CircuitPro PM.

Drilling fiducials (with ProtoMat)

- 1. Load the tool magazine and assign the tools to positions.
- 2. Click on Machining > Process all or on .
- 3. Perform the following phases:
 - MountMaterial
 - MaterialSettings
 - Placement
- □ The *DrillFiducial* phase is performed.
- 4. When the message *Processing Phase: ProtoLaserStructuring* is displayed, remove the PCB from the system and click on [Cancel] to temporarily stop the process.
- The fiducials have been drilled.



For detailed information on the ProtoMat phases refer to the how-to guides of **ProtoMat**.

Preparing the data in CircuitPro PL



For detailed information on preparing the data in CircuitPro PL refer to chapter 2.2.

- 1. In the user guidance step New click on [Open project...].
- □ The dialog *Open* is displayed.
- 2. Navigate to the folder that contains the file you previously saved in CircuitPro PM.
- 3. Set the file type to CBF document (*.cbf).
- 4. Select the appropriate .cbf file and click on [Open].
- □ The dialog *Upgrade project* is displayed.
- 5. Perform the following steps:
 - Select Double-sided from the drop-down list.
 - Select your laser system from the drop-down list (in this example PL U4).
 - Activate the check box Supported by ProtoMat.
 - Select the template Double-sided, ProtoLaser U4, ProtoMat, ProConduct.

		Upgrade	project	×
		Opgrade	project	
The file C:\Users\User name\Desktop\Double-sided PCB.cbf contains old format data and requires to be upgraded. Choose a template for the upgrade.				
Double-side	d	PL U4	~	✓ Supported by ProtoMat
LPKF templa	ites			
Double	-sided, ProtoLas	er U4, ProtoMat, no t	hrough-hole pla	ting
Double-sided, ProtoLaser U4, ProtoMat, galvanic through-hole plating				
Double	-sided, ProtoLas	er U4, ProtoMat, Easy	/Contac	
Double	-sided, ProtoLas	er U4, ProtoMat, Pro	Conduct	
🗌 Ignore	fiducial and inf	ormation layers		
		Upgra	de	Close

□ The view changes as follows:

Fig. 140: Upgrade selected

- 6. Click on [Upgrade] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 7. Select the material *FR4 Double-sided*, *1.5 mm*, *18/18 μm*.
- 8. Click on [Select material] or double-click on the material.
- □ The opened project is displayed in the user guidance step *Layout*.
- 9. Switch to the user guidance step *Toolpaths*.
- 10. Click on [Compute toolpaths] or on **[**].
- □ The dialog *Computing toolpaths* is displayed.
- 11. Click on [Continue].
- □ The message *Computation results* is displayed.
- 12. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 13. Click on [Close].
- 14. Save your project.
- The data have been prepared in CircuitPro PL.

Structuring the PCB (with ProtoLaser)



For detailed information on structuring the PCB with the ProtoLaser refer to chapter 2.2.

- 1. Switch to the user guidance step *Processing*.
- 2. Click on b or click on [Start production].
- When the message Processing phase: Process drilling on a ProtoMat is displayed, click on [OK].
- 4. When the message *Processing phase: Mount material bottom* is displayed, place the base material with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.
- 5. Click on 😬.
- □ The base material is fastened onto the processing table by vacuum.
- 6. Click on [OK].
- □ The dialog *Placement* is displayed.
- 7. Move the dialog *Placement* to get a better overview.
- 8. Click on the tab *Processing data*.
- 9. Place the processing data. The location of the layout **must match** the location of the PCB and fiducials on the processing table.
- For detailed information on project placement according to fiducial positions, refer to chapter 1.1.
- 10. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Bottom side (*BottomLayer*).
- For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.



When finished, the dialog *Test tool settings* is displayed.



In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 11. Click on [Resume].
- Structuring the Bottom side (*BottomLayer*) is started.
- 12. When the message *Processing phase: Flip material* is displayed, turn the PCB over around the symmetry axis of the system and click on [OK].
- □ The dialog *Placement* is displayed.
- 13. Place the processing data **matching** the location of the PCB and fiducials on the processing table.
- 14. When project placement is complete, click on [OK].

- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 15. Click on [Resume].
- 16. Structuring the Top side (*TopLayer*) is started.
- □ When the message *Processing phase: Process drilling on a ProtoMat* is displayed, remove the PCB and proceed with drilling and cutting on the ProtoMat.
- The PCB has been structured.

Drilling unplated through holes and cutting out the PCB (with ProtoMat)

For detailed information on drilling unplated through holes and cutting out the PCB with ProtoMat refer to chapter 2.2.

- 1. Open the *.cbf file you previously saved in CircuitPro PM.
- 2. Switch to the pane *Processing*.
- 3. Select *MountMaterialTop* from the drop-down list and click on **I**.
- 4. Place the board onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** and fasten it using adhesive tape.
- 5. Perform the following phases:
 - MaterialSettings_1
 - Placement_1
- □ The phases *ReadFiducialsTop*, *MarkingDrills*, *DrillingUnplated* and *ContourRouting* are performed.
- 6. When the message *Processing Phase: DismountMaterial* is displayed, remove the PCB from the system and click on [OK].
- 7. Rinse the PCB with tap water and dry it using compressed air.
- The unplated through holes have been drilled and the PCB has been cut out.



The PCB **must remain attached** to the original material. Do not break or cut the breakout tabs yet.

Drilling plated through holes (with ProtoMat)

- 1. When the message *Processing Phase: ApplyProtectionFilm* is displayed, apply the protective film onto both sides of the board and click on [OK].
- 2. When the message *Processing Phase: MountMaterialTop_1* is displayed, place the board with the **Top side** (*TopLayer*) **facing upwards** onto the system's processing table and click on [OK].
- 3. Perform the phase Placement_2.
- □ The phases *ReadFiducialsTop_1* and *DrillingPlated* are performed.
- 4. When the message *Processing Phase: DismountMaterial_1* is displayed, remove the board from the system and check it for burrs and blockages.



Check whether the fiducials are covered by the protective film and expose them if necessary.

The plated through holes have been drilled.

Plating through holes (with ProConduct)

- 1. Place the mat onto the vacuum table (ProtoMat mounted or standalone).
- 2. Place the board onto the mat.
- 3. Apply the ProConduct paste along the edge of the PCB.
- 4. Spread the paste on the whole surface of the PCB (do not spread over the fiducials).
- 5. Turn on the extraction system on the highest setting and wait for at least 30 seconds.
- 6. Spread the paste remaining on the PCB surface with the vacuum turned on.
- 7. Switch off the extraction system.
- 8. Turn the board over and repeat the steps 2 to 7 on the other side.
- 9. Slowly peel off the protective film at an angle of 90° from both sides of the board.
- 10. Place the board horizontally into the convection oven for 30 minutes (160 $^\circ\text{C}$ / 320 $^\circ\text{F}$).
- 11. Remove the board and let it cool down to ambient temperature.
- 12. Spray the board with LPKF Cleaner and use a brush to clean the PCB.
- 13. Rinse the board with tap water and dry it with compressed air.
- 14. Cut out or break out the PCB from the board.
- ✓ The through holes have been plated.



For detailed information on non-galvanic through-hole plating refer to the process description of **ProConduct**.

The PCB production is finished.

System

ProtoLaser R4

Contac S4

2.4 Producing a double-sided flexible PCB with galvanic through-hole plating

This chapter describes how to create a double-sided flexible PCB using a laser system and a galvanic through-hole plating system.



If you do not want to perform the through-hole plating:

- Select the following template in the system software: Double-sided, ProtoLaser R4, no through-hole plating.
- Skip the step Galvanic through-hole plating the flexible PCB.

The processing order changes slightly.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

Auxiliaries

1 × Pyralux® CG185018E

LPKF Cleaner (order code: 115891)

- Hand-held microscope
- Brush

•

- Tap water
- PCB holder for flexible PCBs (order code: 10067533)

The following steps are performed in this tutorial:

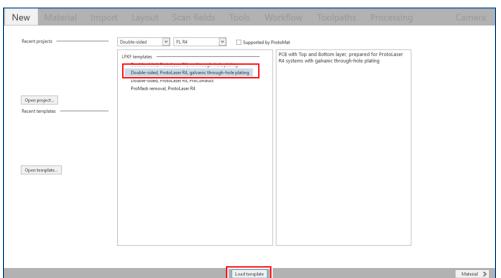
- Prepairing the data in CircuitPro PL
- Computing toolpaths and scan fields
- Preparing the flexible material
- Galvanic through-hole plating the flexible PCB (with Contac S4)
- Structuring the flexible PCB
- Cleaning the flexible PCB

Prepairing the data in CircuitPro PL



For detailed information on preparing the data in CircuitPro PL refer to chapter 2.1.

- 1. In the user guidance step New, select Double-sided from the drop-down list.
- 2. Select your laser system from the drop-down list (in this example PL R4).



□ A list of templates for double-sided materials is displayed:

Fig. 141: List of templates

- 3. Select the template Double-sided, ProtoLaser R4, galvanic through-hole plating.
- 4. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 5. Select the material Pyralux CG185018E.

FR4 Double-sided, 1 mm, 18/18μm	6
FR4 Double-sided, 1,5 mm, 18/18µm	â
FR4 Double-sided, 1,5 mm, 35/35µm	â
FR4 Double-sided, 1,5 mm, 5/5µm	ô
ML104 Double-sided, 0,36 mm, 18/18μm	6
PET + Cu Double-sided, 0,1 mm, 35/35µm	â
Pyralux AP7156E Double-sided, 0,05 mm, 14/14µm	6
Pyralux AP8565R Double-sided, 0,15 mm, 18/18μm	6
Pyralux CG185018E Double-sided, 0,05 mm, 18/18μm	8
Pyralux TK1810018R Double-sided, 0,1 mm, 18/18µm	â
ProMask removal 2 layer multilayer	ô

Fig. 142: Material selection

- 6. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 7. Click on 📑

- □ The dialog *Open* is displayed.
- 8. Navigate to the folder that contains the data you want to import. For the example data that are used for this tutorial, refer to the folder: C:\Program Files\LPKF Laser & Electronics AG\ LPKF CircuitPro PL\Example Data\UseCase DoubleSidedPCB.
- 9. Select the desired files for import (in this example *Tutor.BOA, Tutor.BOT, Tutor.DRL and Tutor.TOP*).
- 10. Click on [Open].
- □ The data are automatically assigned to the correct layers in the user guidance step *Import*.

New	Material	Import	Layout	Scan fields	Tools	Workflow	То	olpaths	Processing	Camera
[*	Tutor.BOA		GerberX	~	BoardApe		~	File		7
(⁻	Source		Target		Size/Format			Unit	Inches 🗸	
	GerberX data		BoardOutline	×	80.632 x 45.72	mm		Values	Absolute ¥	
足	Tutor.BOT		GerberX	×	BotApe		~	Decimal	Omit leading zeros	
	Source		Target		Size/Format			Digits m.n	1 4	
	GerberX data		BottomLayer	*	66.42 x 34.003	mm				
	Tutor.TOP		GerberX	*	ТорАре		~			
	Source		Target		Size/Format					
	GerberX data		TopLayer	~	76.403 x 40.526	mm				
	Tutor.DRL		Excellon	~	Tutor.DRL		~			
	Source		Target		Size/Format					
	✓ Tutor.DRL		DrillPlated	×	76.543 x 42.57	mm				
2D	Apertures/Tools	Text	Tutor.DRL BoardOutline	Â						
51.			DrillPlated							
51.		•	TopLayer		•					
			TextTop SilkScreenTop							
			Sincocreantop	Carlos and a						
- 30.										
8.8		•			•					
mm				52.92						
< Materia	l l				Discard	Import				Layout 🔉

Fig. 143: User guidance step Import

- 11. Assign the holes to the layer DrillPlated (in the column Target).
- 12. Click on [Import].
- □ The user guidance step *Layout* is displayed:

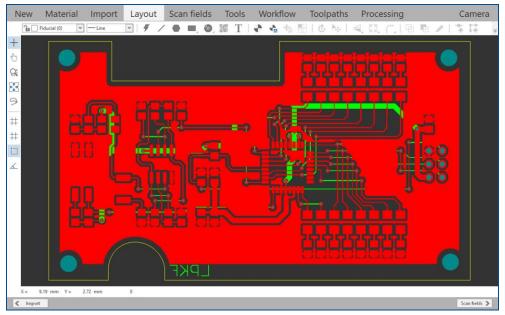


Fig. 144: User guidance step Layout

13. Click on Insert > Automatic fiducial creation or click on \P_{a} .

The function *Automatic fiducial creation* is only available in the license level Advanced of CircuitPro PL. If you have any questions contact the LPKF sales department.

- □ The dialog *Automatic fiducial creation* is displayed:
- 14. Adapt the settings for the fiducials.

In this example four fiducials with a distance of 4 mm from the *BoardOutline* are created.

- 15. Click on [OK].
- □ The fiducials are automatically added to the layout.
- For detailed information on creating fiducials refer to page 74.

The data have been prepared in CircuitPro PL.

Computing toolpaths and scan fields

For detailed information on computing toolpaths and scan fields refer to page 76.

- 1. Switch to the user guidance step *Scan fields*.
- 2. Click on [Compute scan fields] or on **I**.
- □ The dialog *Compute scan fields* is displayed.

i

CircuitPro PL computes different points at the scan field intersections for each task of the tool *Hatch* in order to prevent damage to the substrate at the scan field intersections. Thus, computing the scan fields should be done with the function *Overlap*.

The function *Offset* creates an offset between the scan fields of the layers *Top* and *Bottom*. This avoids the perforation of flexible substrates at the scan field intersections.

For detailed information on the overlap and on the offset between the scan fields, refer to the TechNote **ProtoLaser: Optimizing the processing quality**.

3. Deactivate the check box Without top to bottom scan field offset.

Com	put	te scan fi	eld	s	x
Scan fields set		< <all sca<="" td=""><td>n fie</td><td>ld sets>></td><td>~</td></all>	n fie	ld sets>>	~
Size	x	50 mm	у	50 mm	
Safe margin	x	5 mm	у	5 mm	
Without overlap					
Overlap	x	5 mm	у	5 mm	
Without top to bot	tom	scan field o	offse	t	
Top to bottom offset	x	2 mm	у	2 mm	
		Com			se

Fig. 145: Dialog Compute scan fields

4. Click on [Compute].

□ The scan fields are computed automatically. The *Overlap* is displayed:

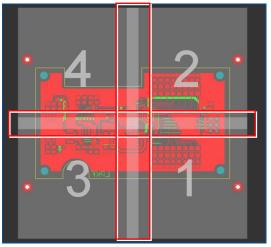


Fig. 146: Scan fields computed | Overlap displayed

5. Click on [Compute toolpaths] or on

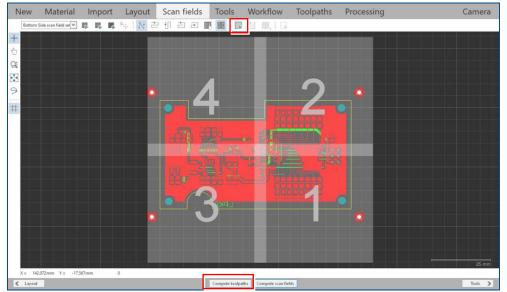


Fig. 147: Computing toolpaths

- □ The toolpaths are computed automatically with default settings. The message *Computation results* is displayed.
- 6. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 7. Click on [Close].
- 8. Save your project.
- The toolpaths and scan fields have been computed.



Flexible materials require a hatching processing method in most cases. You can adjust the parameters in the user guidance step *Workflow* by selecting the work package *Structure*.

The recommended *Pattern* is *xy parallel* with a *Hatching grid* that is approximately half the beam diameter of the tool *Hatch*. The parameter *Hatching contour overlap* determines how much overlap there is between the isolation toolpath and the hatching toolpaths. In most cases it should be set to 100 %. A lower value can result in incomplete copper removal on the scan field intersections.

Pattern	xy parallel	*
Hatching grid (default: ½ beam Ø)	8 µm	
Hatching contour overlap	100 %	

When processing sensitive materials, it is recommended to activate and adjust the *Skywriting* parameters of the tool *Hatch* in the user guidance step *Tools*.

Skywriting			
Start move length	300 µm	End move length	300 µm
	ore prin		ore prin

For detailed information on the hatching processing method and on the *Skywriting* parameters, refer to the TechNote **ProtoLaser: Optimizing the processing quality**.

Preparing the flexible material

The flexible material is very thin and delicate. Handle it with care!

- 1. Switch to the user guidance step *Processing*.
- 2. Click on > or click on [Start production].
- 3. When the message *Processing phase: Mount material* is displayed, place the flexible material onto the processing table and fasten it with adhesive tape.

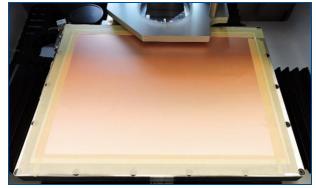


Fig. 148: Flexible material fastened to the processing table

- 4. Click on 🐫.
- □ The flexible material is fastened onto the processing table by vacuum.
- 5. Click on [OK].
- □ The following dialog is displayed:

		F	lacemen	t	-	
Material]	Process	ing data				
				ctive head to the a t and right rear co		
x 000,5 i	nm	x 305	,5 mm			
y 018,0 i	nm	y 247	,0 mm			
Width	305 mm	Length	229 mm	Set defau	t size	
Use a calip	er to meas	ure the mat	erial thickn	ess		
Material th	ickness	0.086 mm		Measure materi	al thickn	ess
				ОК		ancel

Fig. 149: Dialog Placement

- 6. Move the dialog *Placement* to get a better overview.
- 7. Determine the processing area. The base material position and base material size must match the processing area used by the CircuitPro PL software.



For detailed information on project placement by determining the processing area, refer to chapter 1.1.

- 8. Click on the tab *Processing data*.
- 9. Click on [Center on material].
- 10. Click on [OK].

□ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

i

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the flexible material is being prepared (fiducials and plated through holes are being drilled).

- 11. When the message *Processing phase: Dismount material* is displayed, remove the flexible material from the system and click on [OK].
- □ The following message is displayed:

	Processing phase: Galvanic through-hole	-	×
0	Carry out the galvanic through-hole plating process.		
	OK Cancel		

- Fig. 150: Message Processing phase: Galvanic through-hole plating
- 12. Click on [Cancel].
- The flexible material has been prepared.

Galvanic through-hole plating the flexible PCB (with Contac S4)

- 1. Switch on the system.
- 2. Select a profile.

It is recommended that the **total copper thickness** after through-hole plating is as low as possible. Too much copper deposition can lead to poorer processing quality.

The required copper deposition depends on the actual application. In this example 5 μ m were added to the original copper cladding.

- 3. Start the process.
- 4. Insert the flexible PCB into the PCB holder for flexible PCBs.



Fig. 151: PCB holder for flexible PCBs



The flexible PCB is fastened on three sides in the PCB holder. Thus, it remains properly secured and flat during plating. This way a good plating quality is achieved.

- 5. Tighten the screws on the sides until the flexible PCB is safely fastened.
- 6. Rinse the flexible PCB for at least 30 seconds with tap water.
- 7. Clean the flexible PCB.
- 8. Condition the flexible PCB.
- 9. Activate the flexible PCB.

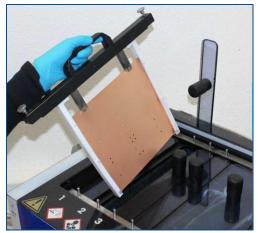


Fig. 152: Inserting the flexible PCB into tank 3

- 10. Clean the holes with LPKF ViaCleaner.
- 11. Copper-plate the flexible PCB.
- 12. Switch off the system.
- The flexible PCB has been galvanically through-hole plated.



For detailed information on galvanic through-hole plating, refer to the user manual of **Contac S4**.

Structuring the flexible PCB

- 1. Open the *.cp2d file you previously saved in CircuitPro PL.
- □ The opened project is displayed in the user guidance step *Processing*.
- 2. In the pane Workflow setup expand the group Processing.

Workflow setup	д
Material composition	
Layout	
Scan fields	
D Tools	
Workflow	
Toolpaths	
▲ Processing	
1. Galvanic THP	
2. Bottom Side	
3. Top Side	

Fig. 153: Pane Workflow setup

- 3. Right-click on the entry Bottom Side.
- □ The following context menu is displayed:

Workflow setup	д
Material com	position
Layout	
Scan fields	
D Tools	
Workflow	
Toolpaths	
Processing	
1. Galvanic THI	p
2. Bottom Side	
3. Top Side	Check tool settings
	<process selected=""></process>
	Process from here

Fig. 154: Context menu Process from here

- 4. Select Process from here.
- 5. When the message *Processing phase: Mount material bottom* is displayed, place the flexible PCB with the Bottom side (*BottomLayer*) facing upwards onto the processing table and fasten it with adhesive tape.
- 6. Click on 🐫.
- □ The flexible PCB is fastened onto the processing table by vacuum.
- 7. Click on [OK].

□ The following dialog is displayed:

Copper th	ickness	-	×
Enter the copper thickness:	23 µm		
	OK		Cancel

Fig. 155: Dialog Copper thickness

8. Enter the total copper thickness after galvanic through-hole plating into the input field (in this example 23 μ m).

You can determine the total copper thickness by calculating the thickness from the copper deposition rate in Contac S4 (approx. 0.15 μ m/min).

- 9. Click on [OK].
- □ The dialog *Placement* is displayed.
- 10. Move the dialog *Placement* to get a better overview.
- 11. Place the processing data.

The location of the layout **must match** the location of the flexible PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 12. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Bottom side (*BottomLayer*).
- For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the following dialog is displayed:

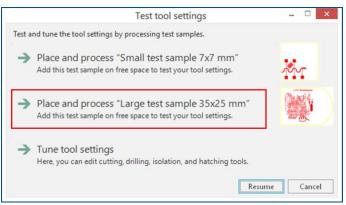


Fig. 156: Dialog Test tool settings



It is **recommended** that you process a **test sample** on your flexible PCB before you continue with production.

13. Select Place and process "Large test sample 35x25 mm".



For detailed information on testing and setting the tools, refer to chapter 1.5.

- 14. Check the processed test sample with a hand-held microscope.
- 15. Determine whether the processing quality is satisfactory.
- 16. If the processing quality is not satisfactory, tune the tool settings.
- For detailed information on tuning the tool settings, refer to chapter 1.5.
 - 17. If the processing quality is satisfactory, click on [Resume].
 - Structuring of the Bottom side (*BottomLayer*) is started.
 - 18. When the message *Processing phase: Flip material* is displayed, turn the flexible PCB over around the symmetry axis of the system and fasten it using adhesive tape.
 - 19. Click on [OK].
 - □ The dialog *Placement* is displayed.
 - 20. Place the processing data **matching** the location of the flexible PCB and fiducials on the processing table.
 - 21. When project placement is complete, click on [OK].
 - 22. The laser system reads the fiducials on the Top side (*TopLayer*).

After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed. When finished, the dialog *Test tool settings* is displayed.

- 23. Repeat the steps 13 to 17.
- 24. If the processing quality is satisfactory, click on [Resume].
- □ The laser system structures the Top side (*TopLayer*), drills the unplated through holes and cuts out the flexible PCB.
- 25. When the message *Board production finished* is displayed, remove the flexible PCB from the system.
- The flexible PCB has been structured.

Cleaning the flexible PCB

The flexible PCB is extremely thin and needs to be handled with care!

- 1. Spray the flexible PCB with LPKF Cleaner and use a brush to clean it.
- 2. Rinse the flexible PCB with tap water and dry it.
- The flexible PCB has been cleaned.

The PCB production is finished.



Residual copper or damaged substrate

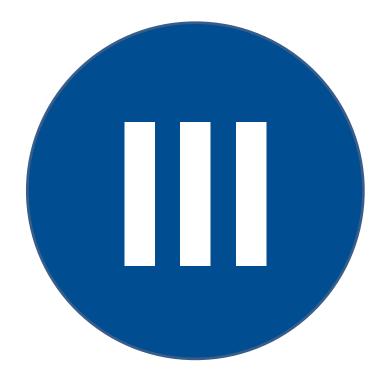
If there is still copper on the PCB or if the substrate is damaged, check the material and the tools with the laser parameters specific to the material.

Adjust the parameters of the tools in such a way that no copper remains on the PCB after processing. For detailed information on the tool settings, refer to chapter 1.5.

In some cases, it is not possible to remove all copper by structuring without damaging the delicate substrate. Use the LPKF ViaCleaner to remove the remaining copper.

For detailed information on cleaning the PCB with LPKF ViaCleaner, refer to the TechNote **ProtoLaser: Optimizing the processing quality**.

Multi-layer PCBs



i

3 **Producing multi-layer PCBs**

The production of multi-layer PCBs is only available in the license level Advanced of CircuitPro PL. If you have any questions contact the LPKF sales department.

This chapter describes the production of four different types of multi-layer PCBs:

- 1. Producing a multi-layer PCB with galvanic through-hole plating by using a laser system only.
- 2. Producing a multi-layer PCB with galvanic through-hole plating.
- 3. Producing a multi-layer PCB with non-galvanic through-hole plating.
- 4. Producing a multi-layer PCB with blind vias and buried vias.

The following LPKF systems are required for the procedures:

Procedure	LPKF system
1	ProtoLaser U4/S4/R4, MultiPress S, Contac S4
2	ProtoLaser U4/S4/R4, ProtoMat S or E, MultiPress S, Contac S4
3	ProtoLaser U4/S4/R4, ProtoMat S or E, MultiPress S, ProConduct
4	ProtoLaser U4/S4/R4, ProtoMat S or E, MultiPress S, Contac S4

Table 5: Required LPKF systems

3.1 How to produce a multi-layer PCB with galvanic through-hole plating by using a laser system only

Auxiliaries

This chapter describes how to create a 4-layer PCB using a laser system, a multi-layer press and a galvanic through-hole plating system.

Convection oven

LPKF Cleaner

Brush

• Tap water

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

(order code: 115877)

(order code: 115891)

Oil-free compressed air

Consumables

- 1 × Base material FR4 229 mm × 305 mm × 1 mm, 18/18 µm copper-clad (order code: 119574)
- 2 × Thin Laminate 104 ML 0/5 µm, 229 mm × 305 mm × 0.2 mm with protection foil (order code: 119571)
- 4 × Prepreg Type 2125, 200 mm × 275 mm × 0.1 mm (order code: 119572)

System

- ProtoLaser U4/S4/R4
- MultiPress S
- Contac S4

Ĭ

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PL
- Preparing the laminate materials (with ProtoLaser)
- Structuring the core material (with ProtoLaser)
- Assembling and pressing the multi-layer stack (with MultiPress S)
- Drilling plated through holes into the multi-layer PCB (with ProtoLaser)
- Galvanic through-hole plating the multi-layer PCB (with Contac S4)
- Structuring the outer layers of the multi-layer PCB (with ProtoLaser)

Preparing the data in CircuitPro PL

For detailed information on preparing the data in CircuitPro PL refer to chapter 2.1.

- 1. In the user guidance step New select 4-layer-Multilayer from the drop-down list.
- 2. Select your laser system from the drop-down list (in this example PL U4).
- A list of templates for 4-layer multi-layer materials is displayed:

Open project		PCB with four predefined layers, prepared for ProtoLaser U4 systems with galvanic through-hole plating and MultiPress S	
Open template			
	< >		

Fig. 157: List of multi-layer templates

- 3. Select the template *4 Layers, ProtoLaser U4, galvanic through-hole plating, MultiPress S.*
- 4. Click on [Load template] or double-click on the template.

- □ The user guidance step *Material* is displayed.
- 5. Select the material LPKF 4-Layer_Galvanic.

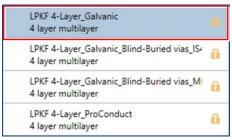


Fig. 158: Material selection

- 6. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 7. Click on 📑
- □ The dialog *Open* is displayed.
- 8. Navigate to the folder that contains the data you want to import. For the example data that are used for this tutorial, refer to the folder: C:\Program Files\LPKF Laser & Electronics AG\ LPKF CircuitPro PL\Example Data\UseCase Multilayer.
- 9. Select the files you want to import. In this example:
 - Tutor_MultiLayer_4_Lagen.BOA, Tutor_MultiLayer_4_Lagen.BOT, Tutor_MultiLayer_4_Lagen.DRL, Tutor_MultiLayer_4_Lagen.LY2, Tutor_MultiLayer_4_Lagen.LY3 and Tutor_MultiLayer_4_Lagen.TOP.
- 10. Click on [Open].
- □ The data are automatically assigned to the correct layers and the user guidance step *Import* is displayed:

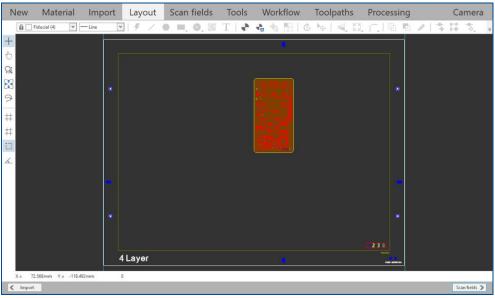
New	Material	Import	Layout Scan fie	lds	Tools Workflow	Т	00	lpaths	Processing	Camera
6	Source		larget		Size/Format		^	File Fi	ormat	
4	✓ BoardOutline		BoardOutline	~	40.205 x 75.13 mm					
Ū	Tutor_MultiLayer_4_La	gen.BOT	GerberX	×	BotApe	~	ł	Unit Values	Millimeters V Absolute V	
ξ	Source		Target		Size/Format			values	ADJOIDTE	
2	BottomLayer		BottomLayer	~	36.643 x 67.958 mm			Decimal	Omit leading zeros	
	Tutor_MultiLayer_4_La	gen.LY2	GerberX	~	Tutor_MultiLayer_4_Lagen.LY2	~	l	Digits m.n	2 4	
	Source		Target		Size/Format					
	✓ Layer2		Layer2	~	38.43 x 73.381 mm					
	Tutor_MultiLayer_4_La	gen.LY3	GerberX	~	Tutor_MultiLayer_4_Lagen.LY3	~	l			
	Source		Target		Size/Format					
	✓ Layer3		Layer3	~	38.43 x 73.381 mm					
	Tutor_MultiLayer_4_La	gen.TOP	GerberX	~	ТорАре	~	l			
	Source		Target		Size/Format					
	✓ TopLayer		TopLayer 🗸		36.467 x 70.166 mm					
	Tutor_MultiLayer_4_La	gen.DRL	Excellon	~	Tutor_MultiLayer_4_Lagen.DRL	~				
	Source		Target		Size/Format					
	Tutor_MultiLaye	er_4_Lagen.DRL	Tutor_MultiLayer_4_Lagen.DRL	~	31.775 x 65.608 mm		~			
2D			Core	^						
20	Apertures/Tools	Text	DrillPlated			_				
			Uniforpiated	_	<u>,</u>					
69			TopLayer							
			Layer2 Layer3							
47			BottomLayer	~						
-0										
mm			-0.10 20.00 40.11							
≪ Materi	al				Discard Import					Layout 🔰

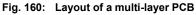
Fig. 159: User guidance step Import

11. Assign the holes to the layer DrillPlated (in the column Target).

i

- 12. Click on [Import].
- □ The layout is displayed in the user guidance step Layout.





Multi-layer templates already include among others fiducials, register holes, prepreg area, base material area etc. that are required for precise layer assembly in the press mold.

13. If desired, multiply the layout (in this example the layout is not multiplied).

The layout has to be within the prepreg material frame. It is preferable to move it **to the center**.

- 14. Select the entire layout.
- □ The layout is highlighted and changes its color.
- 15. Right-click on the highlighted layout.
- □ The following context menu is displayed:

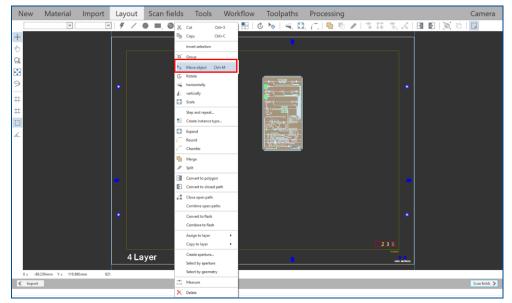
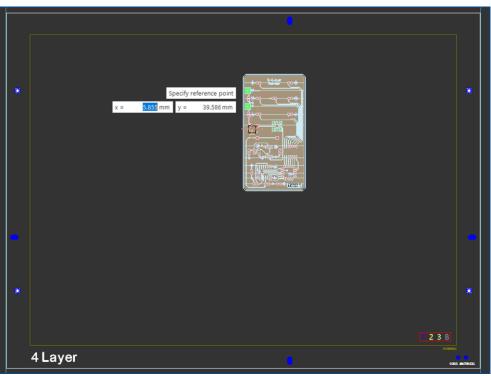


Fig. 161: Context menu Move object

16. Select *Move object* or click on \mathbb{A}_{+} .



□ The input fields for *Specify reference point* are displayed:

Fig. 162: Specify reference point

- 17. Click on a point in the **original layout** to specify the **reference point**.
- 18. A copy of the layout (in orange) and the input fields for *Specify distance* are displayed:

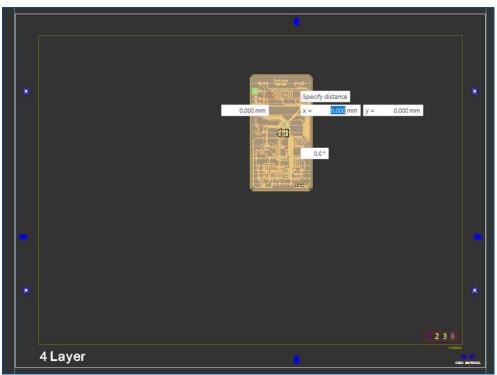
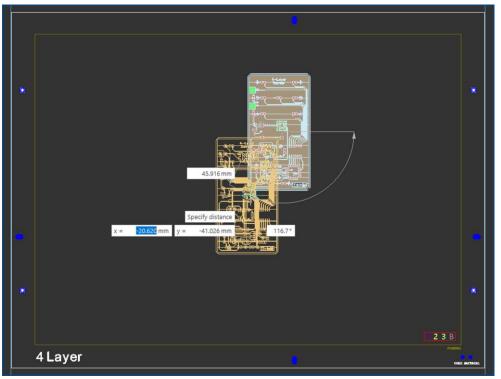


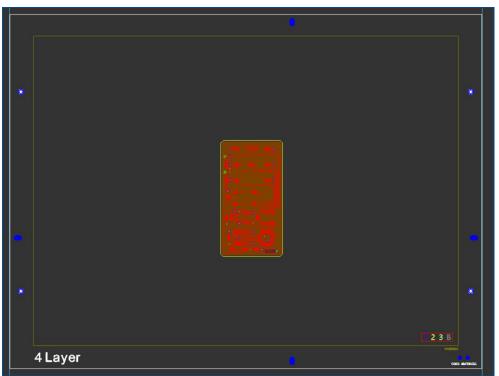
Fig. 163: Specify distance



19. Move the copied layout with the mouse to the desired position:

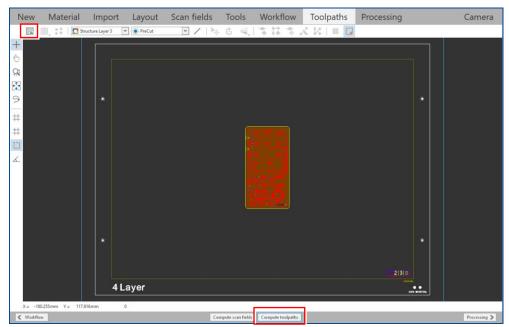


- 20. When you reach the desired position, left-click to specify the target point.
- 21. In the user guidance step *Layout* click anywhere on the black background or press [Esc] to disable the *Move object* function.



□ The layout has been moved to the desired position:

Fig. 165: Layout moved to the desired position



22. Switch to the user guidance step Toolpaths.

Fig. 166: Opened project

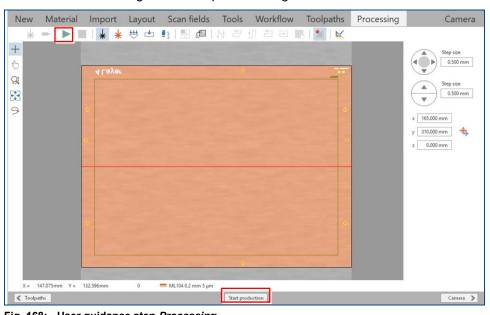
23. Click on [Compute toolpaths] or on **[**].

□ The following dialog is displayed:

	Computing toolpaths ×							
A	Scan fields sets "Prepare Laminate Bottom scan field set", "Prepare Laminate Top scan field set", "[Core] Structure Layer 3 scan field set", "[Core] Structure Layer 2 scan field set", "[Top-Bot] Galvanic THP scan field set", "[Top-Bot] Bottom Side scan field set", "[Top-Bot] Top Side scan field set" have no scan fields. They will be computed automatically.							
	Continue							

Fig. 167: Dialog Computing toolpaths

- 24. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 25. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 26. Click on [Close].
- 27. Save your project.
- The data have been prepared in CircuitPro PL.



Preparing the laminate materials (with ProtoLaser)

1. Switch to the user guidance step *Processing*.

Fig. 168: User guidance step Processing

2. Click on [Start production] or on .

The following message is displayed:

	Processing phase: Mount laminate 🛛 – 🗖 🗙
0	Mount the laminate onto the machine's processing area.
	OK Cancel

Fig. 169: Message Processing phase: Mount laminate

3. Place the first laminate with the **copper side facing upwards** in the center of the processing table.



The laminate material is very thin and therefore usually bent. Click on $\stackrel{\text{\tiny HH}}{=}$ and fasten the laminate material along its edges with adhesive tape.

- 4. Click on [OK].
- □ The following dialog is displayed.

	Pla	cement		-		×
Material	Processing data					
	ne processing area to mo se the buttons to set the					
x 000.5 m	nm x 305.5 r	nm				
y 018,0 m	nm y 247.0 r	nm				
Width	05 mm Length 22	9 mm	Set default s	ize		
Use a calip	r to measure the materia	l thickness				
Material th	ckness 0.205 mm	Me	asure material th	ickne	ss	
			OK		Canc	el



5. Move the dialog *Placement* to get a better overview.

6. Determine the processing area. The base material position and base material size **must match** the processing area used by the CircuitPro PL software.

For detailed information on project placement by determining the processing area, refer to chapter 1.1.

- 7. Click on the tab *Processing data*.
- 8. Click on [Center on material].
- 9. Click on [OK].
- □ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

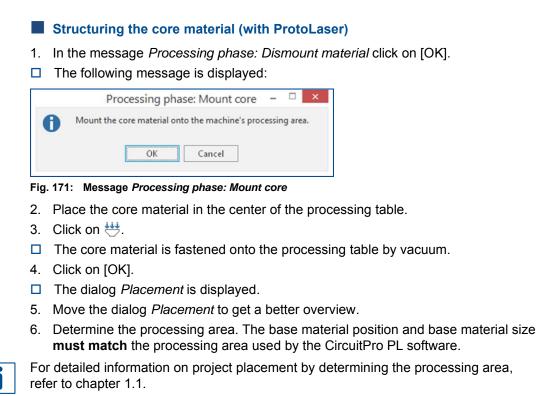
i

When finished, the first laminate (*BottomLayer*) is being prepared (positioning holes and register holes are being drilled).

- 10. When the message *Processing phase: Dismount material* is displayed, remove the first laminate from the system and click on [OK].
- 11. When the message *Processing phase: Mount laminate* is displayed, place the second laminate with the **copper side facing upwards** in the center of the processing table.
- 12. Click on [OK].
- □ The dialog *Placement* is displayed.
- 13. Move the dialog *Placement* to get a better overview.
- 14. Determine the processing area. The base material position and base material size must match the processing area used by the CircuitPro PL software.
- 15. Click on the tab *Processing data*.
- 16. Click on [Center on material].
- 17. Click on [OK].
- □ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

When finished, the second laminate (*TopLayer*) is being prepared (positioning holes and register holes are being drilled).

- 18. When the message *Processing phase: Dismount material* is displayed, remove the second laminate from the system.
- The laminate materials have been prepared.



- 7. Click on the tab Processing data.
- 8. Click on [Center on material].
- 9. Click on [OK].
- □ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

i

For detailed information on material thickness measurement, refer to chapter 1.3.

After material thickness measurement, the core material is being prepared (fiducials and register holes are being drilled).

When finished, the dialog *Test tool settings* is displayed.

In this example, testing and setting the tools is to be skipped.



For detailed information on testing and setting the tools, refer to chapter 1.5.

- 10. Click on [Resume].
- Structuring the Bottom side of the core material (*Layer3*) is started.
- 11. When the message *Processing phase: Flip material* is displayed, turn the core material over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.

i

Two positioning holes (3 mm) help you with the correct orientation of the core material. When the **Top side** of the core is facing **upwards**, the positioning holes should be located in the **right front corner** of the core material:



- □ The dialog *Placement* is displayed.
- 12. Place the processing data.

The location of the layout **must match** the location of the core material and fiducials on the processing table.

i

For detailed information on project placement according to fiducial positions refer to chapter 1.1.

- 13. When project placement is complete, click on [OK].
- □ The laser system reads the fiducials on the Top side of the core material (*Layer2*).



For detailed information on fiducial recognition refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed. When finished, the dialog *Test tool settings* is displayed.

- 14. Click on [Resume].
- Structuring the Top side of the core material (*Layer2*) is started. The following message is displayed:

	Processing phase: Press all layers	-	×
0	Press all layers		
	OK Cancel		

Fig. 172: Message Processing phase: Press all layers

- 15. Click on [Cancel] to temporarily stop the process.
- 16. Remove the core material from the system.
- 17. Spray the core material with LPKF Cleaner and use a brush to clean it.
- 18. Rinse the core material with tap water and dry it with compressed air.
- The core material has been structured.

Assembling and pressing the multi-layer stack (with MultiPress S)



For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.6.

- 1. Start the pre-heating process of the MultiPress S.
- 2. Prepare the materials.



Dry the materials in a convection oven at 100 $^{\circ}$ C (212 $^{\circ}$ F) for 30 minutes before assembling them in the press mold. Do not dry the prepreg material.

- 3. Assemble the multi-layer stack in the press mold.
- 4. Press the multi-layer stack.
- The multi-layer stack has been assembled and pressed.

Drilling plated through holes into the multi-layer PCB (with ProtoLaser)

1. Click on *File* > *Open* or =.



You can open the file also by clicking in the user guidance step *New* on [Open project...].

- □ The dialog *Open* is displayed.
- 2. Select the file you previously saved.
- 3. Click on [Open].
- The opened project is displayed in the user guidance step Processing.
- 4. In the pane Workflow setup expand the entry [Top-Bot] Galvanic THP.

Workflow setup 7
Material composition
Layout
Scan fields
D Tools
Workflow
Toolpaths
Processing
1. Prepare Laminate Bottom
2. Prepare Laminate Top
3. [Core] Structure Layer 3
4. [Core] Structure Layer 2
▲ 5. [Top-Bot] Galvanic THP
1. Press All Layers
2. Mount Material Top
3. Material Thickness
4. Material Placement
5. Read Fiducials Top
6. Drill Plated Through-Holes
7. Process Galvanic THP
6. [Top-Bot] Bottom Side
7. [Top-Bot] Top Side

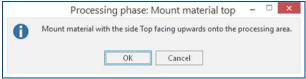
Fig. 173: Pane Workflow setup

- 5. Right-click on Mount Material Top.
- □ The following context menu is displayed:

Workflow setup	ф
Material composition	
Layout	
Scan fields	
Tools	
Workflow	
Toolpaths	
Processing	
1. Prepare Laminate Bottom	
2. Prepare Laminate Top	
3. [Core] Structure Layer 3	
4. [Core] Structure Layer 2	
 5. [Top-Bot] Galvanic THP 	
1. Press All Layers	_
2. Mount Material Top	
<process selected=""></process>	
Process from here	
6. Drill Plated Through-Holes	
7. Process Galvanic THP	
6. [Top-Bot] Bottom Side	
7. [Top-Bot] Top Side	
Fig. 174: Context menu Process from here	

- 6. Select Process from here.





- Fig. 175: Message Processing phase: Mount material top
- 7. Place the multi-layer PCB with the Top side (TopLayer) facing upwards onto the processing table.

Ensure that the positioning holes are located in the right front corner.

- 8. Click on 🐫.
- □ The multi-layer PCB is fastened to the processing table by vacuum.
- 9. Click on [OK].

□ The dialog *Placement* is displayed:

			Placeme	ent	-		×
Material	Processi	ng data					
				active head to the ont and right rear co			
		_ *					
x 000.5 r	nm	x 30	05,5 mm				
y 018.0 r	mm	y 24	47.0 mm				
Width	305 mm	Length	229 mm	Set defau	lt size		
Use a calip	er to measi	ure the m	aterial thick	iness			
Material th	ickness	1.806 m	ım	Measure material	thickne	255	
				oly OK		Cano	el

Fig. 176: Dialog Placement

- 10. Move the dialog *Placement* to get a better overview.
- 11. Click on the tab *Processing data*.
- 12. Place the processing data. The location of the layout **must match** the location of the multi-layer PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 13. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Top side (*TopLayer*).
- For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.



For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the laser system drills the plated through holes. The following message is displayed:

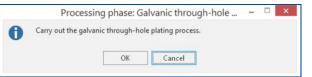


Fig. 177: Message Processing phase: Galvanic through-hole plating

- 14. Click on [Cancel].
- 15. Remove the multi-layer PCB from the system.
- The plated through holes have been drilled into the multi-layer PCB.

Galvanic through-hole plating the multi-layer PCB (with Contac S4)

- 1. Switch on the system.
- 2. Select a profile.



The recommended total copper thickness after through-hole plating is 30 μm to 35 $\mu m.$

- 3. Start the process.
- 4. Prepare the multi-layer PCB for through-hole plating.
- 5. Clean the multi-layer PCB.
- 6. Condition the multi-layer PCB.
- 7. Activate the multi-layer PCB.
- 8. Clean the holes with LPKF ViaCleaner.
- 9. Copper-plate the multi-layer PCB.
- 10. Switch off the system.
- The multi-layer PCB has been galvanically through-hole plated.



For detailed information on galvanic through-hole plating, refer to the user manual of **Contac S4**.

Structuring the outer layers of the multi-layer PCB (with ProtoLaser)

- 1. Open the *cp2d file you previously saved in CircuitPro PL.
- □ The opened project is displayed in the user guidance step *Processing*.
- 2. In the pane Workflow setup expand the group Processing.
- 3. Right-click on the entry [Top-Bot] Bottom Side.
- 4. Select Process from here in the context menu.
- 5. When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.

Ensure that the positioning holes are located in the right rear corner.

- 6. Click on 👑.
 - 7. The multi-layer PCB is fastened to the processing table by vacuum.
- 8. Click on [OK].
- 9. The following dialog is displayed:

Copper t	hickness	-	□ ×
Enter the copper thickness:	5 µm]	
1	OK		Cancel

Fig. 178: Dialog Copper thickness

10. Enter the total copper thickness after galvanic through-hole plating into the input field.



You can determine the total copper thickness by calculating the thickness from the copper deposition rate in Contac S4 (approx. $0.15 \mu m/min$).

11. Click on [OK].

- □ The dialog *Placement* is displayed.
- 12. Move the dialog *Placement* to get a better overview.
- 13. Click on the tab Processing data.
- 14. Place the processing data. The location of the layout **must match** the location of the multi-layer PCB and fiducials on the processing table.

i

For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 15. After project placement is complete click on [OK].
- The laser system reads the fiducials on the Bottom side (BottomLayer).



For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the dialog *Test tool settings* is displayed.

In this example, testing and setting the tools is to be skipped.

i

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 16. Click on [Resume].
- □ Structuring the Bottom side (*BottomLayer*) is started.
- 17. When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 18. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
- 19. When project placement is complete, click on [OK].
- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 20. Click on [Resume].
- Structuring the Top side (*TopLayer*) is started.
- 21. When the message *Board production finished* is displayed, remove the multi-layer PCB from the system.
- 22. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean it.
- 23. Rinse the multi-layer PCB with tap water and dry it with compressed air.
- The outer layers of the multi-layer PCB have been structured.

The multi-layer PCB production is finished.

3.2 How to produce a multi-layer PCB with galvanic through-hole plating

This chapter describes how to create a **4-layer** PCB using a laser system, a circuit board plotter, a multi-layer press, and a galvanic through-hole plating system.

Convection oven

(order code: 115877)

(order code: 115891)

Oil-free compressed air

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

1 × Base material FR4 229 mm × 305 mm × 1 mm, 18/18 µm copper-clad (order code: 119574)

- 2 × Thin Laminate 104 ML 0/5 µm, 229 mm × 305 mm × 0.2 mm with protection foil (order code: 119571)
- 200 mm × 275 mm × 0.1 mm (order code: 119572)

Auxiliaries

LPKF Cleaner

Tap water

Brush

•

System

- 1 set of tools for ProtoMat ProtoLaser U4/S4/R4
 - MultiPress S
 - ProtoMat S or E •
 - Contac S4

- •
- 4 × Prepreg Type 2125,

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PM •
- Preparing the core material and laminate materials (with ProtoMat)
- Preparing the data in CircuitPro PL •
- Structuring the core material (with ProtoLaser)
- Assembling and pressing the multi-layer stack (with MultiPress S)
- Drilling plated through holes into the multi-layer PCB (with ProtoMat) •
- Galvanic through-hole plating the multi-layer PCB (with Contac S4)
- Structuring the outer layers of the multi-layer PCB (with ProtoLaser)
- Drilling unplated through holes and cutting out the multi-layer PCB (with ProtoMat)

Preparing the data in CircuitPro PM



For detailed information on preparing the data in CircuitPro PM refer to chapter 2.2.

- 1. In the tab *Templates* of the dialog *New document*, select the template: PM_PL_4Layer_GalvanicTHP_MultiPressS.cbf.
- 2. Click on File > Import or on $\stackrel{\bullet}{\ge}$.
- 3. Navigate to the folder that contains the data you wish to import. The example data used for this tutorial are located in the folder: C:\Program Files (x86) \LPKF Laser & Electronics \ LPKF CircuitPro PM\Example Data\UseCase Multilayer.
- 4. Select the files you wish to import. In this example: Tutor_MultiLayer_4_Lagen.BOA, Tutor_MultiLayer_4_Lagen.BOT, Tutor_MultiLayer_4_Lagen.DRL, Tutor_MultiLayer_4_Lagen.LY2, Tutor_MultiLayer_4_Lagen.LY3 and Tutor_MultiLayer_4_Lagen.TOP.
- 5. Click on [Open].

□ The data are automatically assigned to the correct layers and the following dialog is displayed:

	File Name	Format		Aperture/Tool List		Layer/Template		Size	/Format			
	Tutor_MultiLayer_4_Lagen.BOA	GerberX 🗸 BoardApe 🛛 BoardOutline		BoardOutline	~	40.205 x 75.13 mm						
	Tutor_MultiLayer_4_Lagen.BOT	GerberX	×	BotApe	~	BottomLayer	~	37.6	37.643 x 67.958 mm			
	Tutor_MultiLayer_4_Lagen.LY2	GerberX	~	Tutor_MultiLayer_4_Lagen.LY	Ŷ	Layer2	~	38.4	3 x 73.3	81 m	m	
	Tutor_MultiLayer_4_Lagen.LY3	GerberX	~	Tutor_MultiLayer_4_Lagen.LY	~	Layer3	~	38.4	3 x 73.3	81 m	m	
	Tutor_MultiLayer_4_Lagen.TOP	GerberX	Y	TopApe	~	TopLayer	~	-	67 x 70			
	Tutor_MultiLayer_4_Lagen.DRL	Excellon	Y	Tutor_MultiLayer_4_Lagen.DR	Y	DrilPlated	~	31.7	75 x 65	.608 r	nm	
			-									
D View	Apertures/Tools Text View	Messag	e V	iew -			Gene	ral				
							Size		31.775	x 65.60	8 mm	
							Unit		Millim	eters		
							Value		Absolu			-
							Decin	nal	Omitle	eading	zeros	_
							Digits	m.n	2	-	4	
26.06												
20.00												
							Ad	just d	rill files			
		2.46		34.24								
Perdi								-				
Ready												

Fig. 179: Dialog Import | Assigned layers

- 6. Assign the holes to the layer DrillPlated (in the Layer/Template column).
- 7. Click on [OK].
- □ The imported data is displayed in the *CAM view*.

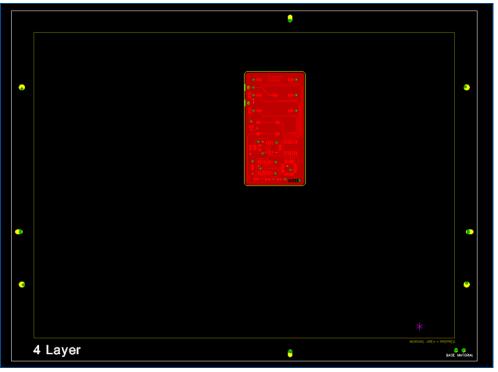


Fig. 180: CAM view multi-layer PCB

Multi-layer templates already include fiducials, register holes, working area frames etc., which are needed for precise layer assembly in the press mold.

- 8. Press and hold the <u>Ctrl</u> key and click on all holes that are not to be plated.
- □ The selected holes are highlighted.
- 9. Right-click on the layout and under Assign objects to layer select DrillUnplated.
- □ An additional layer is created automatically. Holes that are not to be plated are assigned to the layer *DrillUnplated*.
- 10. If desired, multiply the layout (in this example the layout is not multiplied).



The layout has to be within the working area and prepreg material frame. It is preferable to move it **to the center**.

- 11. Select the layout.
- □ The layout is highlighted in gray.
- 12. Click on 🕂.
- 13. Using the left mouse button, drag and drop the layout to the desired location.
- □ The layout has been moved.
- 14. Click on *Toolpath* > *Technology Dialog*... or on \mathbb{N} .
- □ The dialog *Technology Dialog* is displayed.
- 15. Click on the tab *Contour routing* and select *Edge gaps* using the arrow buttons. This is usually the preferred method of contour routing.
- 16. Click on the tab *Pockets* and deactivate the check box *Process*, since there are no pockets in this project.
- 17. Click on [Start].
- □ The Computation results are displayed in a dialog.
- 18. Click on [Close].
- □ The toolpaths have been calculated.
- 19. Save the file.
- The data have been prepared in CircuitPro PM.

Preparing the core material and laminate materials (with ProtoMat)

- 1. Load the tool magazine and assign the tools to positions.
- 2. Click on Machining > Process all or on >.
- 3. Perform the following phases with the core material:
 - MountMaterial
 - MaterialSettings
 - Placement
- □ The phases *DrillFiducial* and *PrepareCore* are performed.
- 4. When the message *Processing Phase: DismountMaterial* is displayed, remove the core material from the system and click on [OK].
- 5. Rinse the core material with tap water and dry it with compressed air.
- 6. Perform the following phases with the first laminate:
 - MountLaminate
 - MaterialSettings_1
 - Placement_1
- □ The *PrepareLaminateBottom* phase is performed.
- 7. When the message *Processing Phase: DismountMaterial_1* is displayed, remove the first laminate from the system and click on [OK].
- 8. Perform the following phases with the second laminate:
 - MountLaminate_1
 - MaterialSettings_2
 - Placement_2
- □ The *PrepareLaminateTop* phase is performed.
- 9. When the message *Processing Phase: ProtoLaserStructuring* is displayed, remove the second laminate from the system and click on [Cancel] to temporarily stop the process.
- The core material and laminate materials have been prepared.



For detailed information on ProtoMat phases refer to the how-to guides of ProtoMat.

Preparing the data in CircuitPro PL

i

For detailed information on preparing the data in CircuitPro PL refer to chapter 2.2.

- 1. Switch to the user guidance step New.
- 2. Click on [Open project...].
- □ The dialog *Open* is displayed.
- 3. Navigate to the folder that contains the file you previously saved in CircuitPro PM.
- 4. Select CBF document (*.cbf) from the drop-down list.
- □ Only .cbf files are displayed.
- 5. Select the appropriate file and click on [Open].
- □ The dialog *Upgrade project* is displayed.
- 6. Perform the following steps:
 - Select 4-layer-Multilayer from the drop-down list.
 - Select your laser system from the drop-down list (in this example *PL U4*).
 - Activate the check box Supported by ProtoMat.
 - Select the template 4 Layers, ProtoLaser U4, ProtoMat, galvanic through-hole plating, MultiPress S.

□ The dialog *Upgrade project* changes as follows:

Upgrade project	×
The file C:\Users\User name\Desktop\Double-sided PCB.cbf contains old format data and requires to be upgraded. Choose a template for the upgrade.	
4-layer-Multilayer V PL U4 V Supported by ProtoMa	t
LPKF templates 4 Lavers, ProtoLaser U4, ProtoMat, galvanic through-hole plating, MultiPress S 4 Layers, ProtoLaser U4, ProtoMat, ProConduct, MultiPress S 4 Layers, ProtoLaser U4, ProtoMat, ProConduct, MultiPress S 4 Layers, ProtoLaser U4, ProtoMat, galvanic through-hole plating, MultiPress S, blind-b 4 Layers, ProtoLaser U4, ProtoMat E44, galvanic through-hole plating, MultiPress S 4 Layers, ProtoLaser U4, ProtoMat E44, ProConduct, MultiPress S 4 Layers, ProtoLaser U4, ProtoMat E44, ProConduct, MultiPress S	
Ignore fiducial and information layers Upgrade Close	

Fig. 181: Upgrade selected

- 7. Click on [Upgrade] or double-click on the template.
- □ The user guidance step *Material* is displayed.

8. Select the material *LPKF 4-Layer_Galvanic*.

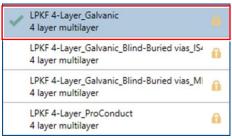


Fig. 182: Material selection

- 9. Click on [Select material] or double-click on the material.
- □ The layout of the opened project is displayed in the user guidance step *Layout*.

10. Switch to the user guidance step Toolpaths.

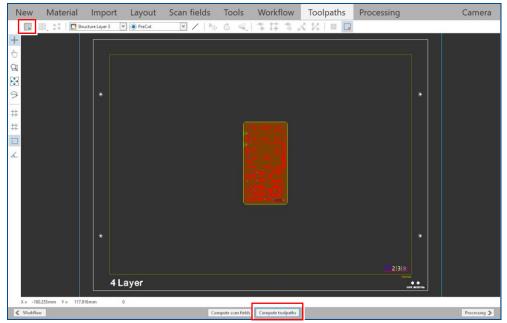


Fig. 183: Opened project

- 11. Click on [Compute toolpaths] or on
- □ The following dialog is displayed:

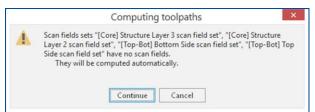
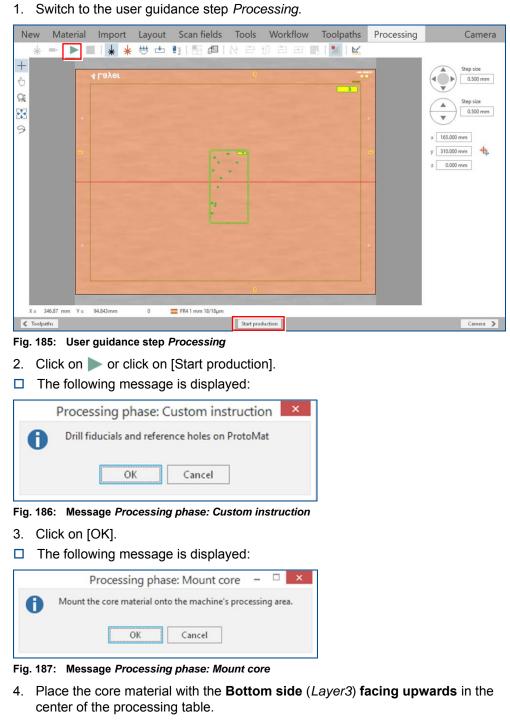


Fig. 184: Dialog Computing toolpaths

- 12. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 13. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 14. Click on [Close].
- 15. Save your project.
- The data have been prepared in CircuitPro PL.



Structuring the core material (with ProtoLaser)

Ensure that the positioning holes are located in the right rear corner.

- 5. Click on 🐫.
- □ The core material is fastened onto the processing table by vacuum.
- 6. Click on [OK].

□ The dialog *Placement* is displayed:

			Pl	acem	ent				-		×
Material	Processi	ng data									
	the processi lse the butto										
			i								
× 000,5 r	mm	x	305,5	i mm							
y 018.0 r	mm	У	247,0) mm							
Width	305 mm	Length		229 mr	n		Set de	fault	size		
Use a calip	er to measu	ire the r	mate	rial thio	kness:						
Material th	ickness	1.036	mm		Me	asure	e mate	erial tl	hickne	55	
							0	K		Cano	el

Fig. 188: Dialog Placement

- 7. Move the dialog *Placement* to get a better overview.
- 8. Click on the tab Processing data.
- 9. Place the processing data.

The location of the layout **must match** the location of the core material and fiducials on the processing table.

For detailed information on project placement according to fiducial positions refer to chapter 1.1.

- 10. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Bottom side of the core material (*Layer3*).



After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.



For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the dialog *Test tool settings* is displayed.



In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 11. Click on [Resume].
- Structuring the Bottom side of the core material (*Layer3*) is started.
- 12. When the message *Processing phase: Flip material* is displayed, turn the core material over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 13. Place the processing data **matching** the location of the core material and fiducials on the processing table.
- 14. When project placement is complete, click on [OK].

- The laser system reads the fiducials on the Top side of the core material (*Layer2*). After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed. When finished, the dialog *Test tool settings* is displayed.
- 15. Click on [Resume].
- Structuring the Top side of the core material (*Layer2*) is started. The following message is displayed:

	Processing phase: Press all layers	-	×
0	Press all layers		
	OK Cancel		

Fig. 189: Message Processing phase: Press all layers

- 16. Click on [Cancel] to temporarily stop the process.
- 17. Remove the core material from the system.
- 18. Spray the core material with LPKF Cleaner and use a brush to clean it.
- 19. Rinse the core material with tap water and dry it with compressed air.
- The core material has been structured.

Assembling and pressing the multi-layer stack (with MultiPress S)

For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.6.

- 1. Start the pre-heating process of the MultiPress S.
- 2. Prepare the materials.



Dry the materials in a convection oven at 100 $^{\circ}$ C (212 $^{\circ}$ F) for 30 minutes before assembling them in the press mold. Do not dry the prepreg material.

- 3. Assemble the multi-layer stack in the press mold.
- 4. Press the multi-layer stack.
- The multi-layer stack has been assembled and pressed.

Drilling plated through holes into the multi-layer PCB (with ProtoMat)

- 1. Open the *.cbf file you previously saved in CircuitPro PM.
- 2. Switch to the pane Processing.
- 3. Select *MountMaterialTop* from the drop-down list and click on .
- □ The message Processing Phase: MountMaterialTop is displayed.
- Place the multi-layer PCB onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** and fasten it using adhesive tape.
 Ensure that the positioning holes are located in the right front corner.
 - Ensure that the positioning noies are located in the right from
- 5. Perform the following phases:
 - MaterialSettings_3
 - Placement_3
- □ The phases ReadFiducialsTop, MarkingDrills and DrillingPlated are performed.
- 6. When the message *Processing Phase: ThroughHolePlating* is displayed, remove the multi-layer PCB from the system and click on [Cancel] to temporarily stop the process.
- ✓ The plated through holes have been drilled into the multi-layer PCB.



For detailed information on ProtoMat phases refer to the how-to guides of ProtoMat.

Galvanic through-hole plating the multi-layer PCB (with Contac S4)

- 1. Switch on the system.
- 2. Select a profile.

The recommended total copper thickness after through-hole plating is $30 \ \mu m$ to $35 \ \mu m$.

- 3. Start the process.
- 4. Prepare the multi-layer PCB for through-hole plating.
- 5. Clean the multi-layer PCB.
- 6. Condition the multi-layer PCB.
- 7. Activate the multi-layer PCB.
- 8. Clean the holes with LPKF ViaCleaner.
- 9. Copper-plate the multi-layer PCB.
- 10. Switch off the system.
- The multi-layer PCB has been galvanically through-hole plated.



For detailed information on galvanic through-hole plating, refer to the user manual of **Contac S4**.

Structuring the outer layers of the multi-layer PCB (with ProtoLaser)

- 1. Open the **cp2d* file you previously saved in CircuitPro PL.
- □ The opened project is displayed in the user guidance step *Layout*.
- 2. Switch to the user guidance step *Processing*.
- 3. In the pane Workflow setup expand the group Processing.
- 4. Right-click on [Top-Bot] Bottom Side and select Process from here.

Workflow setup	ф
Material composition	
Layout	
Scan fields	
D Tools	
Workflow	
Toolpaths	
Processing	
1. [Core] Structure Layer 3	
2. [Core] Structure Layer 2	
3. [Top-Bot] Galvanic THP	
4. [Top-Bot] Bottom Side	
Process selected>	
Process from here	

Fig. 190: Context menu Process from here

5. When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.

Ensure that the positioning holes are located in the right rear corner.

- 6. Click on 👑.
- □ The multi-layer PCB is fastened to the processing table by vacuum.
- 7. Click on [OK].
- □ The following dialog is displayed:

Copper th	nickness	-	×
Enter the copper thickness:	5 µm		
Γ	OK		Cancel

Fig. 191: Dialog Copper thickness

8. Enter the total copper thickness after galvanic through-hole plating into the input field.



You can determine the total copper thickness by performing one of the following steps:

- Calculate the copper thickness from the copper deposition rate in Contac S4 (approx. 0.15 µm/min).
- Measure the copper thickness with the ProtoMat S104.
- 9. Click on [OK].
- □ The dialog *Placement* is displayed.
- 10. Move the dialog *Placement* to get a better overview.
- 11. Click on the tab *Processing data*.
- 12. Place the processing data. The location of the layout **must match** the location of the multi-layer PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 13. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Bottom side (*BottomLayer*).
- For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the dialog *Test tool settings* is displayed.



In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 14. Click on [Resume].
- □ Structuring the Bottom side (*BottomLayer*) is started.
- 15. When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 16. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
- 17. When project placement is complete, click on [OK].

- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 18. Click on [Resume].
- Structuring the Top side (*TopLayer*) is started.
 The following message is displayed:

	Processing phase: Process drilling on a Pr	-	□ ×
0	Process drilling on a ProtoMat		
	OK Cancel		

Fig. 192: Message Processing phase: Process drilling on a ProtoMat

- 19. Click on [Cancel].
- 20. Remove the multi-layer PCB from the system.
- 21. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean it.
- 22. Rinse the multi-layer PCB with tap water and dry it with compressed air.
- The outer layers of the multi-layer PCB have been structured.

Drilling unplated through holes and cutting out the multi-layer PCB (with ProtoMat)

- 1. Open the *.cbf file you previously saved in CircuitPro PM.
- 2. Switch to the pane Processing.
- 3. Select *MountMaterialTop_1* from the drop-down list and click on \mathbb{P}_1 .
- □ The message *Processing Phase: MountMaterialTop_1* is displayed.
- Place the multi-layer PCB onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** and fasten it using adhesive tape.
 Ensure that the positioning holes are located in the right front corner.
- 5. Perform the following phases:
 - MaterialSettings 4
 - Placement 4
- □ The phases *ReadFiducialsTop_1*, *DrillingUnplated* and *ContourRouting* are performed.
- 6. When the message *Process finished* is displayed, remove the multi-layer PCB from the system.
- The unplated through holes have been drilled and the multi-layer PCB has been cut out.



For detailed information on ProtoMat phases refer to the how-to guides of ProtoMat.

The multi-layer PCB production is finished.

3.3 How to produce a multi-layer PCB with non-galvanic through-hole plating

This chapter describes how to create a **4-layer** PCB using a laser system, a circuit board plotter, a multi-layer press, and a non-galvanic through-hole plating system.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

Auxiliaries 1 × Base material FR4

System

•

• 1 set of tools for ProtoMat • ProtoLaser U4/S4/R4 MultiPress S

ProtoMat S or E

ProConduct

- Convection oven •
- (order code: 115877) LPKF Cleaner
- (order code: 115891) • Brush
- 2 × Thin Laminate 104 ML 0/18 µm, 229 mm ×

229 mm × 305 mm ×

1 mm, 18/18 µm

copper-clad

305 mm × 0.2 mm (order code: 119818)

(order code: 119574)

- 4 × Prepreg Type 2125, 200 mm × 275 mm × 0.1 mm (order code: 119572)
- Oil-free compressed air
- Tap water
- The following steps are performed in this tutorial:
- Preparing the data in CircuitPro PM •
- Preparing the core material and laminate materials (with ProtoMat) •
- Preparing the data in CircuitPro PL •
- Structuring the core material (with ProtoLaser) .
- Assembling and pressing the multi-layer stack (with MultiPress S) •
- Structuring the outer layers of the multi-layer PCB (with ProtoLaser) •
- Drilling unplated through holes and cutting out the multi-layer PCB (with ProtoMat) •
- Drilling plated through holes into the multi-layer PCB (with ProtoMat) •
- Plating through holes (with ProConduct)

Preparing the data in CircuitPro PM



For detailed information on preparing the data in CircuitPro PM refer to chapter 2.2.

- 1. In the tab *Templates* of the dialog *New document*, select the template: PM PL 4Layer ProConduct MultiPressS.cbf.
- 2. Click on File > Import or on $\stackrel{\bullet}{\ge}$.
- 3. Navigate to the folder that contains the data you wish to import. The example data used for this tutorial are located in the folder: C:\Program Files (x86) \LPKF Laser & Electronics \

LPKF CircuitPro PM\Example Data\UseCase Multilayer.

- Select the files you wish to import. In this example: Tutor_MultiLayer_4_Lagen.BOA, Tutor_MultiLayer_4_Lagen.BOT, Tutor_MultiLayer_4_Lagen.DRL, Tutor_MultiLayer_4_Lagen.LY2, Tutor_MultiLayer_4_Lagen.LY3 and Tutor_MultiLayer_4_Lagen.TOP.
- 5. Click on [Open].
- The data are automatically assigned to the correct layers and the following dialog is displayed:

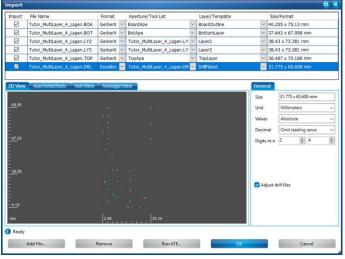


Fig. 193: Dialog Import | Assigned layers

- 6. Assign the holes to the layer DrillPlated (in the Layer/Template column).
- 7. Click on [OK].
- □ The imported data is displayed in the CAM view.

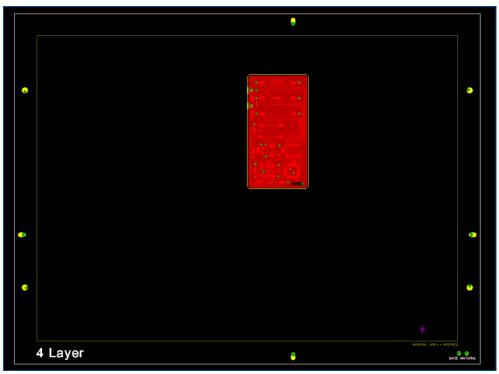


Fig. 194: CAM view multi-layer PCB



Multi-layer templates already include fiducials, register holes, working area frames etc., which are needed for precise layer assembly in the press mold.

- 8. Press and hold the <u>Ctrl</u> key and click on all holes that are not to be plated.
- □ The selected holes are highlighted.
- 9. Right-click on the layout and under Assign objects to layer select DrillUnplated.
- □ An additional layer is created automatically. Holes that are not to be plated are assigned to the layer *DrillUnplated*.
- 10. If desired, multiply the layout (in this example the layout is not multiplied).



The layout has to be within the multi-layer working area and prepreg material frame. It is preferable to move it **to the center**.

- 11. Select the layout.
- □ The layout is highlighted in gray.
- 12. Click on 🕂.
- 13. Using the left mouse button, drag and drop the layout to the desired location.
- □ The layout has been moved.
- 14. Click on *Toolpath* > *Technology Dialog*... or on \mathbb{N} .
- □ The dialog *Technology Dialog* is displayed.
- 15. Click on the tab *Contour routing* and select *Edge gaps* using the arrow buttons. This is usually the preferred method of contour routing.
- 16. Click on the tab *Pockets* and deactivate the check box *Process*, since there are no pockets in this project.
- 17. Click on [Start].
- □ The Computation results are displayed in a dialog.
- 18. Click on [Close].
- □ The toolpaths have been calculated.
- 19. Save the file.
- The data have been prepared in CircuitPro PM.

Preparing the core material and laminate materials (with ProtoMat)

- 1. Load the tool magazine and assign the tools to positions.
- Click on Machining > Process all or on .
- 3. Perform the following phases with the core material:
 - MountMaterial
 - MaterialSettings
 - Placement
- □ The phases *DrillFiducial* and *PrepareCore* are performed.
- 4. When the message *Processing Phase: DismountMaterial* is displayed, remove the core material from the system and click on [OK].
- 5. Rinse the core material with tap water and dry it with compressed air.
- 6. Perform the following phases with the first laminate:
 - MountLaminate
 - MaterialSettings_1
 - Placement_1
- □ The *PrepareLaminateBottom* phase is performed.
- 7. When the message *Processing Phase: DismountMaterial_1* is displayed, remove the first laminate from the system and click on [OK].
- 8. Perform the following phases with the second laminate:
 - MountLaminate_1
 - MaterialSettings_2
 - Placement_2
- □ The *PrepareLaminateTop* phase is performed.
- 9. When the message *Processing Phase: ProtoLaserStructuring* is displayed, remove the second laminate from the system and click on [Cancel] to temporarily stop the process.
- The core material and laminate materials have been prepared.



For detailed information on ProtoMat phases refer to the how-to guides of ProtoMat.

Preparing the data in CircuitPro PL

For detailed information on preparing the data in CircuitPro PL refer to chapter 2.2.

- 1. Switch to the user guidance step New.
- 2. Click on [Open project...].
- □ The dialog *Open* is displayed.
- 3. Navigate to the folder that contains the file you previously saved in CircuitPro PM.
- 4. Select *CBF document (*.cbf)* from the drop-down list.
- □ Only .cbf files are displayed.
- 5. Select the appropriate file and click on [Open].
- □ The dialog *Upgrade project* is displayed.
- 6. Perform the following steps:
 - Select 4-layer-Multilayer from the drop-down list.
 - Select your laser system from the drop-down list (in this example PL S4).
 - Activate the check box Supported by ProtoMat.
 - Select the template 4 Layers, ProtoLaser S4, ProtoMat, ProConduct, MultiPress S.

□ The dialog *Upgrade project* changes as follows:

		Upgrade project	
4		name\Desktop\Multi-layer F res to be upgraded. Choose a	
4-layer-Mu	ultilayer 💙 PL :	S4 🗸	✓ Supported by ProtoMat
LPKF temp 4 Laye		at, galvanic through-hole pl	ating
	ers, ProtoLaser S4, ProtoM ers, ProtoLaser S4, ProtoM	lat, galvanic through-hole pl lat. ProConduct	ating, MultiPress S
4 Laye	ers, ProtoLaser S4, ProtoM	lat, ProConduct, MultiPress S	
		lat E44, galvanic through-hol lat E44, ProConduct, MultiPro	
✓ Ignor	re fiducial and informatio	n layers	
		Upgrade	Close

Fig. 195: Upgrade selected

7. Click on [Upgrade] or double-click on the template.

- □ The user guidance step *Material* is displayed.
- 8. Select the material LPKF 4-Layer_ProConduct.

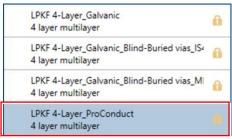


Fig. 196: Material selection

- 9. Click on [Select material] or double-click on the material.
- □ The layout of the opened project is displayed in the user guidance step *Layout*.
- 10. Switch to the user guidance step *Toolpaths*.
- 11. Click on [Compute toolpaths] or on **[**].
- □ The dialog *Computing toolpaths* is displayed:
- 12. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 13. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 14. Click on [Close].
- 15. Save your project.
- The data have been prepared in CircuitPro PL.

Structuring the core material (with ProtoLaser)

1. Switch to the user guidance step Processing.

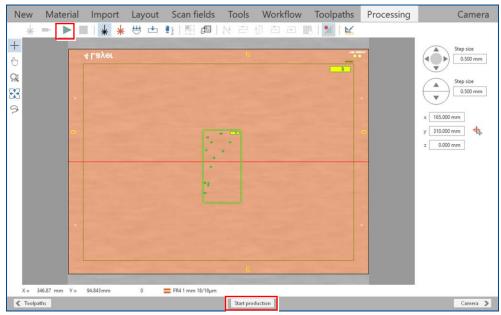
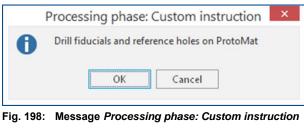


Fig. 197: User guidance step Processing

2. Click on local or click on [Start production].

□ The following message is displayed:



- 3. Click on [OK].
- □ The following message is displayed:

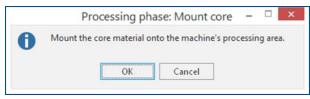


Fig. 199: Message Processing phase: Mount core

4. Place the core material with the **Bottom side** (*Layer3*) **facing upwards** in the center of the processing table.

Ensure that the positioning holes are located in the right rear corner.

- 5. Click on 🐫.
- □ The core material is fastened onto the processing table by vacuum.
- 6. Click on [OK].
- □ The dialog *Placement* is displayed:

	Placement	-		×
Material	Processing data			
	he processing area to move the active head to the ass se the buttons to set the left front and right rear corne			
	•			
× 000,5 r	nm x 305.5 mm			
y 018.0 r	nm y 247.0 mm			
Width	305 mm Length 229 mm Set default si	ze		
Use a calip	er to measure the material thickness			
Material th	ickness 1.036 mm Measure material thi	ckne	ss	
	Apply OK		Cano	el

Fig. 200: Dialog Placement

- 7. Move the dialog *Placement* to get a better overview.
- 8. Click on the tab *Processing data*.
- Place the processing data.
 The location of the layout **must match** the location of the core material and fiducials on the processing table.



For detailed information on project placement according to fiducial positions refer to chapter 1.1.

10. After project placement is complete click on [OK].

□ The laser system reads the fiducials on the Bottom side of the core material (*Layer3*).



After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.



For detailed information on material thickness measurement, refer to chapter 1.5.

When finished, the dialog Test tool settings is displayed.

i

In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter .

- 11. Click on [Resume].
- Structuring the Bottom side of the core material (*Layer3*) is started.
- When the message *Processing phase: Flip material* is displayed, turn the core material over around the symmetry axis of the system and click on [OK].
 Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 13. Place the processing data **matching** the location of the core material and fiducials on the processing table.
- 14. When project placement is complete, click on [OK].
- The laser system reads the fiducials on the Top side of the core material (*Layer2*). After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed. When finished, the dialog *Test tool settings* is displayed.
- 15. Click on [Resume].
- □ Structuring the Top side of the core material (*Layer*2) is started.

The following message is displayed:

	Processing phase: Press all layers	-	×
0	Press all layers		
	OK Cancel		

Fig. 201: Message Processing phase: Press all layers

- 16. Click on [Cancel] to temporarily stop the process.
- 17. Remove the core material from the system.
- 18. Spray the core material with LPKF Cleaner and use a brush to clean it.
- 19. Rinse the core material with tap water and dry it with compressed air.
- The core material has been structured.

Assembling and pressing the multi-layer stack (with MultiPress S)



For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.6.

- 1. Start the pre-heating process of the MultiPress S.
- 2. Prepare the materials.



Dry the materials in a convection oven at 100 $^{\circ}$ C (212 $^{\circ}$ F) for 30 minutes before assembling them in the press mold. Do not dry the prepreg material.

- 3. Assemble the multi-layer stack in the press mold.
- 4. Press the multi-layer stack.
- The multi-layer stack has been assembled and pressed.

Structuring the outer layers of the multi-layer PCB (with ProtoLaser)

- 1. Open the *cp2d file you previously saved in CircuitPro PL.
- □ The opened project is displayed in the user guidance step *Layout*.
- 2. Switch to the user guidance step *Processing*.
- 3. In the pane Workflow setup expand the group Processing.
- 4. Right-click on [Top-Bot] Bottom Side and select Process from here.

Workflow setup	џ
Material composition	
Layout	
Scan fields	
▷ Tools	
Workflow	
Toolpaths	
4 Processing	
1. [Core] Structure Layer 3	
2. [Core] Structure Layer 2	
3. [Top-Bot] Pressing	
4. [Top-Bot] Bottom Side	
▷ 5. [To Check tool settings	
▷ 6. [To <process selected=""></process>	
Process from here	

Fig. 202: Context menu Process from here

5. When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.

Ensure that the positioning holes are located in the right rear corner.

- 6. Click on 🐫.
- □ The multi-layer PCB is fastened to the processing table by vacuum.
- 7. Click on [OK].

- □ The dialog *Placement* is displayed.
- 8. Move the dialog *Placement* to get a better overview.
- 9. Click on the tab Processing data.
- 10. Place the processing data. The location of the layout **must match** the location of the multi-layer PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 11. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Bottom side (*BottomLayer*).



For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the dialog Test tool settings is displayed.

In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 12. Click on [Resume].
- □ Structuring the Bottom side (*BottomLayer*) is started.
- 13. When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 14. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
- 15. When project placement is complete, click on [OK].
- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 16. Click on [Resume].
- Structuring the Top side (*TopLayer*) is started.
 The message *Processing phase: Process drilling on a ProtoMat* is displayed.
- 17. Click on [Cancel].
- 18. Remove the multi-layer PCB from the system.
- 19. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean it.
- 20. Rinse the multi-layer PCB with tap water and dry it with compressed air.
- The outer layers of the multi-layer PCB have been structured.

Drilling unplated through holes and cutting out the multi-layer PCB (with ProtoMat)

- 1. Open the *.cbf file you previously saved in CircuitPro PM.
- 2. Switch to the pane *Processing*.
- 3. Select *MountMaterialTop* from the drop-down list and click on .
- □ The message *Processing Phase: MountMaterialTop* is displayed.
- Place the multi-layer PCB onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** and fasten it using adhesive tape.
 Ensure that the positioning holes are located in the right front corner.
- 5. Perform the following phases:
 - MaterialSettings_3
 - Placement_3
- □ The phases *ReadFiducialsTop*, *MarkingDrills*, *DrillingUnplated* and *ContourRouting* are performed.
- 6. When the message *Processing Phase: DismountMaterial_2* is displayed, remove the multi-layer PCB from the system and click on [OK].
- 7. Rinse the multi-layer PCB with tap water and dry it using compressed air.
- The unplated through holes have been drilled and the multi-layer PCB has been cut out.

-	_	
		<u> </u>
		-

The multi-layer PCB **must remain attached to the original material**. Do **not** break or cut the breakout tabs **yet**.

Drilling plated through holes into the multi-layer PCB (with ProtoMat)

- 1. When the message *Processing Phase: ApplyProtectionFilm* is displayed, apply the protective film onto both sides of the multi-layer PCB and click on [OK].
- □ The message *Processing Phase: MountMaterialTop_1* is displayed.
- Place the multi-layer PCB with the **Top side** (*TopLayer*) **facing upwards** onto the system's processing table, fasten it with adhesive tape and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- 3. Perform the *Placement_4* phase.
- □ The phases *ReadFiducialsTop_1* and *DrillingPlated* are performed.
- 4. When the message *Processing Phase: DismountMaterial_3* is displayed, remove the multi-layer PCB from the system and check it for burrs and residual material within the holes.



Check whether the fiducials are covered by the protective film and expose them if necessary.

The plated through holes have been drilled into the multi-layer PCB.

Plating through holes (with ProConduct)

- 1. Place the mat onto the vacuum table (ProtoMat mounted or standalone).
- 2. Place the multi-layer PCB onto the mat.
- 3. Apply the ProConduct paste along the edge of the multi-layer PCB.
- 4. Spread the paste on the whole surface of the multi-layer PCB (do not spread over the fiducials).
- 5. Turn on the extraction system on the highest setting and wait for at least 30 seconds.
- 6. Spread the paste remaining on the multi-layer PCB surface with the **vacuum turned on**.
- 7. Switch off the extraction system.
- 8. Turn the multi-layer PCB over and repeat the steps 1 to 7 on the other side.
- Slowly peel off the protective film at an angle of 90° from both sides of the multilayer PCB.
- 10. Place the multi-layer PCB horizontally into the convection oven for 30 minutes (160 °C / 320 °F).
- 11. Remove the multi-layer PCB and let it cool down to ambient temperature.
- 12. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean the multi-layer PCB.
- 13. Rinse the multi-layer PCB with tap water and dry it with compressed air.
- 14. Cut out or break out the multi-layer PCB from the board.
- The through holes have been plated.

For detailed information on non-galvanic through-hole plating refer to the process description of **ProConduct**.

The multi-layer PCB production is finished.

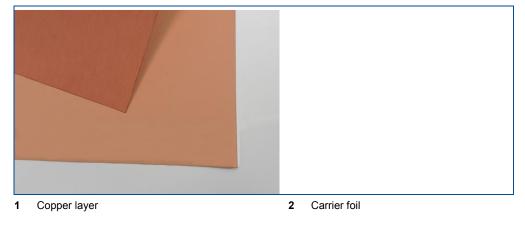
3.4 How to produce a multi-layer PCB with blind vias and buried vias

This chapter describes how to create a **4-layer** PCB with blind vias and buried vias using a laser system, a circuit board plotter, a multi-layer press, and a galvanic through-hole plating system.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Co	nsumables	Auxiliaries	System
•	1 × Base material IS400 235 mm × 305 mm × 0.46 mm, 18/18 μm copper-clad (order code 10092752)	 1 set of tools for ProtoMat Spiral Drill Blind via for ProtoMat 1/8", 38 mm, d = 0.20 mm (order code 10099689) 	 ProtoLaser U4/S4/R4 MultiPress S ProtoMat S or E Contac S4
•	2 × Copper foil 0/5 μm, 212 mm × 282 mm × 0.2 mm with carrier foil (order code 10097444) 2 × Prepreg IS400, type 1080, 205 mm × 275 mm × 0.1 mm (order code 10092750)	 Convection oven (order code 115877) LPKF Cleaner (order code 115891) Brush Portable hand-held microscope Oil-free compressed air Tap water 	

The copper foil consists of a copper layer and the carrier foil that is attached to the copper layer. The side with the copper layer has a darker color tone, the side with the carrier foil has a lighter color tone. It is important to know the difference of both sides. The following figure shows both sides of the copper foil.



The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PM
- Drilling buried vias into the core material (with ProtoMat)
- Galvanic through-hole plating the core material (with Contac S4)
- Preparing the data in CircuitPro PL
- Structuring the core material (with ProtoLaser)
- Assembling and pressing the multi-layer stack (with MultiPress S)
- Drilling plated through holes into the multi-layer PCB (with ProtoMat)
- Drilling blind vias into the multi-layer PCB (with ProtoLaser)

- Galvanic through-hole plating the multi-layer PCB (with Contac S4)
- Structuring the outer layers of the multi-layer PCB (with ProtoLaser)
- Drilling unplated through holes and cutting out the multi-layer PCB (with ProtoMat)

i

It is essential that you read the design guidelines described in chapter 1.8 before starting this tutorial.

Preparing the data in CircuitPro PM

i

For detailed information on preparing the data in CircuitPro PM refer to chapter 2.2.

- 1. In the tab *Templates* of the dialog *New document*, select the template: *PM_PL_4Layer_GalvanicTHP_MultiPressS_Blind (IS400).cbf*.
- 2. Click on *File* > *Import* or on \clubsuit .
- 3. Navigate to the folder that contains the data you want to import.
- 4. Select the files you want to import and click on [Open].



If you are not sure which files to select, then select all of them. You will see the preview of the files later and you can deselect those you do not need.

□ The following dialog is displayed:

port	File Name	Format	Aperture/Tool List		Layer/Template		Size/F	format		
	4L_BindBuriedVia.BOT	GerberX 🚩	BotApe	×	BottomLayer	~	61.40	1 x 39.502 mm		
	4L_BindBuriedVia.d12	GerberX 🛩	4L_BlindBuriedVia.d12	4	Blind via (Top-L2)	~	60.599	9 x 39.101 mm	i .	1
	4L_BindBuriedVia.d14	GerberX 🗠	4L_BlindBuriedVia.d14	4	DrilPlated	>	60.701	1 x 39.202 mm	1	
	4L_BindBuriedVia.d23	GerberX 🛩	4L_BlindBuriedVia.d23	~	Buried via (L2-L3)	~		1 x 39.202 mm		
	4L_BindBuriedVia.d34	GerberX ~	4L_BlindBuriedVia.d34	4	Bind via (Bot-L3)	>		9 x 39.101 mm	1	
	4L_BindBuriedVia.LY2	GerberX 🗠	4L_BindBuriedVia.LY2	~	Layer2	~		5.001 mm		
Ø	4L_BindBuriedVia.LY3	GerberX 🗠	4L_BlindBuriedVia.LY3	~	Layer3	~	70 x 4	5.001 mm		
View	Apertures/Tools Text V	ew Message	View			Ger	eral	Options		
						Size		61.401 x 39.50	2 mm	
67.50						Uni		Inches		1
						1.00				
						Vali	ues	Absolute		
						Dec	timal	Omit leading	zeros	
						Dig	its m.n	2	4	1
47.75										
28.00										
	169.85	190.32	210.78							
n Ready										

Fig. 203: Dialog Import | Assigned layers

 Assign the data to the corresponding layers (in the Layer/Template column). Pay special attention to correct assignment of drill files, since multiple drill layers are used in this process.



If the drop-down lists in the *Layer/Template* column cannot be activated, proceed as follows:

- Click on the sub-tab Options (on the right side of the dialog Import).
- Deactivate Use layer name.
- Activate Apply to all Gerber files.
- 6. Click on [OK].

□ The imported data are displayed in the CAM view.



Fig. 204: CAM view multi-layer PCB with blind vias and buried vias

- 7. If desired, multiply the layout (in this example the layout is not multiplied).
- 8. Select the layout.
- □ The layout is highlighted in gray.
- 9. Click on 🕂.
- 10. Keep the left mouse button pressed to drag and drop the layout to the desired location.
- □ The layout has been moved.
- 11. Click on Toolpath > Technology Dialog... or on \mathbb{N} .
- □ The dialog *Technology Dialog* is displayed.
- 12. Click on [Start].
- □ The *Computation results* are displayed in a dialog.
- 13. Click on [Close].
- □ The toolpaths have been calculated.
- 14. Save the file.
- The data have been prepared in CircuitPro PM.

Drilling buried vias into the core material (with ProtoMat)

- 1. Load the tool magazine and assign the tools to positions.
- Click on Machining > Process all or on .
- 3. Perform the following phases with the core material:
 - MountCore
 - MaterialSettings
 - Placement
- □ The phases *DrillFiducial*, *PrepareCore*, *Marking DrillBuriedVias* (*L2-L3*) and *DrillBuriedVias* (*L2-L3*) are performed.
- When the message Processing Phase: ThroughHolePlating_1 is displayed, remove the core material from the system and click on [Cancel] to temporarily stop the process.
- The buried vias have been drilled into the core material.



For detailed information on ProtoMat phases refer to the how-to guides of ProtoMat.

Galvanic through-hole plating the core material (with Contac S4)

i

During and after galvanic plating, **handle the core material with care**. The core material is thin and cracks in the copper barrels can occur, if it is bent too much.

- 1. Switch on the system.
- 2. Select a profile.

i

The recommended total copper thickness after through-hole plating is $30 \ \mu m$ to $35 \ \mu m$.

- 3. Start the process.
- 4. Prepare the core material for through-hole plating.
- 5. Clean the core material.
- 6. Condition the core material.
- 7. Activate the core material.
- 8. Clean the holes with LPKF ViaCleaner.
- 9. Copper-plate the core material.
- 10. Switch off the system.
- The core material has been galvanically through-hole plated.



For detailed information on galvanic through-hole plating, refer to the user manual of **Contac S4**.

Preparing the data in CircuitPro PL

For detailed information on preparing the data in CircuitPro PL refer to chapter 2.2.

- 1. Switch to the user guidance step New.
- 2. Click on [Open project...].
- □ The dialog *Open* is displayed.
- 3. Navigate to the folder that contains the file you previously saved in CircuitPro PM.
- 4. Select CBF document (*.cbf) from the drop-down list.
- □ Only .cbf files are displayed.
- 5. Select the appropriate file and click on [Open].
- □ The dialog *Upgrade project* is displayed.
- 6. Perform the following steps:
 - Select 4-layer-Multilayer from the drop-down list.
 - Select your laser system from the drop-down list (in this example *PL U4*).
 - Activate the check box Supported by ProtoMat.
 - Select the template 4 Layers, ProtoLaser U4, ProtoMat, galvanic through-hole plating, MultiPress S, blind-buried vias (IS400).

□ The dialog *Upgrade project* changes as follows:

Fig. 205: Upgrade selected

7. Click on [Upgrade] or double-click on the template.

- □ The user guidance step *Material* is displayed.
- 8. Select the material LPKF 4-Layer_Galvanic_Blind-Buried vias_IS400.

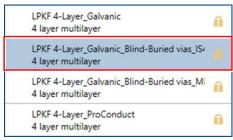


Fig. 206: Material selection

- 9. Click on [Select material] or double-click on the material.
- □ The layout of the opened project is displayed in the user guidance step *Layout*.
- 10. Switch to the user guidance step *Toolpaths*.

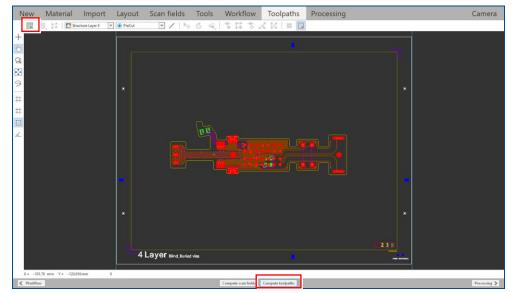
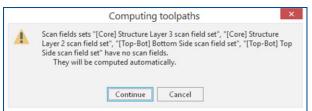


Fig. 207: Opened project

11. Click on [Compute toolpaths] or on [2].

The following dialog is displayed:



- Fig. 208: Dialog Computing toolpaths
- 12. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 13. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 14. Click on [Close].
- 15. Save your project.
- The data have been prepared in CircuitPro PL.



1. Switch to the user guidance step *Processing*.

Ne	
4 0 0 0 0 4	
	1212/Timm V z 244.482mm 6 = 6400.846 mm 16/18pm puts
	209: User guidance step Processing
2.	Click on b or click on [Start production].
	The following message is displayed:
-	
	Drill fiducials, reference holes and buried vias on ProtoMat
	OK Cancel
Fig.	210: Message Processing phase: Custom instruction
3.	Click on [OK].
	When the message <i>Processing phase: Galvanic through-hole plating</i> is displayed click on [OK].
	When the message <i>Processing phase: Mount material bottom</i> is displayed, place the core material with the Bottom side (<i>Layer3</i>) facing upwards in the center of the processing table.
	Ensure that the positioning holes are located in the right rear corner.

- 4. Click on 🐫.
- □ The core material is fastened onto the processing table by vacuum.
- 5. Click on [OK].
- □ The following dialog is displayed:

Copper t	hickness	-	×
Enter the copper thickness:	18 µm		
Г	OK		Cancel

Fig. 211: Dialog Copper thickness

6. Enter the total copper thickness after galvanic through-hole plating into the input field.



You can determine the total copper thickness by performing one of the following steps:

- Calculate the copper thickness from the copper deposition rate in Contac S4 (approx. 0.15 µm/min).
- Measure the thickness with the ProtoMat S104.
- 7. Click on [OK].
- □ The dialog *Placement* is displayed:

Placement – 🗆	×				
Material Processing data					
Click into the processing area to move the active head to the associated position. Use the buttons to set the left front and right rear corner of the material.					
x 000.5 mm x 305.5 mm					
y 018.0 mm y 247.0 mm					
Width 305 mm Length 229 mm Set default size					
Use a caliper to measure the material thickness					
Material thickness 0.686 mm Measure material thickness					
Apply OK Can	cel				

Fig. 212: Dialog Placement

- 8. Move the dialog *Placement* to get a better overview.
- 9. Click on the tab *Processing data*.
- 10. Place the processing data.

The location of the layout **must match** the location of the core material and fiducials on the processing table.



For detailed information on project placement according to fiducial positions refer to chapter 1.1.

- 11. After project placement is complete, click on [OK].
- □ The laser system reads the fiducials on the Bottom side of the core material (*Layer3*).



For detailed information on fiducial recognition refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.



When finished, the dialog Test tool settings is displayed.



In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

12. Click on [Resume].

- Structuring the Bottom side of the core material (*Layer3*) is started.
- 13. When the message *Processing phase: Flip material* is displayed, turn the core material over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 14. Place the processing data **matching** the location of the core material and fiducials on the processing table.
- 15. When project placement is complete, click on [OK].
- The laser system reads the fiducials on the Top side of the core material (*Layer2*). After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed. When finished, the dialog *Test tool settings* is displayed.
- 16. Click on [Resume].
- Structuring the Top side of the core material (*Layer2*) is started. The following message is displayed:

	Processing phase: Press all layers 🛛 – 🗖 🗙					
0	Press all layers					
	OK Cancel					

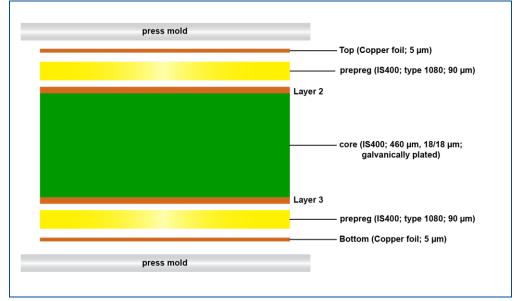
Fig. 213: Message Processing phase: Press all layers

- 17. Click on [Cancel] to temporarily stop the process.
- 18. Remove the core material from the system.
- 19. Spray the core material with LPKF Cleaner and use a brush to clean it.
- 20. Rinse the core material with tap water and dry it with compressed air.
- The core material has been structured.

Assembling and pressing the multi-layer stack (with MultiPress S)



For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.6.

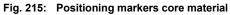


The materials are assembled in the press mold according to the following figure:

Fig. 214: Assembly of the materials in the press mold

Two outer positioning markers (1) and two inner positioning markers (2) on the core material help you with the correct positioning of materials during the assembly in the press mold.





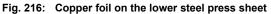
- 1 Outer positioning marker (for copper foil)
- 2 Inner positioning marker (for prepreg material)

- 1. Start the pre-heating process of the MultiPress S.
- 2. Prepare the materials.

Dry the materials in a convection oven at 100 °C (212 °F) for 30 minutes before assembling them in the press mold. **Do not dry the prepreg material**.

- 3. Assemble the lower part of the aluminum press mold, the press cushion and the steel press sheet.
- 4. Place the copper foil with the **carrier foil downwards** at the center of the steel press sheet.







The copper foil is extremely thin and needs to be handled with care!



5. Place the prepreg material at the center of the copper foil.

Fig. 217: Prepreg material on the copper foil

6. Place the core material on the dowel pins of the press mold.

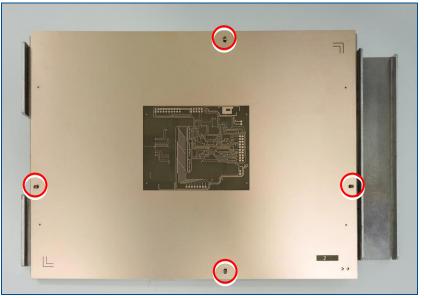


Fig. 218: Core material on the dowel pins

7. Place the prepreg material on the core material. Align it with the inner positioning markers (2).

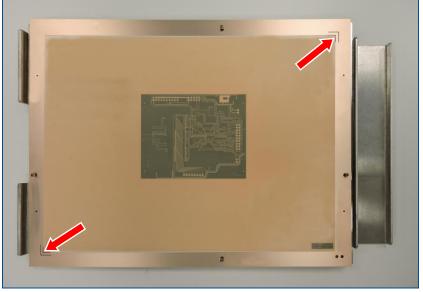


Fig. 219: Prepreg material on the core material

8. Place the copper foil on the prepreg material. Align it with the outer positioning markers (1).

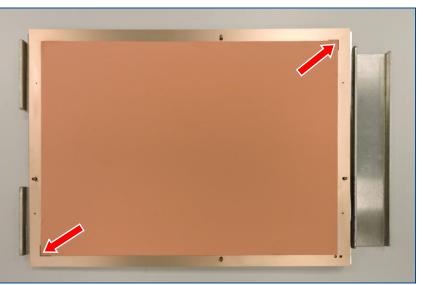


Fig. 220: Copper foil on the prepreg material

The copper foil is extremely thin and needs to be handled with care!

- 9. Assemble the steel press sheet, the press cushion and the upper part of the aluminum press mold.
- 10. Select the profile IS400 (available in the firmware version 1.12).
- 11. Press the multi-layer stack.
- The multi-layer stack has been assembled and pressed.

Using sealing rings to assemble the multi-layer stack

You do not need to use sealing rings when producing a 4-layer PCB with blind vias and buried vias.

When producing a 6-layer or an 8-layer PCB with blind vias and buried vias, you need to use two sealing rings wherever 2 sheets of material are used together – e.g. between two core materials (see page 56).

Drilling plated through holes into the multi-layer PCB (with ProtoMat)



Do not remove the carrier foil from the top and from the bottom of the multi-layer PCB yet!

- 1. Open the *.cbf file you previously saved in CircuitPro PM.
- 2. Switch to the pane Processing.
- 3. Select *MountMaterialBottom* from the drop-down list and click on $[]_{I}^{\bullet}$.
- □ The message *Processing Phase: MountMaterialBottom* is displayed.
- Place the multi-layer PCB onto the system's processing table with the Bottom side (*BottomLayer*) facing upwards and fasten it using adhesive tape.
 Ensure that the positioning holes are located in the right rear corner.
- 5. Perform the following phases:
 - MaterialSettings_1
 - Placement_2
- □ The phases *ReadFiducialsBottom*, *MarkingDrillPlated* and *DrillingPlated* are performed.
- 6. When the message *Processing Phase: ProtoLaserBlindViaDrilling* is displayed, remove the multi-layer PCB from the system and click on [Cancel] to temporarily stop the process.
- 7. Rinse the multi-layer PCB with tap water and dry it with compressed air.
- The plated through holes have been drilled into the multi-layer PCB.

For detailed information on ProtoMat phases refer to the how-to guides of ProtoMat.

Drilling blind vias into the multi-layer PCB (with ProtoLaser)

- 1. Click on *File* > *Open* or \ge .
- □ The dialog *Open* is displayed.
- 2. Select the file you previously saved in CircuitPro PL.
- 3. Click on [Open].
- □ The opened project is displayed in the user guidance step *Processing*.
- 4. In the pane Workflow setup expand [Top-Bot] Drilling Bottom.

Workflow setup	д		
Material composition			
Layout			
Scan fields			
D Tools			
V Workflow			
> Toolpaths			
Processing			
1. [Core] Structure Layer 3			
2. [Core] Structure Layer 2			
 3. [Top-Bot] Drilling Bottom 			
1. Press All Layers			
2. ProtoMat Drilling Plated TH			
3. Mount Material Bottom			
4. Material Placement			
5. Read Fiducials Bottom			
6. Drill Blind vias (Bottom-L3)			
4. [Top-Bot] Drilling Top			
5. [Top-Bot] Structure Bottom			
6. [Top-Bot] Structure Top			

Fig. 221: Pane Workflow setup

5. Right-click on Mount Material Bottom.

- Workflow setup д Material composition Layout Scan fields D Tools Workflow Toolpaths Processing 1. [Core] Structure Layer 3 2. [Core] Structure Layer 2 ▲ 3. [Top-Bot] Drilling Bottom 1. Press All Layers 2. ProtoMat Drilling Plated TH 3. Mount Material Bottom <Process selected> Process from here 4. [Top-Bot] Drilling Top 5. [Top-Bot] Structure Bottom 6. [Top-Bot] Structure Top
- □ The following context menu is displayed:

Fig. 222: Context menu Process from here

6. Select Process from here.

□ When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*TopLayer*) **facing upwards** onto the processing table.

Ensure that the positioning holes are located in the right rear corner.

- 7. Click on 🐫.
- □ The multi-layer PCB is fastened to the processing table by vacuum.
- 8. Click on [OK].
- □ The dialog *Placement* is displayed.
- 9. Move the dialog *Placement* to get a better overview.
- 10. Click on the tab Processing data.
- 11. Place the processing data. The location of the layout **must match** the location of the multi-layer PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

12. After project placement is complete click on [OK].

i

□ The laser system reads the fiducials on the Bottom side (*BottomLayer*). For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the following dialog is displayed:

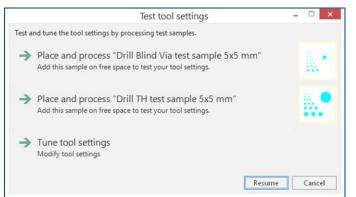


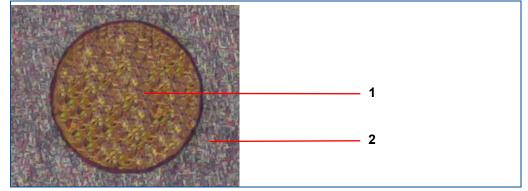
Fig. 223: Dialog Test tool settings

It is **recommended** that you process the **blind via test sample** on your multi-layer PCB before you continue with production.

13. Select Place and process "Drill Blind Via test sample 5x5 mm".

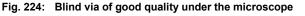
For detailed information on placing and processing the test sample, refer to chapter 1.5.

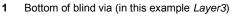
- 14. Check the created test sample with a portable hand-held microscope.
- Select a blind via on the test sample. This should have the same diameters as the blind vias of your future project.
 For the diameters of blind vias on the test sample see page 183.
- 16. Use the following conditions to determine whether the processing quality is satisfactory:
 - The blind via has reached the target copper layer.
 - The copper surface at the bottom of the blind via is clean (e.g., free of epoxy resin remains) and is not punctured.



2

The following figure shows a blind via of good quality:





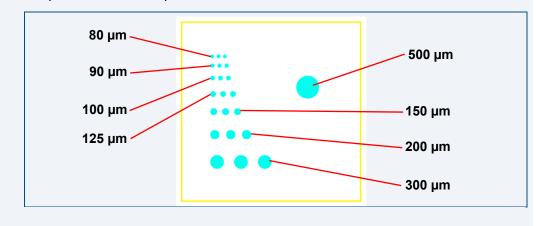
Bottom side of the multi-layer PCB (in this example *BottomLayer*)

17. If you are not satisfied with the result, adjust the tool settings.

- For detailed information on tuning the tool settings, refer to chapter 1.5.
- 18. If you are satisfied with the result, click on [Resume].
- □ The laser system drills the blind vias on the Bottom side (*BottomLayer*).
- When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 20. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
- 21. When project placement is complete, click on [OK].
- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 22. Click on [Resume].
- □ The laser system drills the blind vias on the Top side (*TopLayer*).
- 23. When the message *Processing phase: Galvanic through-hole plating* is displayed, remove the multi-layer PCB from the system and click on [Cancel] to temporarily stop the process.
- 24. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean it.
- 25. Rinse the multi-layer PCB with tap water and dry it with compressed air.
- The blind vias have been drilled into the multi-layer PCB.

Diameters of blind vias on the test sample

The processed test sample includes blind vias with different diameters:



Galvanic through-hole plating the multi-layer PCB (with Contac S4)

- 1. Switch on the system.
- 2. Select a profile.

The recommended **total copper thickness** after through-hole plating is **30 \mum** to **35 \mum**.

- 3. Start the process.
- 4. Prepare the multi-layer PCB for through-hole plating.
- 5. Clean the multi-layer PCB.
- 6. Condition the multi-layer PCB.
- 7. Activate the multi-layer PCB.
- Remove the carrier foil on the bottom side and on the top side of the multi-layer PCB.
- 9. Clean the holes with LPKF ViaCleaner.
- 10. Copper-plate the multi-layer PCB.
- 11. Switch off the system.
- The multi-layer PCB has been galvanically through-hole plated.



For detailed information on galvanic through-hole plating, refer to the user manual of **Contac S4**.

Structuring the outer layers of the multi-layer PCB (with ProtoLaser)

- 1. Open the *cp2d file you previously saved in CircuitPro PL.
- □ The opened project is displayed in the user guidance step *Layout*.
- 2. Switch to the user guidance step *Processing*.
- 3. In the pane Workflow setup expand the group Processing.
- 4. Right-click on [Top-Bot] Structure Bottom and select Process from here.

 Material composition Layout Scan fields Tools Workflow 								
 Scan fields Tools 								
▷ Tools								
Workflow								
Toolpaths								
Processing								
1. [Core] Structure Layer 3								
2. [Core] Structure Layer 2								
3. [Top-Bot] Drilling Bottom								
4. [Top-Bot] Drilling Top								
5. [Top-Bot] Structure Bottom								
▷ 6. Check tool settings								
<process selected=""></process>								
Process from here								
L								

Fig. 225: Context menu Process from here

5. When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.

Ensure that the positioning holes are located in the right rear corner.

- 6. Click on 🐫.
- □ The multi-layer PCB is fastened to the processing table by vacuum.
- 7. Click on [OK].
- □ The following dialog is displayed:

Copper t	hickness		>
Enter the copper thickness:	5 µm]	
Г	OK	Car	

Fig. 226: Dialog Copper thickness

8. Enter the total copper thickness after galvanic through-hole plating into the input field.



You can determine the total copper thickness by performing one of the following steps:

- Calculate the copper thickness from the copper deposition rate in Contac S4 (approx. 0.15 µm/min).
- Measure the copper thickness with the ProtoMat S104.
- 9. Click on [OK].
- □ The dialog *Placement* is displayed.
- 10. Move the dialog *Placement* to get a better overview.
- 11. Click on the tab *Processing data*.
- 12. Place the processing data. The location of the layout **must match** the location of the multi-layer PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 13. After project placement is complete, click on [OK].
- □ The laser system reads the fiducials on the Bottom side (*BottomLayer*).



After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the dialog *Test tool settings* is displayed.



In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 14. Click on [Resume].
- □ Structuring the Bottom side (*BottomLayer*) is started.
- 15. When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 16. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
- 17. When project placement is complete, click on [OK].

- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 18. Click on [Resume].
- Structuring the Top side (*TopLayer*) is started.
 The following message is displayed:

	Processing phase: Process drilling on a Pr	-	□ ×
0	Process drilling on a ProtoMat		
	OK Cancel		

Fig. 227: Message Processing phase: Process drilling on a ProtoMat

- 19. Click on [Cancel].
- 20. Remove the multi-layer PCB from the system.
- The outer layers of the multi-layer PCB have been structured.

Drilling unplated through holes and cutting out the multi-layer PCB (with ProtoMat)

- 1. Open the *.cbf file you previously saved in CircuitPro PM.
- 2. Switch to the pane *Processing*.
- 3. Select *MountMaterialTop* from the drop-down list and click on $[]_1^{\bullet}$.
- □ The message *Processing Phase: MountMaterialTop* is displayed.
- Place the multi-layer PCB onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** and fasten it using adhesive tape.
 Ensure that the positioning holes are located in the right front corner.
- 5. Perform the following phases:
 - MaterialSettings_2
 - Placement_3
- □ The phases *ReadFiducialsTop*, *MarkingDrillUnplated*, *DrillingUnplated* and *ContourRouting* are performed.
- 6. When the message *Process finished* is displayed, remove the multi-layer PCB from the system.
- 7. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean it.
- 8. Rinse the multi-layer PCB with tap water and dry it with compressed air.
- The unplated through holes have been drilled and the multi-layer PCB has been cut out.



For detailed information on ProtoMat phases refer to the how-to guides of ProtoMat.

The multi-layer PCB production is finished.

Other how-to examples



4 Other how-to examples

The production of PCBs as described in this chapter is only available in the license level Advanced of CircuitPro PL. If you have any questions contact the LPKF sales department.

This chapter contains more how-to examples of varying degrees of difficulty. To create a polyimide stencil as well as to structure a solder mask, basic knowledge of handling laser systems and the system software is sufficient. To create a flex-rigid PCB and an RF PCB, advanced knowledge of multi-layer PCBs is required.

4.1 Producing a polyimide stencil

This chapter describes how to create a polyimide stencil using a laser system.

Ensure that the following consumables are available before performing the described tasks:

Consumables

System

ProtoLaser U4/S4/R4

 Polyimide Foil A4, 125 µm, format for Stencils (order code: 108321)

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PL
- Cutting the stencil (with ProtoLaser)

Preparing the data in CircuitPro PL

For detailed information on preparing the data in CircuitPro PL refer to chapter 2.1.

- 1. In the user guidance step New select Single-sided from the drop-down list.
- 2. Select your laser system from the drop-down list (in this example PL U4).
- □ A list of templates for single-sided materials is displayed:

Recent projects	Single-sided Y PL U4 LPKF templates Single-sided, Top, ProtoLaser U4 Single-sided, Top, ProtoPint, A4 for QR Stencil for ProtoPint, A4 for QR	e 114	y ProtoMat Stencil for ProtoPrint, size A4 for QR 266 x 380 frame, prepared for ProtoLaser U4	
Open project	Single-sided, Top, ProtoLaser U4 Single-sided, Rottom, ProtoLaser	e 114		
Recent templates				

Fig. 228: List of single-sided templates

- 3. Select the template Stencil for ProtoPrint, A4 for QR 266 x 380, ProtoLaser U4.
- 4. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 5. Select Foil from the drop-down list.
- 6. Select the material Polyimide Foil, 0.125 mm.

New	Material	Import						Camera
Ξ	Foil	~] 📮 📮				
C*	Copper foil Foil, 0.005 mm		â	Material name Polyi	mide			
×	Polyimide Foil, 0,125 mm		8	Total thickness 0.12	!5 mm			
				Core 0.125	mm		Top Layer	
< N	ew			Disc	ard Save	Select material		Import >

Fig. 229: Material selection

- 7. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 8. Click on 📑
- □ The dialog *Open* is displayed.
- 9. Navigate to the folder that contains the data you want to import. For the example data that are used for this tutorial, refer to the folder: C:\Program Files\LPKF Laser & Electronics AG\ LPKF CircuitPro PL\Example Data\UseCase PolyimideStencil.
- 10. Select the files you want to import. In this example: Tutor.SPT.
- 11. Click on [Open].

□ The data are automatically assigned to the correct layer and the user guidance step *Import* is displayed:

New	Material	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing Camera	3
	20 Apertures/To 0120	ta		berX jet JerPasteTop	~	PasteTopApe Size/Format 69.129 x 38.76 mm		File Format Unit Inches Values Absolute Decimal Omit leading zeros Digits m.n 2	
▲ M	aterial				Discard	Import		Layout 💙	,

Fig. 230: User guidance step Import

- 12. Click on [Import].
- □ The layout is displayed in the user guidance step *Layout*.

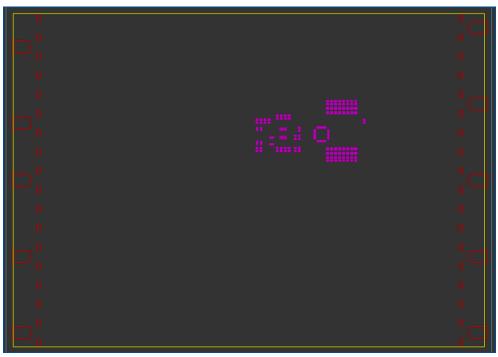


Fig. 231: Layout of a polyimide stencil

The layout needs to be **moved to the center** of the base material.

- 13. Select the entire layout.
- □ The layout is highlighted and changes its color.
- 14. Perform one of the following steps:
 - Right-click on the highlighted layout and select *Move object*.
 - Click on
- 15. Click on a point in the original layout to specify the reference point.

i

- □ A copy of the layout is displayed.
- 16. Move the copied layout with the mouse to the desired position.
- 17. When you reach the desired position, left-click to specify the target point.
- 18. In the user guidance step *Layout* click anywhere on the black background or press [Esc] to disable the *Move object* function.
- □ The layout has been moved to the desired position.
- 19. Switch to the user guidance step *Toolpaths*.
- □ The dialog *Computing toolpaths* is displayed.
- 21. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 22. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 23. Click on [Close].
- 24. Save your project.
- The data have been prepared in CircuitPro PL.

Cutting the stencil (with ProtoLaser)

1. Switch to the user guidance step *Processing*.

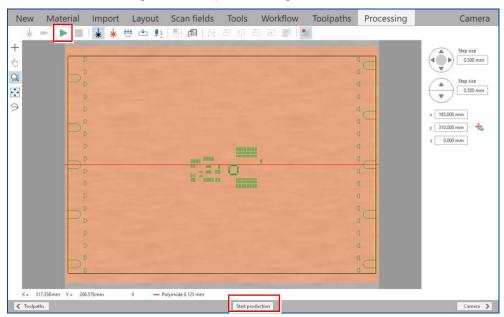


Fig. 232: User guidance step Processing

- 2. Click on [Start production] or on ▶.
- 3. When the message *Processing phase: Mount material* is displayed, place the polyimide foil in the center of the processing table.
- Click on ⁴⁴.
- □ The polyimide foil is fastened onto the processing table by vacuum.
- 5. Click on [OK].
- □ The dialog *Placement* is displayed.
- 6. Move the dialog *Placement* to get a better overview.

7. Determine the processing area. The base material position and base material size **must match** the processing area used by the CircuitPro PL software.

For detailed information on project placement by determining the processing area, refer to chapter 1.1.

- 8. Click on the tab *Processing data*.
- 9. Click on [Center on material].
- 10. Click on [OK].
- □ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

i

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, cutting the polyimide foil is started.

- 11. When the message *Board production finished* is displayed, remove the polyimide stencil from the system.
- 12. Check whether the cut stencil can be fully removed from the residual polyimide foil.
- 13. Carefully detach the stencil from the residual polyimide foil.
- 14. Clean the processing area.
- The stencil has been cut.

The polyimide stencil production is finished.

ProtoLaser U4/S4/R4

MultiPress S

Contac S4

System

4.2 Producing a flex-rigid PCB

This chapter describes how to create a flex-rigid PCB using a laser system, a multi-layer press, and a galvanic through-hole plating system.

A flex-rigid PCB merges the properties of rigid and flexible PCBs and consists of three different materials.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

- 1 × Base material FR4 229 mm × 305 mm × 1.5 mm, 0/18 µm copper-clad (order code 115968)
- 1 × Thin Laminate ML104 0/5 µm, 229 mm × 305 mm × 0.2 mm with protection foil (order code 119571)
- 1 × Prepreg Type 2125, 200 mm × 275 mm × 0.1 mm (order code 119572)

- Auxiliaries
- Convection oven
 (order code 115877)
- LPKF Cleaner
 (order code 115891)
- Spray adhesivePortable hand-held
 - microscope Brush
- Oil-free compressed air
- Tap water

The following steps are performed in this tutorial:

- Selecting a template
- Creating a new material
- Importing the data
- Setting the parameters for the new layers
- Creating the positioning holes for prepreg material and flexible material
- Modifying the workflow settings
- Computing toolpaths
- Preparing the flexible material (with ProtoLaser)
- Preparing the prepreg material (with ProtoLaser)
- Preparing the rigid material (with ProtoLaser)
- Assembling and pressing the flex-rigid stack (with MultiPress S)
- Drilling plated through holes into the flex-rigid PCB (with ProtoLaser)
- Galvanic through-hole plating the flex-rigid PCB (with Contac S4)
- Structuring the outer layers of the flex-rigid PCB (with ProtoLaser)

Selecting a template

- 1. In the user guidance step New select 4-layer-Multilayer from the drop-down list.
- 2. Select your laser system from the drop-down list (in this example PL U4).
- □ A list of templates for 4-layer multilayer materials is displayed.
- 3. Select the template *4 Layers*, *ProtoLaser U4*, *galvanic through-hole plating*, *MultiPress S*.

There is no predefined template for the production of a flex-rigid PCB. For this example, it is essential that you select a template that contains register holes for the MultiPress S, since creating a flex-rigid PCB requires bonding of rigid and flexible materials. The register holes ensure that the different physical layers can be aligned for bonding using the registration system of the MultiPress S.

- 4. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- The template has been selected.

Creating a new material

In CircuitPro PL there is no material suitable for the production of a flex-rigid PCB. Therefore, it needs to be created and saved to the material library.

- 1. In the user guidance step Material select 4 layer multilayer from the drop-down list.
- □ A list of 4-layer multi-layer materials is displayed.
- 2. Select the material LPKF 4-Layer_Galvanic.

LPKF 4-Layer_Galvanic 4 layer multilayer	
LPKF 4-Layer_Galvanic_Blind-Buried vias_IS4 4 layer multilayer	ŧ
LPKF 4-Layer_Galvanic_Blind-Buried vias_Ml 4 layer multilayer	6
LPKF 4-Layer_ProConduct 4 layer multilayer	6

Fig. 233: User guidance step Material

- 3. Click on [].
- □ A copy of the selected material is created:

LPKF 4-Layer_Galvanic 4 layer multilayer	ñ
LPKF 4-Layer_Galvanic 2 4 layer multilayer	
LPKF 4-Layer_Galvanic_Blind-Buried vias_IS4 4 layer multilayer	8
LPKF 4-Layer_Galvanic_Blind-Buried vias_MI 4 layer multilayer	ô
LPKF 4-Layer_ProConduct 4 layer multilayer	6

Fig. 234: Copy of selected material

Click on

aterial 🔇 Modify pa	ameters	
All materials 🗸		
Al2O3_Au22 Single-sided, 0.5 mm, 22 µm	Material name LPKF 4-Layer_Galvanic 2	
Focus determination Single-sided, 1,5 mm, 18 µm	+ Total thickness 1.806 mm	
FR4 Single-sided, 1,5 mm, 18 µm	ML104 Single-sided 0.2 mm Sum X	
FR4 Single-sided, 1,5 mm, 35 µm	Single-sided, 0.2 mm, 5 µm	
ML104 Single-sided, 0,2 mm, 18 µm	+ ML104 type2125 Prepreg. 0.09 mm × Prepreg 1 ×	
ML104 Single-sided, 0,2 mm, 5 µm	ML104 type2125 Prepreg 2	
ML104 type2125 Prepreg, 0,09 mm	Layer 2	
Polyimide Foil, 0,125 mm	FR4 FR4 Double-sided, 1 mm, 18/18µm ×	
ProLegend removal Coating, 0,05 mm	tayer 3 ⊻	
ProMask removal Coating, 0,05 mm	+ ML104 type2125 Prepreg. 0.09 mm × Prepreg 3 ×	
Speed test Single-sided, 1,5 mm, 18 µm		
FR4 Double-sided, 1 mm, 18/18µm	+	
FR4 Double-sided, 1,5 mm, 18/18µm	H104 Single-sided, 0.2 mm, 5 µm X Ut Bottom Layer V	
FR4 Double-sided, 1,5 mm, 35/35µm	+	
FR4 Double-sided, 1,5 mm, 5/5µm	+	
MI 104	- V	
Material	Discard Save	

□ The view *Modify parameters* is displayed:

Fig. 235: View Modify parameters

5. Enter the new material name in the input field *Material name* (in this example *LPKF Flex-rigid*).

Ψ t	Top Layer 🗸 🗸
	Ψ t

Fig. 236: Entering the material name

Delete the top five materials in the set of materials by clicking on X next to the material name.

terial name LPKF Flex-rigid				
tal thickness 1.806 mm				
ML104 Single-sided, 0.2 mm, 5 µm	×	↑ ↓	Top Layer	*
ML104 type2125 Prepreg, 0.09 mm	×		Prepreg 1	*
ML104 type2125 Prepreg, 0,09 mm	×		Prepreg 2	*
FR4 Double-sided, 1 mm, 18/18µm	×		Layer 2	~
			Layer 3	~
ML104 type2125 Prepreg, 0,09 mm	×		Prepreg 3	~
ML104 type2125 Prepreg, 0,09 mm	×		Prepreg 4	~
ML104 Single-sided, 0,2 mm, 5 μm	×	Φ Υ	Bottom Layer	~

Fig. 237: Deleting the materials

□ The view *Modify parameters* changes as follows:

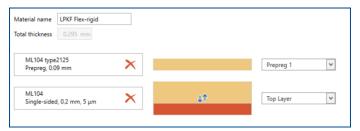


Fig. 238: Materials deleted

- 6. Select Single-sided from the drop-down list.
- □ A list of single-sided materials is displayed.
- Select the material *FR4, 1.5 mm, 18 μm* and add it to the top of the new set of materials by drag & drop or by clicking on +.

Material 🔇 Modify paramet	ters
Single-sided	
Al2O3_Au22 Single-sided, 0,5 mm, 22 µm	vial name LPKF Flex-rigid
Focus determination Single-sided, 1.5 mm, 18 µm	To these 0.295 mm
FR4 Single-sided, 1.5 mm, 18 µm	NL104 type2125 Prepreg. 0.09 mm Prepreg 1 V
FR4 Single-sided, 1.5 mm, 35 µm	
ML104 Single-sided, 0,2 mm, 18 µm	
ML104 Single-sided, 0.2 mm, 5 μm	
Speed test Single-sided, 1,5 mm, 18 µm	1
LPKF Flex-rigid 1 layer multilayer	
< Material	Discard Save

Fig. 239: Adding the material

□ The material *FR4 Single-sided, 1.5 mm, 18 µm* is added to the top of the new set of materials:

al thickness 1.813 mm			
FR4 Single-sided, 1,5 mm, 18 μm	×	ŶŢ	Top Layer 💙
ML104 type2125 Prepreg, 0,09 mm	×		Prepreg 1
ML104 Single-sided, 0.2 mm, 5 µm	×	↓ ↑	Bottom Layer 👻

Fig. 240: Material added to the new set of materials

- 8. Click on [Save].
- □ The material *LPKF Flex-rigid* has been saved.
- 9. Click on the user guidance step Material to leave the view Modify parameters.
- 10. Select All materials from the drop-down list.
- □ A list of all materials is displayed.

11. Select the material LPKF Flex-rigid.

LPKF 4-Layer_Galvanic	A	-
4 layer multilayer		
LPKF 4-Layer_Galvanic_Blind-Buried vias	6	
4 layer multilayer		
LPKF 4-Layer_Galvanic_Blind-Buried vias	A	
4 layer multilayer		
LPKF 4-Layer_ProConduct	6	
4 layer multilayer		
LPKF 6-Layer_Galvanic	0	1
6 layer multilayer	0	
LPKF 6-Layer_Galvanic_Blind-Buried vias	0	
6 layer multilayer	6	
LPKF 6-Layer_Galvanic_SingleCore	6	1
6 layer multilayer		
LPKF 8-Layer_Galvanic	6	1
8 layer multilayer		
LPKF 8-Layer_Galvanic_Blind-Buried vias	0	1
8 layer multilayer	D	
LPKF 8-Layer_Galvanic_SingleCore	0	
8 layer multilayer	6	
LPKF Flex-rigid		
2 layer multilayer		

Fig. 241: Selecting the material

- 12. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- A new material has been created.

Importing the data

- 1. In the user guidance step *Import* click on T.
- □ The dialog *Open* is displayed.
- Navigate to the folder that contains the data you want to import. For the example data that are used for this sequence, refer to the folder:
 C:\Program Files\LPKF Laser & Electronics AG\
 LPKF CircuitPro PL\Example Data\UseCase FlexRigidPCB.
- 3. Select the files you want to import (in this example *FlexRigid.BOT*, *FlexRigid.DRL*, *FlexRigid.TOP*, *PrepregCutout.GKO* and *RigidCutout.GTO*).
- 4. Click on [Open].

	Material Import Layo	out Scan fields	12015	Workflow To	olpaths		ocessin		Came
	FlexRigid.BOT	GerberX	¥	BotApe	¥	^	File F	ormat	1
	Source	Target		Size/Format			Unit	Millimeters \vee	
	✓ BoardOutline	BoardOutline	×	160.205 x 38.456 mm			Values	Absolute 🖂	
	✓ BottomLayer	BottomLayer	¥	116.992 x 25.705 mm			Decimal	Omit leading zeros	
	✓ TextBottom	BottomLayer	*	28.721 x 9.627 mm					
	FlexRigid.TOP	GerberX	~	ТорАре	~		Digits m.n	2 3	
	Source	Target		Size/Format					
	✓ BoardOutline	TopLayer	¥	160.205 x 38.456 mm					
	✓ TextTop	TopLayer	~	28.72 x 9.627 mm					
	✓ TopLayer	TopLayer	~	44.113 x 35.223 mm					
	PrepregCutout.GKO	GerberX	*	PrepregCutout.GKO	¥				
	Source	Target		Size/Format					
	✓ GerberX data	PrepregCutout.GKO	¥	85.598 x 17.97 mm					
	RigidCutout.GTO	GerberX	~	RigidCutout.GTO	~	~			
84									
45	e1	93.66	147		200.47				

□ The user guidance step *Import* changes as follows:

Fig. 242: User guidance step Import

5. For the file *FlexRigid.BOT* assign the board outline to the layer *BoardOutline* in the column *Target*.

FlexRigid.BOT	GerberX	~	BotApe 💙
Source	Target		Size/Format
✓ BoardOutline	BoardOutline	۷	160.205 x 38.456 mm
✓ BottomLayer	BottomLayer	~	116.992 x 25.705 mm
✓ TextBottom	BottomLayer	~	28.721 x 9.627 mm

Fig. 243: Assigning board outline to layer BoardOutline

6. Deactivate the check box BoardOutline in the column Source.

FlexRigid.TOP	GerberX	~	ТорАре
Source	Target		Size/Format
B ardOutline	TopLayer	~	160.205 x 38.456 mm
I extTop	TopLayer	~	28.72 x 9.627 mm
✓ TopLayer	TopLayer	~	44.113 x 35.223 mm

Fig. 244: Deselecting the source BoardOutline

7. In the column Target assign the data FlexRigid.DRL to the layer DrillPlated.

FlexRigid.DRL	Excellon	Y FlexRigid.DRL	~
Source	Target	Size/Format	
FlexRigid.DRL	DrillPlated	✓ 0.5 x 9.435 mm	

Fig. 245: Assigning the holes to layer DrillPlated

- 8. Click on [Import].
- □ The following message is displayed:

Info	^
During import, these new layers have been of PrepregCutout.GKO RigidCutout.GTO	reated:
Assign the correct PCB layer and technology	to them.
Close	
	RigidCutout.GTO Assign the correct PCB layer and technology

Fig. 246: Message Info

- 9. Click on [Close].
- □ The user guidance step *Layout* is displayed:

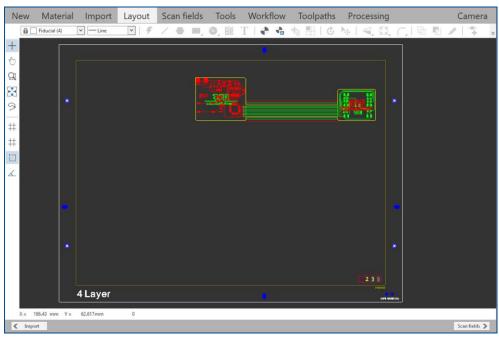


Fig. 247: User guidance step Layout

The data have been imported.

Setting the parameters for the new layers

- 1. Select the entire layout.
- □ The layout is highlighted and changes its color.

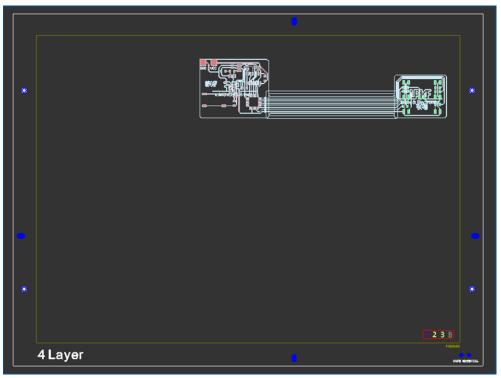


Fig. 248: Highlighted layout

- 2. Perform one of the following steps:
 - Right-click on the highlighted layout and select Move object.
 - Click on
- 3. Click on a point in the original layout to specify the reference point.
- □ A copy of the layout is displayed.
- 4. Move the copied layout with the mouse to the desired position.
- 5. When you reach the desired position, left-click to specify the target point.
- 6. In the user guidance step *Layout* click anywhere on the black background or press [Esc] to disable the *Move object* function.
- □ The layout has been moved to the desired position.

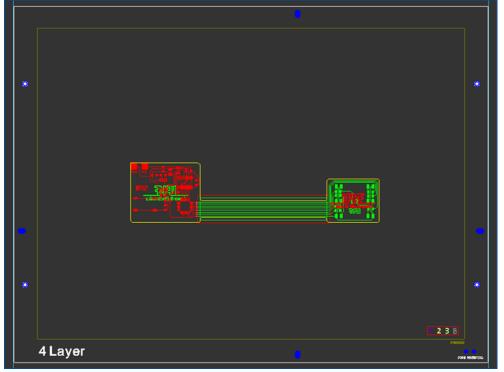


Fig. 249: Layout moved

- 7. In the pane *Workflow setup* expand the group *Layout* and expand the node *Layers*.
- □ A list of layers is displayed.

8. Right-click on the layer *PrepregCutout* and click on the context menu item *PCB layer > Prepreg 1*.

Workf	low setup			џ		
	Դո 📕	SolderMas	kTop (0) [Solder mask]	^		
	ใน 📕	SilkScreen	Bottom (0) [Silk screen]			
	Դո 🗖	SolderPast	eBottom (0) [Solder paste]			
	fa	SolderMas	kBottom (0) [Solder mask]			
	🎧 📕 RuboutIndex (1) [Mechanical]					
_	ില 📕	PrepregCu	tout(8) [Unknown]			
	Select by la	ayer	:(3) [Unknown]			
	Color		-			
	10-11-1-					
~	Visible					
\checkmark	Selectable					
	PCB layer	•	Not defined			
	Display mo	ode 🕨	Top Layer			
	Technolog	y 🕨	Prepreg 1	٦		
	Inverse		Bottom Layer			
	Rename			T		
X	Delete		-			
			-	~		

Fig. 250: Context menu PCB layer > Prepreg 1

9. Right-click on the layer *PrepregCutout* and click on the context menu item *Display mode > True Width.*

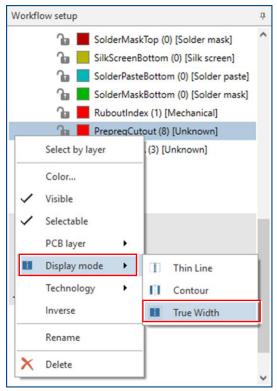


Fig. 251: Context menu Display mode > True Width

10. Right-click on the layer *PrepregCutout* and click on the context menu item *Technology* > *Cutting*.

Workflow setup				џ		
RuboutBottom (0) [Rubout]						
- B -	🎧 📒 SilkScreenTop (0) [Silk screen]					
- 🔁 🗖 I	🌆 📕 PrepregCutout (8) [Unknown]					
Select by lay	er	t(3) [l	Jnknown]			
Color						
✓ Visible						
✓ Selectable						
PCB layer	•					
🚺 Display mod	e 🕨					
Technology	•		Mechanical	l		
Inverse			Fiducials	l		
Rename			Drilling	I		
X Delete			Cutting			
▲ Processing			Wiring	l		
			Rubout	L		
			Solder mask	l		
			Solder paste			
			Silk screen			
			Topography			
			Information			
		~	Unknown	J		
				V		

Fig. 252: Context menu Technology > Cutting

- 11. Right-click on the layer *RigidCutout* and click on the context menu item *PCB layer > Top Layer*.
- 12. Right-click on the layer *RigidCutout* and click on the context menu item *Display mode > True Width*.
- 13. Right-click on the layer *RigidCutout* and click on the context menu item *Technology* > *Cutting*.
- The parameters for the new layers have been set.



The register holes in the selected template are located outside the prepreg's area. Therefore, **three prepreg positioning holes** and **three flexible material positioning holes** with a **2 mm diameter** have to be created in the layout. These positioning holes will be used to position the prepreg material and the flexible material correctly in the press mold.

- 1. In the pane *Workflow setup* expand the group *Layout* and expand the node *Layers*.
- 2. Right-click on the node Layers and click on the menu item Add new layer.

Workflow setup	џ				
Material composition					
▲ Layout					
▲ Layers					
Add new layer [5]					
Hide empty layers 0) [Mechanical]					
repare core (o) [wlechanical]					
Prepreg (2) [Information]					
👔 📃 BoardOutline (1) [Mechanical]					
Core (10) [Information]					
🚹 📕 CutInside (0) [Cutting]					
🎧 📃 DrillPlated (8) [Drilling]					
🌆 📃 DrillUnplated (0) [Drilling]					
🌆 📃 TextLayer3 (1) [Wiring]	~				

Fig. 253: Menu item Add new layer

- □ A new layer is created.
- 3. Right-click on the new layer and click on the context menu item Rename.

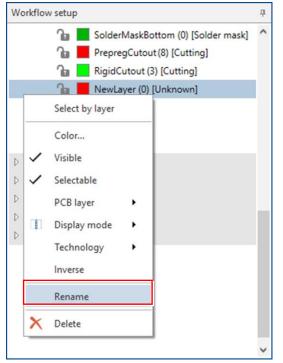


Fig. 254: Context menu item Rename

Workflow setup	ф
🚹 📕 SilkScreenBottom (0) [Silk screen]	^
🎦 📕 PrepregCutout (8) [Cutting]	
👔 🚺 RigidCutout (3) [Cutting]	
🔒 📕 FlexPositioningHoles	
Apertures	
Instances	
Fiducials	
Scan fields	
D Tools	
Workflow	
Toolpaths	
Processing	
	~

4. Enter *FlexPositioningHoles* in the input field.

- Fig. 255: Renaming layer
- 5. Press Enter.
- □ The layer is renamed.
- 6. Set the parameters of the layer *FlexPositioningHoles* by performing the following steps:
 - Right-click on the layer and click on the context menu item PCB layer > Bottom Layer.
 - Right-click on the layer and click on the context menu item Display mode > True Width.
 - Right-click on the layer and click on the context menu item Technology > Cutting.
- 7. In the user guidance step *Layout*, right-click on the black background and select *Circle by diameter* or click on .
- □ The input fields for *Specify center point* are displayed:

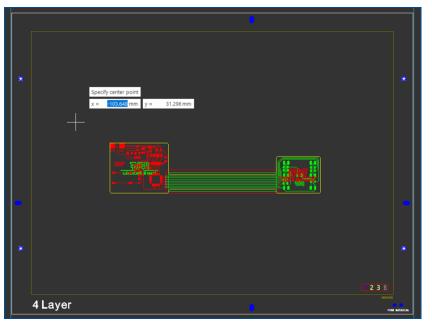


Fig. 256: Input fields Specify center point

i

The three prepreg positioning holes and three flexible material positioning holes should be located anywhere inside the prepreg area. It is best to position them approximately **30 mm off the corners** of the flex-rigid PCB data.

- 8. Click on a desired point on the black background to specify the center point of the first flexible material positioning hole.
- □ The center point is specified and the input field for *Specify diameter* is displayed:

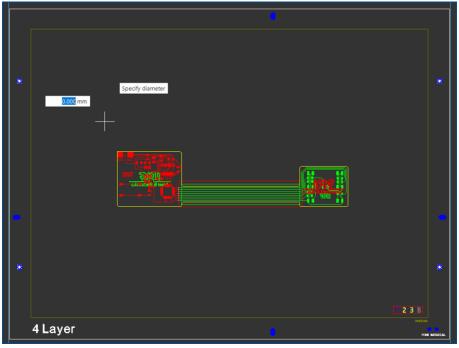


Fig. 257: Input fields Specify diameter

- 9. Enter the diameter in the input field (in this example 2 mm) and press [Enter].
- □ The first flexible material positioning hole is created in the layout. The input fields for specifying the center point of the next flexible material positioning hole are displayed:

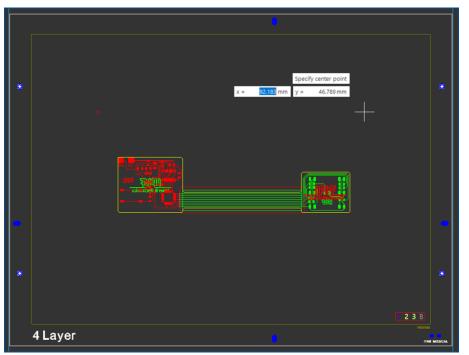


Fig. 258: Specifying the next center point

- Repeat the steps 8 and 9 for creating another two flexible material positioning holes.
- 10. Press Esc to exit the function *Circle by diameter*.
- □ The three flexible material positioning holes have been created on the layer *FlexPositioningHoles*:

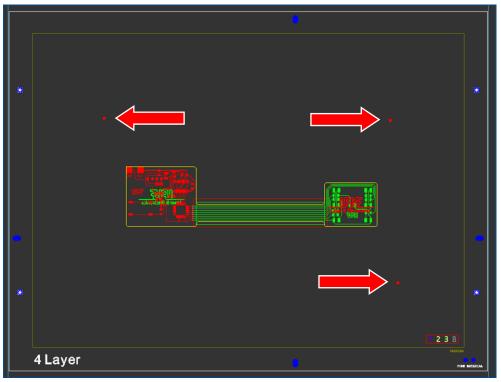


Fig. 259: Flexible material positioning holes

- 11. In the pane *Workflow setup*, right-click on the node *Layers* and click on the menu item *Add new layer*.
- □ A new layer is created.
- 12. Right-click on the new layer and click on the context menu item *Rename*.
- 13. Enter PrepregPositioningHoles in the input field.
- 14. Press Enter.
- □ The layer is renamed.
- 15. Set the parameters of the layer *PrepregPositioningHoles* by performing the following steps:
 - Right-click on the layer and click on the context menu item PCB layer > Prepreg 1.
 - Right-click on the layer and click on the context menu item Display mode > True Width.
 - Right-click on the layer and click on the context menu item Technology > Cutting.
- 16. Right-click on the layer *FlexPositioningHoles* and click on the context menu item *Select by layer*.
- □ All the objects on this layer are selected and highlighted.

17. Right-click on the black background and click on the context menu items *Copy to layer > PrepregPositioningHoles.*

Ж	Cut Ctrl+X	BoardOutline
	Copy Ctrl+C	CutInside
	Invert selection	DrillPlated
D.	Group	DrillUnplated
	Group	TopLayer
₩	Move object Ctrl+M	Layer2
Ċ	Rotate	Layer3
	horizontally	BottomLayer
	vertically	TextTop
К Я Ц У	Scale	TextLayer2
	Step and repeat	TextLayer3
88	Create instance type	TextBottom
**	Expand	RuboutTop
5	Merge	RuboutLayer2
0	Split	RuboutLayer3
		RuboutBottom
4	Convert to closed path	SilkScreenTop
	Convert to flash	SolderPasteTop
	Combine to flash	SolderMaskTop
4	Create fiducial with defaults	SilkScreenBottom
	Assign to layer	SolderPasteBottom
	Copy to layer	SolderMaskBottom
_	Create aperture	RuboutIndex
	Select by geometry	PrepregCutout
·		RigidCutout
huul	Measure	FlexPositioningHoles
×	Delete	PrepregPositioningHoles

Fig. 260: Context menu Copy to layer

□ All the objects on the layer *FlexPositioningHoles* are copied to the layer *PrepregPositioningHoles*.



The prepreg positioning holes need to be larger than the flexible material positioning holes. This prevents the prepreg material from spreading onto the flexible material during the bonding process.

- 18. Right-click on the layer *PrepregPositioningHoles* and click on the context menu item *Select by layer*.
- □ All the objects on this layer are selected and highlighted.
- 19. Right-click on the black background and click on the context menu item *Scale* or click on .

□ The input fields for *Non-uniform scaling (in percent)* and a drop-down list are displayed:

	Non-uniform scaling (in percent) Scale factor x = 100.000 % Scale factor y = 100.000 % Scale factor x = 100.000 % Scale factor y = 100.000 %	
•		
	• [2 3 B	*
	4 Layer	LANDAL

Fig. 261: Input fields Non-uniform scaling (in percent)

20. Enter 150 % for Scale factor x and press the Tab key.

21. Enter 150 % for Scale factor y and press the Tab key.

- □ The drop-down list is activated.
- 22. Press the $[\downarrow]$ key twice.
- □ The option *Individually* is selected in the drop-down list.

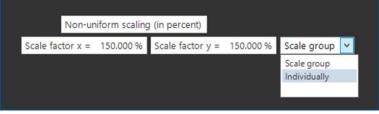
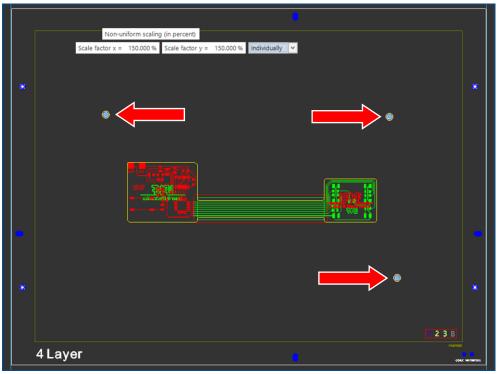


Fig. 262: Values for Non-uniform scaling (in percent)

23. Press Enter to confirm the selection.



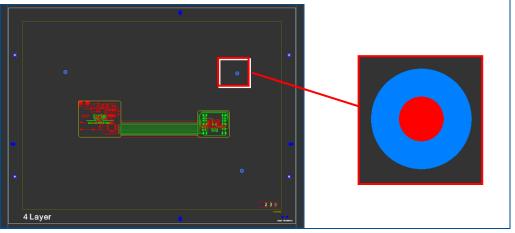
□ The scaled prepreg positioning holes is highlighted in orange:

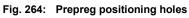
Fig. 263: Preview of the scaled positioning holes

24. Press Enter to exit the function Scale.

25. Press Esc to deselect the objects.

□ The prepreg positioning holes are increased in size.





26. Save your project.

The positioning holes for prepreg material and flexible material have been created.

Modifying the workflow settings

- 1. Switch to the user guidance step Workflow.
- 2. In the pane *Workflow setup* expand the group *Workflow* and right-click on the phase *Prepare Laminate Bottom*.
- 3. Click on the context menu item Rename.

Workf	flow se	etup	д
⊳ M	ater	ial composition	
⊳ La	you	t	
D Sc	an f	ields	
⊳ To	ols		
⊿ W	orkf	low	
₽	1. Pro	epare Laminate Bottom	
₽	Ж	Cut	
	e.	Сору	
Þ		Compute toolpaths	
Þ		Process	
▷ To	►	Process all	
▷ Pr		Rename	
	×	Delete	
	_		-

Fig. 265: Rename

- 4. Enter the name *Prepare Flex* in the input field.
- 5. Press Enter.
- □ The phase is renamed and selected.
- 6. In the user guidance step Workflow select the work package Cut Reference Holes.

	New Materia	al Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
Mount Laminate Material Placement Cut Reference Holes Dismount Material None Cut Reference Holes Type Cut Material None Cut Reference Holes Type Cut Material Cut Material Persone Laminate Top Persone Lamina	Prepare Flex		9 1						
Mount Laminate Material Placement Cut Reference Holes Dismount Material None Cut Reference Holes Type Cut Material None Cut Reference Holes Type Cut Material Cut Material Persone Laminate Top Persone Lamina									
Mount Laminate Material Placement Cut Reference Holes Dismount Material None Cut Reference Holes Type Cut Material None Cut Reference Holes Type Cut Material Cut Material Persone Laminate Top Persone Lamina									
Mumit Laminate Material Ournount Material Prepare Laminate Top Work package options Activated Test tool settings Name Cut Material Tool Cut Material Prepare Caeminate Tool Cut Material Input layer Prepare Laminate Tool Cut Material Prepare Laminate Tool Cut Material Prepare Laminate Tool Cut Material Prepare Laminate Tool Cut Material Prepare Laminate Tool Cut Material Prepare Laminate									
Activated Instructions Tool Cutting Input layer Prepare Correin BourdOutline Cutinide San field set Prepare Fies scan field set Prepare Fies scan field set Parameters	Mount Laminate	→ Material Place	ement	Cut Reference Holes	→ Dist	mount Material	Prepare Laminate T	op	
Activated Instructions Tool Cutting Input layer Prepare Correin BourdOutline Cutinide San field set Prepare Fies scan field set Prepare Fies scan field set Parameters									
Activated Instructions Tool Cutting Input layer Prepare Correin BourdOutline Cutinide San field set Prepare Fies scan field set Prepare Fies scan field set Parameters									
Activated Instructions Tool Cutting Input layer Prepare Correin BourdOutline Cutinide San field set Prepare Fies scan field set Prepare Fies scan field set Parameters	Work package options -								
Name Cut Reference Holes Type Cutting Tool Cut Material Input layer Prepare Core BoordOutline Cuthinded BoordOutline Cuthinded Scan field set Prepare Flex scan field set Prepare Prepare Flex scan field set Control Contrectica									
Type Cutting V Tool Cut Material V Input layer Prepare Core Core Core Cut Prepare Core Core Cut Cutting Cutt	Test tool settings		_						
Tool Cut Material V Input tayer Prepare Core Core Cut Cut Material V Prepare Laminate Control Cut Material V Scan field set Prepare Fiex scan field set V Prepare Fiex scan field set V Parameters V	Name Cut Refe	ence Holes							
Input layer	Type Cutting	~	1						
Scan field set V Prepare Limited Cutinide Cutinide Cutinide Prepare Fier scan field set V Prepare Fier scan field set	Tool Cut I	Aaterial	~						
BeardOutine Cuthide Scan field set Prepare Fiex scan field set Prepare fiex scan field set Parameters	Input layer	Prepare Core							
Scan field set	v								
Scan field set Prepare Flex scan field set Parameters									
Parameters •									
Parameters	Scan field set Prepare	Flex scan field set	·						
				2 🕄					
	Parameters								
	f Tab					Complete to the st			Toolpaths 3

Fig. 266: Work package selected

Work package of	options
✓ Activated	
Test tool se	ttings
Name	Cut Reference Holes
Туре	Cutting
Tool	Cut Material
Input layer	PrepregCutout RigidCutout FlexPositioning PrepregPositioni Y
Scan field set	Prepare Flex scan field set V
Parameters —	·

7. Under Input layer activate the check box FlexPositioningHoles.

Fig. 267: Modified settings

- 8. Click on [Apply].
- □ The workflow settings for the phase *Prepare Flex* are modified.
- 9. In the pane *Workflow setup* right-click on the phase *Prepare Laminate Top* and click on the context menu item *Rename*.
- 10. Enter the name *Prepare Prepreg* in the input field.
- 11. Press Enter.
- □ The phase is renamed and selected.
- 12. In the user guidance step Workflow select the work package Cut Reference Holes.

New N	1aterial	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
Prepare Prepr	reg 🗸	q q []	ej 🖻 🕨						
Mount Lamin	ate $ ightarrow$	Material Place	ment	Cut Reference Holes	→ Disr	nount Material	[Core] Structure Lay	er 3	
Work package									
Name	Cut Laminate	Pinholes							
Туре	Cutting	~							
Tool	Cut Materi	ial	×						
Input layer	Prep Prep Boar Cuti	pare Core pare Laminate rdOutline Inside							
Scan field set	Prepare Lamin	nate Top scan f 🔽							
				21 🕄					
Parameters —									
Tools				Discard	Apply	Compute toolpaths			Toolpaths 义

Fig. 268: Work package selected

- 13. Under Input layer perform the following steps:
 - Deactivate the check box *PrepareLaminate*.
 - Activate the check box *PrepregCutout*.
 - Activate the check box *PrepregPositioningHoles*.
- □ The settings of the work package *Cut Reference Holes* are displayed as follows:

Work package of	options			
Activated				
Test tool se	ttings			
Name	Cut Reference Holes			
Туре	Cutting	~		
Tool	Cut Material		~	
Input layer	Prepare Lamin ate PrepregCutou PrepregPositioni FlexPositioning DisidCutout	^		
Scan field set	Prepare Laminate Top scan f	~		
Parameters —				

Fig. 269: Settings of the work package Cut Reference Holes

- 14. Expand the group Parameters.
- 15. Enter 20 μ m in the input field *Channel width*.

Position	Inside	~	
Channel width	20 µm		
Target beam overlap	10 %		
Actual beam overlap	67 %		
Sorting	Outside in	V Unidirectional	
Tab positions	NoGaps	~	
Tab width	0 mm		
Distance	0 mm		

Fig. 270: Channel width modified

- 16. Click on [Apply].
- □ The workflow settings for the phase *Prepare Prepreg* are modified.
- 17. In the pane *Workflow setup* right-click on the phase [Core] StructureLayer 3 and click on the context menu item *Rename*.
- 18. Enter the name *Prepare Rigid* in the input field.
- 19. Press Enter.
- □ The phase is renamed and selected.

20. In the user guidance step *Workflow* deactivate the check box of the work package *Structure Layer 3.*

New Material	Import Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
Prepare Rigid 🗸	⊑ [9] ≞ ▶						
v	v		V			A	
Mount Core		Drill Fiducials	\rightarrow	Reference Holes			
Mount Core	Material Placement	Unil Fiducials	Cut	Reference Holes	Structure Layer :	[Core] Structure Layer 2	
		_					
Work package options							
Activated							
✓ Test tool settings							
Name Structure La	yer 3						
Type Structuring	~						
Scan field set [Core] Struc	ture Layer 3 scar 🔽						
Sorting policy ProtoLaser	tandard 🗸						
EE	EE						
Parameters							
Tools		Discard	Apply	Compute toolpaths			Toolpaths 义

Fig. 271: Work package deselected

- □ The work package is deselected.
- 21. Select the work package *Cut Reference Holes*.

New	Material	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
Prepare	Rigid 🗸		ē 🖬 🕨						
					v			A	
Mour	nt Core \rightarrow	Material Place	$_{ment}$ \rightarrow	Drill Fiducials	-) Cut	Reference Holes	Structure Layer 3	→ [Core] Structure Layer 2	
Work pac	kage options				-				
Activ									
	tool settings Cut Reference								
Name									
Туре	Cutting	~							
Tool	Cut Mater	ial	*						
Input la		oare Core ^							
		oare Laminate							
		Inside							
		eutladeu Y							
Scan fie	ld set [Core] Structu	re Layer 3 scar 💙							
				2 🕄					
Paramet	ers —								
Tools				Discard	Apply	Compute toolpaths			Toolpaths 义

Fig. 272: Work package selected

i

Work package o	potions	
Activated		
Test tool set	ttings	
Name	Cut Reference Holes]
Туре	Cutting 🗸]
Tool	Cut Material	~
Input layer	PrepreaCutout ^ RigidCutout _ HexPositioning _ PrepregPositioni _	
Scan field set	[Core] Structure Layer 3 scar 🗸	
		2 2 3
Parameters —		🐑

22. Under Input layer activate the check box RigidCutout.

Fig. 273: Modified settings

- 23. Click on [Apply].
- □ The workflow settings for the phase *Prepare Rigid* are modified.
- 24. In the pane *Workflow setup* select the phase *[Core] StructureLayer* 2 and press Delete or click on X.
- □ The phase is deleted.

When you use a newly created material, it is recommended that you test the tool settings before processing.

25. In the user guidance step *Workflow* select the phase [*Top-Bot*] *Galvanic THP* from the drop-down list and select the work package *Drill Plated Through-Holes*.

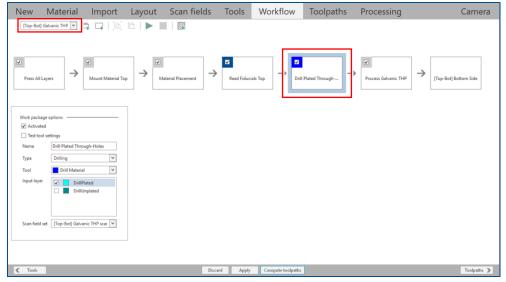


Fig. 274: Phase and work package selected

26. Activate the check box Test tool settings.

Name	Drill Plated Through-Holes	
Туре	Drilling	~
Tool	Drill Material	~
Input layer	✓ DrillPlated	
	DrillUnplated	

Fig. 275: Modified settings

- 27. Click on [Apply].
- □ The workflow settings for the phase [Top-Bot] Galvanic THP are modified.
- The workflow settings have been modified.

Computing toolpaths

- 1. Switch to the user guidance step *Toolpaths*.
- 2. Click on [Compute toolpaths] or on **[**].
- □ The dialog *Computing toolpaths* is displayed.
- 3. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 4. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 5. Click on [Close].
- 6. Save your project.
- The toolpaths have been computed.

Preparing the flexible material (with ProtoLaser)

1. Switch to the user guidance step *Processing*.

New	Material	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
* * * * *	-	 4 La 		9 № 2 初					Step size 300 mm Step size 300 mm x 155.000 mm y 310.000 mm z 0.000 mm
		J.	0 M L10	102 mm 5 µm	0		, 0	-	
< Toolpat	ths				Start pro	duction			Camera 🔉

Fig. 276: User guidance step Processing

- 2. Click on [Start production] or on ▶.
- □ When the message *Processing phase: Mount laminate* is displayed, place the flexible material with the **copper side facing upwards** in the center of the processing table.
- 3. Click on 👯.
- □ The flexible material is fastened to the processing table by vacuum.
- 4. Click on [OK].
- □ The dialog *Placement* is displayed.
- 5. Move the dialog *Placement* to get a better overview.
- 6. Determine the processing area. The base material position and base material size **must match** the processing area used by the CircuitPro PL software.



For detailed information on project placement by determining the processing area, refer to chapter 1.1.

- 7. Click on the tab Processing data.
- 8. Click on [Center on material].
- 9. Click on [OK].
- □ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.



For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the laser system drills the fiducials, the positioning holes, the register holes, and the positioning holes of the flexible material.

- 10. When the message *Processing phase: Dismount material* is displayed, remove the flexible material from the system.
- The flexible material has been prepared.

Preparing the prepreg material (with ProtoLaser)

- 1. In the message Processing phase: Dismount material click on [OK].
- □ When the message *Processing phase: Mount laminate* is displayed, place the prepreg material in the center of the processing table.
- 2. Click on 👑.
- □ The prepreg material is fastened to the processing table by vacuum.
- 3. Click on [OK].
- □ The dialog *Placement* is displayed.
- 4. Move the dialog *Placement* to get a better overview.
- 5. Determine the processing area. The base material position and base material size **must match** the processing area used by the CircuitPro PL software.

For detailed information on project placement by determining the processing area, refer to chapter 1.1.

- 6. Click on the tab *Processing data*.
- 7. Click on [Center on material].
- 8. Click on [OK].
- □ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.
- For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the laser system drills the prepreg positioning holes and the cutout is being created in the prepreg material.

- 9. When the message *Processing phase: Dismount material* is displayed, remove the prepreg material from the system.
- The prepreg material has been prepared.

Preparing the rigid material (with ProtoLaser)

- 1. In the message Processing phase: Dismount material click on [OK].
- □ When the message *Processing phase: Mount core* is displayed, place the rigid material in the center of the processing table.
- 2. Click on 🐫.
- □ The rigid material is fastened to the processing table by vacuum.
- 3. Click on [OK].
- □ The dialog *Placement* is displayed.
- 4. Move the dialog *Placement* to get a better overview.
- 5. Determine the processing area. The base material position and base material size **must match** the processing area used by the CircuitPro PL software.



For detailed information on project placement by determining the processing area, refer to chapter 1.1.

- 6. Click on the tab Processing data.
- 7. Click on [Center on material].
- 8. Click on [OK].

□ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.



For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the laser system drills the fiducials, the positioning holes and the register holes into the rigid material. The cutout is being created in the rigid material.



Save the residual material that has been cut out of the rigid material. You will need it later for assembling and pressing of the flex-rigid stack.

- 9. When the message *Processing phase: Press all layers* is displayed, remove the rigid material from the system and click on [Cancel] to temporarily stop the process.
- 10. Spray the rigid material with LPKF Cleaner and use a brush to clean it.
- 11. Rinse the rigid material with tap water and dry it with compressed air.
- The rigid material has been prepared.

Assembling and pressing the flex-rigid stack (with MultiPress S)



For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.6.

- 1. Start the pre-heating process of the MultiPress S.
- 2. Prepare the materials.



Dry the materials in a convection oven at 100 °C (212 °F) for 30 minutes before assembling them in the press mold. **Do not dry the prepreg material**.

- 3. Assemble the flex-rigid stack in the press mold using the following order:
 - Flexible material (ML104)

Assemble the flexible material according to the pins in the press mold. Make sure that the **copper side** faces **downwards**.

Prepreg material

Align the prepreg material to the flexible material. Match the position of the three prepreg positioning holes to the flexible material positioning holes. Apply some spray adhesive on the corners of the prepreg material to ensure that its position will remain correctly during assembling.

Rigid material (FR4)

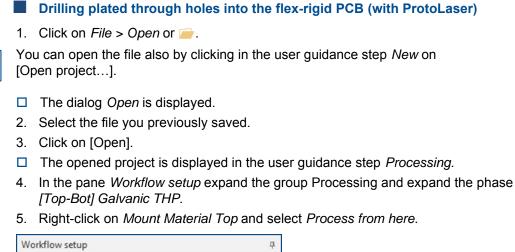
Assemble the rigid material according to the pins in the press mold.



Insert a strip from the base material in a matching size into the cutout of the rigid material. For the strip you can use the residual material that has been created during the cutout in the sequence *Preparing the rigid material* (see page 217). This strip supports the flexible material during pressing. This way, better pressing results can be achieved.

Apply self-adhesive sealing rings on positioning holes, register holes and fiducials. This prevents the prepreg materials from entering the holes.

- 4. Press the flex-rigid stack.
- The flex-rigid stack has been assembled and pressed.



Workflow setup	д
Material composition	
Layout	
Scan fields	
> Tools	
Vorkflow	
Toolpaths	
Processing	
1. Prepare Flex	
2. Prepare Prepreg	
3. Prepare Rigid	
 4. [Top-Bot] Galvanic THP 	
1. Press All Layers	
2. Mount Material Top	
<process selected=""></process>	
Process from here	
6. Process Galvanic THP	
5. [Top-Bot] Bottom Side	
6. [Top-Bot] Top Side	

Fig. 277: Context menu Process from here

6. When the message *Processing phase: Mount material top* is displayed, place the flex-rigid PCB with the **Top side** (*TopLayer*) **facing upwards** onto the processing table.

Ensure that the positioning holes are located in the right front corner.

- 7. Click on 🐫.
- □ The flex-rigid PCB is fastened to the processing table by vacuum.
- 8. Click on [OK].
- □ The dialog *Placement* is displayed.

- 9. Move the dialog *Placement* to get a better overview.
- 10. Click on the tab Processing data.
- 11. Place the processing data. The location of the layout **must match** the location of the flex-rigid PCB and fiducials on the processing table.



For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 12. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Top side (*TopLayer*).



For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the following dialog is displayed:

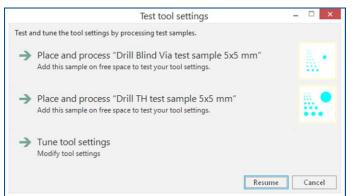


Fig. 278: Dialog Test tool settings



It is **recommended** that you process the **through-hole test sample** on your flex-rigid PCB before you continue with production.

13. Select Place and process "Drill TH test sample 5x5 mm".



For detailed information on placing and processing the test sample, refer to chapter 1.5.

- 14. Check the processed test sample with a portable hand-held microscope.
- 15. Determine if the processing quality is satisfactory.
- 16. If the processing quality is not satisfactory, tune the tool settings.



- 17. If the processing quality is satisfactory, click on [Resume].
- □ The laser system drills the plated through holes.
- 18. When the message *Processing phase: Galvanic through-hole plating* is displayed, click on [Cancel] to temporarily stop the process.
- 19. Remove the flex-rigid PCB from the system.
- The plated through holes have been drilled into the flex-rigid PCB.

Galvanic through-hole plating the flex-rigid PCB (with Contac S4)

- 1. Switch on the system.
- 2. Select a profile.



The recommended total copper thickness after through-hole plating is 30 μm to 35 $\mu m.$

- 3. Start the process.
- 4. Prepare the flex-rigid PCB for through-hole plating.
- 5. Clean the flex-rigid PCB.
- 6. Condition the flex-rigid PCB.
- 7. Activate the flex-rigid PCB.
- 8. Clean the holes with LPKF ViaCleaner.
- 9. Copper-plate the flex-rigid PCB.
- 10. Switch off the system.
- The flex-rigid PCB has been galvanically through-hole plated.



For detailed information on galvanic through-hole plating, refer to the user manual of **Contac S4**.

Structuring the outer layers of the flex-rigid PCB (with ProtoLaser)

- 1. Open the **cp2d* file you previously saved.
- □ The opened project is displayed in the user guidance step *Processing*.
- 2. In the pane *Workflow setup* expand the group *Processing*.
- 3. Right-click on [Top-Bot] Bottom Side.
- 4. Select Process from here in the context menu.
- 5. When the message *Processing phase: Mount material bottom* is displayed, place the flex-rigid PCB with the **flexible side** (*BottomLayer*) **facing upwards** onto the processing table.

Ensure that the positioning holes are located in the right rear corner.

- 6. Click on 👑.
- □ The flex-rigid PCB is fastened to the processing table by vacuum.
- 7. Click on [OK].
- □ The following dialog is displayed:

Copper t	hickness	-	
Enter the copper thickness:	0 µm		
	OV		Cancel

Fig. 279: Dialog Copper thickness

8. Enter the total copper thickness after galvanic through-hole plating into the input field.

You can determine the total copper thickness by calculating the thickness from the copper deposition rate in Contac S4 (approx. $0.15 \mu m/min$).

- 9. Click on [OK].
- □ The dialog *Placement* is displayed.
- 10. Move the dialog *Placement* to get a better overview.
- 11. Click on the tab Processing data.
- Place the processing data. The location of the layout **must match** the location of the flex-rigid PCB and fiducials on the processing table.

For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 13. After project placement is complete click on [OK].
- □ The laser system reads the fiducials on the Bottom side (*BottomLayer*).

For detailed information on fiducial recognition, refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the following dialog is displayed:

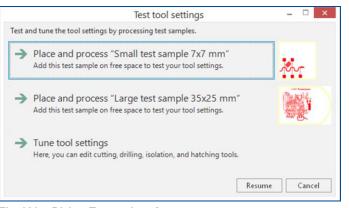


Fig. 280: Dialog Test tool settings



It is **recommended** that you process a **test sample** on your flex-rigid PCB before you continue with production.

14. Select Place and process "Large test sample 35x25 mm".

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 15. Check the processed test sample with a portable hand-held microscope.
- 16. Determine if the processing quality is satisfactory.

17. If the processing quality is not satisfactory, tune the tool settings.

For detailed information on tuning the tool settings, refer to chapter 1.5.

18. If the processing quality is satisfactory, click on [Resume].

- Structuring the flexible part of the flex-rigid PCB (*BottomLayer*) is started.
- 19. When the message *Processing phase: Flip material* is displayed, turn the flex-rigid PCB over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 20. Place the processing data **matching** the location of the flex-rigid PCB and fiducials on the processing table.
- 21. When project placement is complete, click on [OK].
- The laser system reads the fiducials on the Top side (*TopLayer*).
 After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed.
 When finished, the dialog *Test tool settings* is displayed.
- 22. Repeat the steps 14 to 17.
- 23. If the processing quality is acceptable, click on [Resume].
- □ The laser system structures the Top side (*TopLayer*) and cuts out the flex-rigid PCB.
- 24. When the message *Board production finished* is displayed, remove the flex-rigid PCB from the system.
- 25. Spray the flex-rigid PCB with LPKF Cleaner and use a brush to clean it.
- 26. Rinse the flex-rigid PCB with tap water and dry it with compressed air.
- The outer layers of the flex-rigid PCB have been structured.

The flex-rigid PCB production is finished.

4.3 Structuring the solder mask with the laser system

This chapter describes how to remove solder resist selectively from solder pads on a PCB with the laser system. The **benefit** of this procedure is avoiding printing the artwork and avoiding handling of chemicals.



The solder resist is removed with the hatching method.

For detailed information on processing methods of the ProtoLaser refer to chapter 1.7.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

Auxiliaries

System

- Fully structured and through-plated PCB
- LPKF Cleaner (order code: 115891)
 Convection oven
- ProtoLaser U4/S4/R4
- ProMask
- Convection oven
 (order code: 115877)
-
- Brush
- Oil-free compressed air
- Tap water

The following steps are performed in this tutorial:

- Applying the solder resist on the PCB (with ProMask)
- Preparing the data for solder mask structuring in CircuitPro PL
- Structuring the solder mask (with ProtoLaser)

When any PCB production is finished, a green solder resist can be applied on its surface. A solder mask eliminates the risk of short circuits by soldering of SMDs or conventional components on the PCB.



Work with PCBs that have **not yet been broken** at the breakout tabs! Thus, perform the **contour routing** only **after** the completion of this application example.

If contour routing has already been done, make sure you **do not break or cut the breakout tabs**. This way the PCB remains attached to the base material.





Spray the already structured PCB with LPKF Cleaner and use a brush to clean the PCB, rinse with tap water and dry it with compressed air.

- 1. Mix Component A and Component B.
- 2. Coat the PCB.



Ensure that the fiducials are not coated!

- 3. Predry the PCB in a convection oven.
- 4. Expose the PCB without the artwork to the UV light.
- 5. Postcure the PCB in a convection oven.
- The solder resist has been applied on the PCB.



For detailed information on applying the solder mask on the PCB refer to the process description of ProMask/ProLegend.

Preparing the data for solder mask structuring in CircuitPro PL



For detailed information on preparing the data in CircuitPro PL refer to chapter 2.1.

1. Open the file you have been using for the production of your PCB.

Your file needs to include solder mask layers.

Should your file not include these layers, you can import them by clicking in the user guidance step *Import* on T. Make sure you assign them to the layers SolderMaskBottom and SolderMaskTop.

- 2. Switch to the user guidance step Layout.
- □ The layout changes as follows:

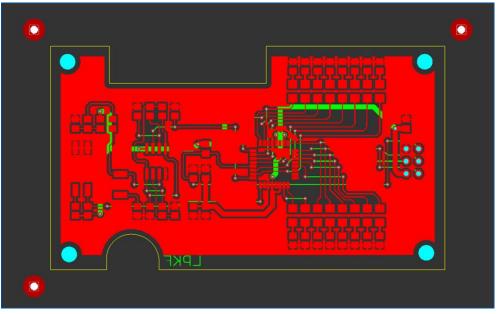


Fig. 281: User guidance step Layout

- 3. Select the entire layout.
- □ The layout is highlighted and changes its color.
- 4. Click on *Edit* > *Copy* or press Ctrl + C.
- □ The layout has been copied to the clipboard.
- 5. Switch to the user guidance step *New* and select *Double-sided* from the drop-down list.
- 6. Select your laser system from the drop-down list (in this example PL U4).
- 7. Select the template ProMask_Removal, ProtoLaser U4.
- 8. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 9. Select the material ProMask removal, 2 layer multilayer.

RT6002 Double-sided, 0.51 mm, 18/18µm RT6010 Double-sided, 0.25 mm, 18/18µm Double-sided, 0.25 mm, 35/35µm Image: Constraint of the state of the st	ктооо2 Double-sided, 0.25 mm, 18/18µm	â	^
Double-sided, 0.25 mm, 18/18µm RT6010 Double-sided, 0.25 mm, 35/35µm RT6010 Double-sided, 0.64 mm, 18/18µm RT6010 Double-sided, 0.64 mm, 35/35µm RT6010 Double-sided, 0.64 mm, 35/35µm TACONIC RF10-0100 Double-sided, 0.27 mm, 18/18µm TMM101 Double-sided, 0.76 mm, 35/35µm XT_Duroid8000 Double-sided, 0.05 mm, 18/18µm XT_Duroid8100 Double-sided, 0.1 mm, 18/18µm		â	
Double-sided, 0.25 mm, 35/35µm RT6010 Double-sided, 0.64 mm, 18/18µm RT6010 Double-sided, 0.64 mm, 35/35µm RT6010 Double-sided, 0.64 mm, 35/35µm RTACONIC RF10-0100 Double-sided, 0.27 mm, 18/18µm RTM101 Double-sided, 0.76 mm, 35/35µm RT_Duroid8000 Double-sided, 0.05 mm, 18/18µm RT_Duroid8100 Double-sided, 0.1 mm, 18/18µm RT_Duroid8100		â	
Double-sided, 0.64 mm, 18/18µm RT6010 Double-sided, 0.64 mm, 35/35µm TACONIC RF10-0100 Double-sided, 0.27 mm, 18/18µm TMM101 Double-sided, 0.76 mm, 35/35µm XT_Duroid8000 Double-sided, 0.05 mm, 18/18µm XT_Duroid8100 Double-sided, 0.1 mm, 18/18µm		â	
Double-sided, 0.64 mm, 35/35µm TACONIC RF10-0100 Double-sided, 0.27 mm, 18/18µm TMM101 Double-sided, 0.76 mm, 35/35µm XT_Duroid8000 Double-sided, 0.05 mm, 18/18µm XT_Duroid8100 Double-sided, 0.1 mm, 18/18µm		â	
Double-sided, 0.27 mm, 18/18µm TMM10I Double-sided, 0.76 mm, 35/35µm XT_Duroid8000 Double-sided, 0.05 mm, 18/18µm XT_Duroid8100 Double-sided, 0.1 mm, 18/18µm		A	
Double-sided, 0.76 mm, 35/35µm XT_Duroid8000 Double-sided, 0.05 mm, 18/18µm XT_Duroid8100 Double-sided, 0.1 mm, 18/18µm		â	
Double-sided, 0.05 mm, 18/18µm XT_Duroid8100 Double-sided, 0.1 mm, 18/18µm		0	
Double-sided, 0.1 mm, 18/18µm	-	â	
ProMask removal	-	ô	
2 layer multilayer	ProMask removal 2 layer multilayer	6	v

Fig. 282: Material selection

- 10. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 11. Switch to the user guidance step Layout.
- 12. Click on *Edit* > *Paste* or press [Ctrl] + [V].

□ A copy of the layout (in orange) and the input fields for *Specify target point* are displayed:

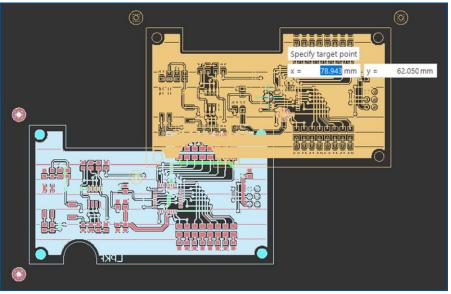


Fig. 283: Specify target point

- 13. Move the copied layout with the mouse to the desired position.
- 14. Left-click to specify the target point.
- 15. Click anywhere on the black background or press Esc to deselect the layout.
- □ The layout has been pasted and is displayed in the user guidance step *Layout*.

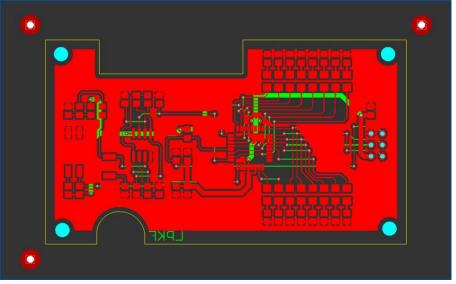


Fig. 284: Pasted layout

- 16. Switch to the user guidance step *Toolpaths*.
- 17. Click on [Compute toolpaths] or on **[**].
- □ The dialog *Computing toolpaths* is displayed.
- 18. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 19. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 20. Click on [Close].
- The data have been prepared for solder mask structuring in CircuitPro PL.

i

Structuring the solder mask (with ProtoLaser)

- 1. Switch to the user guidance step *Processing*.
- 2. Click on b or click on [Start production].
- 3. When the message *Processing phase: Mount material bottom* is displayed, place the coated PCB with the **Bottom side** (*SolderMaskBottom*) **facing upwards** onto the processing table.
- 4. Click on 🐫.
- □ The coated PCB is fastened onto the processing table by vacuum.
- 5. Click on [OK].
- □ The dialog *Placement* is displayed.
- 6. Move the dialog *Placement* to get a better overview.
- 7. Click on the tab Processing data.
- 8. Place the processing data.

The location of the layout **must match** the location of coated PCB and fiducials on the processing table.

For detailed information on project placement according to fiducial positions, refer to chapter 1.1.

- 9. After project placement is complete, click on [OK].
- □ The laser system reads the fiducials on the Bottom side of the coated PCB (*SolderMaskBottom*).

For detailed information on fiducial recognition refer to chapter 1.2.

After fiducial recognition, the laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.

For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, removing the solder resist from the solder pads on the Bottom side of the coated PCB (*SolderMaskBottom*) is started.

- 10. When the message *Processing phase: Flip material* is displayed, turn the coated PCB over around the symmetry axis of the system and click on [OK]. Ensure that the positioning holes are located in the right front corner.
- □ The dialog *Placement* is displayed.
- 11. Place the processing data **matching** the location of the coated PCB and fiducials on the processing table.
- 12. When project placement is complete, click on [OK].

□ The laser system reads the fiducials on the Top side of the coated PCB (*SolderMaskTop*).

After fiducial recognition, the laser system starts the material thickness measurement and the message *Thickness measurement running* is displayed. When finished, removing the solder resist from the solder pads the Top side of the coated PCB (*SolderMaskTop*) is started.

- 13. When the message *Board production finished* is displayed, remove the PCB from the system.
- 14. Spray the PCB with LPKF Cleaner and use a brush to clean it.
- 15. Rinse the PCB with tap water and dry it with compressed air.
- The solder mask has been structured.

The solder resist has been removed from the solder pads of a PCB with the laser system.

4.4 Producing an RF PCB

This chapter describes how to create an RF PCB using a laser system.

The RF PCB is produced with the hatching method.

For detailed information on processing methods of the ProtoLaser refer to chapter 1.7.

Ensure that the following consumables and auxiliaries are available before performing the described tasks:

Consumables

Auxiliaries

- System
- ProtoLaser U4/S4/R4
 (arder adde: 115201)
- oxide Al₂O₃, 0.5 mm, 0/23 µm gold-plated

Base material Aluminum

- (order code: 115891) • Brush
- Oil-free compressed air
- Tap water

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PL
- Processing the PCB (with ProtoLaser)

Preparing the data in CircuitPro PL

For detailed information on preparing the data in CircuitPro PL refer to chapter 2.1.

- 1. In the user guidance step New select Single-sided from the drop-down list.
- 2. Select your laser system from the drop-down list (in this example PL R4).
- □ A list of templates for single-sided materials is displayed.
- 3. Select the template Single-sided, Top, ProtoLaser R4.
- 4. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 5. Select the material Al2O3_Au22, Single-sided, 0.5 mm, 22 µm.

Al2O3_Au22 Single-sided, 0.5 mm, 22 μm	
Focus determination	6
Single-sided, 1.5 mm, 18 µm	
FR4	6
Single-sided, 1.5 mm, 18 µm	
FR4	-
Single-sided, 1.5 mm, 35 µm	
ML104	1
Single-sided, 0.2 mm, 18 µm	
ML104	6
Single-sided, 0.2 mm, 5 µm	
Speed test	
Single-sided, 1.5 mm, 18 µm	

Fig. 285: Material selection

- 6. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 7. Click on 📑
- □ The dialog *Open* is displayed.
- 8. Navigate to the folder that contains the data you want to import.

- 9. Select the files you want to import.
- 10. Click on [Open].
- □ The data are automatically assigned to the correct layer and the user guidance step *Import* is displayed:

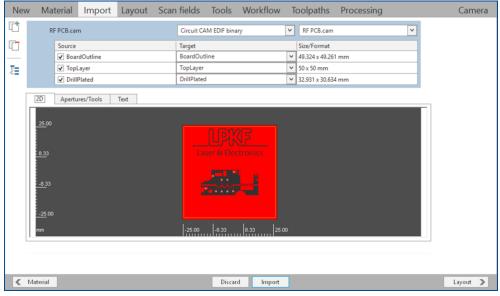


Fig. 286: User guidance step Import

- 11. Click on [Import].
- □ The layout is displayed in the user guidance step *Layout*.

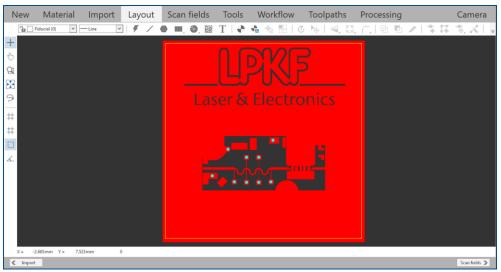


Fig. 287: Layout of an RF PCB

Before processing an RF PCB, the workflow settings need to be modified.

i

New	Material	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
New Top Side Mount M Work packa Work packa Work packa Name Type Instruction Message	v v v	V Material Placemen					Structure Top	Cutting Group	Camera
					Discard A	pply Compute toolpa	ths		Toolpaths 义

12. Switch to the user guidance step *Workflow*.

Fig. 288: User guidance step Workflow

13. Select the work package Structure Top.

Fig. 289: Work package Structure Top selected

ameters							
lsolate ✔ Generate		Printed circuit board -	indaries	Tools	_		
	TopLayer TextTop SolderMaskTop	Outline layer	BoardOutline			V Isolate	>
Isolation width	20 µm	Hatching Generate		V		Hatch	¥
Pad isolation	20 µm	Hatching layer	BoardOutline RuboutTop				V
	0 % olation 57 % Pad 0 %		L Mubbliop				Y
Isolation distance scan field to scan field	0 µm	Pattern Hatching gid (default ½ beam 0) Hatching contour overlap Use wobble in anrer Maximum channel width Extension for narrow channels at junctions Extension for terminat narrow channels	ed 0 %		Heat		¥

14. Expand the group Parameters.

Fig. 290: Group Parameters

15. In the sub-group Hatching select xy parallel from the drop-down list Pattern.

- 16. Enter 7.5 μm in the input field Hatching grid.
- □ The sub-group *Hatching* changes as follows:

Hatching	
✓ Generate	
Hatching layer	BoardOutline
	RuboutTop
Pattern	xy parallel 💙
Hatching grid (default: ½ beam Ø)	7.5 µm
Hatching contour overlap	100 %

Fig. 291: Sub-group Hatching

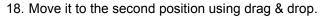


The order of the tools *Isolate* and *Hatch* needs to be reversed for this how-to example.

17. Select the tool Isolate.

ools -	_		
	PreCut	V PreCut	V
•	Isolate	V Isolate	~
•	Hatch	✓ Hatch	~
	Divide		
	ShortHeat	¥.	×
	Heat	Y	×

Fig. 292: Tool selected



To	ools –	_		
		PreCut	V PreCut	
	✓	Hatch	✓ Hatch	~
	✓	Isolate	V Isolate	~
		Divide	×	V
		ShortHeat	×	V
		Heat		×

Fig. 293: Tool moved



Alternatively, you can change the order of the tools via the drop-down lists.

- □ The order of the tools *Isolate* and *Hatch* has been reversed.
- 19. Click on [Apply].
- □ The workflow settings have been modified.
- 20. Switch to the user guidance step Toolpaths.
- 21. Click on [Compute toolpaths] or on [2].
- □ The dialog *Computing toolpaths* is displayed:
- 22. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 23. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 24. Click on [Close].
- 25. Save your project.
- The data have been prepared in CircuitPro PL.

Processing the PCB (with ProtoLaser)

1. Switch to the user guidance step Processing.

New			Processing	Camera
*		AX0/Ad255mm2ym		Sep size 0.500 mm 4 9 top size 0.500 mm 1 105.000 mm 2 0.000 mm
Toolg	paths	Start production		Camera 🔉

Fig. 294: User guidance step Processing

- Click on [Start production] or on ▶.
- 3. When the message *Processing phase: Mount material* is displayed, place the base material onto the processing table.
- 4. Click on 👑.
- □ The base material is fastened onto the processing table by vacuum.
- 5. Click on [OK].
- □ The dialog *Placement* is displayed.
- 6. Move the dialog *Placement* to get a better overview.
- 7. Determine the processing area. The base material position and base material size **must match** the processing area used by the CircuitPro PL software.



For detailed information on project placement by determining the processing area, refer to chapter 1.1.

- 8. Click on the tab Processing data.
- 9. Click on [Center on material].
- 10. Click on [OK].
- □ The laser system starts the material thickness measurement. The message *Thickness measurement running* is displayed.



For detailed information on material thickness measurement, refer to chapter 1.3.

When finished, the through holes are drilled and the dialog *Test tool settings* is displayed.



In this example, testing and setting the tools is to be skipped.

For detailed information on testing and setting the tools, refer to chapter 1.5.

- 11. Click on [Resume].
- □ Structuring the Top side (*TopLayer*) is started.
- 12. When the message *Board production finished* is displayed, remove the PCB from the system.
- 13. Spray the PCB with LPKF Cleaner and use a brush to clean it.
- 14. Rinse the PCB with tap water and dry it with compressed air.
- The PCB has been processed.

The RF PCB production is finished.

Processing files



5 Processing files

This chapter describes how to import different file formats (CAD files) and prepare them for processing. It also shows typical errors during the import and suitable measures for troubleshooting.

5.1 Processing DXF files in CircuitPro PL



Processing DXF files is only available in the license level Advanced of CircuitPro PL. If you have any questions contact the LPKF sales department.

This chapter describes how to import and convert DXF files in CircuitPro PL.

The following steps are performed in this tutorial:

- Importing DXF files
- Combining open paths
- Modifying the workflow settings
- Computing toolpaths

Importing DXF files

1. In the user guidance step *New*, select the template that suits the type of project you have.

In this example the template Single-sided, Top, ProtoLaser U4 is used.

- 2. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 3. Select the material AL2O3_Au22, Single-sided, 0.5 mm, 22 µm.

	2O3_Au22 ngle-sided, 0,5 mm, 22 μm	8	^
	2O3_Cu25 ngle-sided, 0.5 mm, 25 µm	â	
	2O3_Cu300 ngle-sided, 1.5 mm, 300 μm	6	
	2O3_Cu9 ngle-sided, 0.5 mm, 9 μm	â	
	nrome mask on glass_fast ablation ngle-sided, 2.3 mm, 0,1 μm	A	
	rrome mask on glass_high quality abla ngle-sided, 2.3 mm, 0,1 μm	6	
	cus determination ngle-sided, 1.5 mm, 18 μm	â	
FR	4 ngle-sided, 1.5 mm, 18 μm	â	
FR Sir	4 ngle-sided, 1.5 mm, 35 μm	â	
	D_fast ablation ngle-sided, 0.2 mm, 0,1 μm	â	
ITC	D_high quality ablation	6	~

Fig. 295: Select material

4. Click on [Select material] or double-click on the material.

- □ The user guidance step *Import* is displayed.
- 5. Click on T.
- 6. Navigate to the folder that contains the data you want to import. The example data used for this tutorial are located in the folder: C:\Program Files\LPKF Laser & Electronics AG\

LPKF CircuitPro PL\Example Data\UseCase_DXFFiles.

- 7. Select the files you want to import (in this example Tutor_dxf.dxf).
- 8. Click on [Open].
- □ The user guidance step *Import* is displayed.
- 9. In the drop-down list of the column *Target*, select the entry *CutInside*.

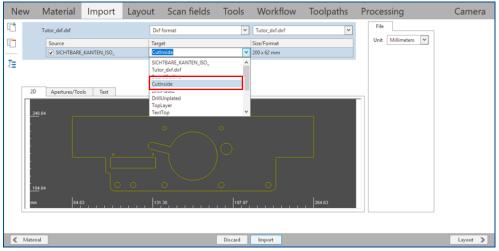


Fig. 296: Select CutInside

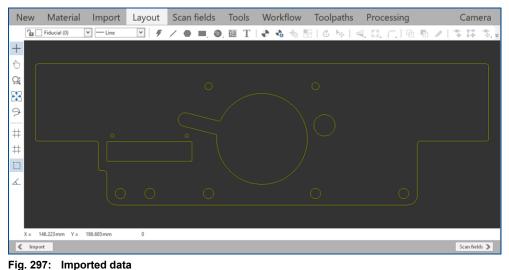
□ The data to be imported are assigned to the layer *CutInside*.

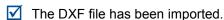
In the column *Size/Format* check the measurement unit which the DXF file was originally created with (in this example it is mm). Make sure the same unit is displayed in the sub-tab *File* under *Unit*. Should this not be the case, select the correct unit from the drop-down list.

10. Click on [Import].

i

□ The imported data is displayed in the user guidance step *Layout*.





Combining open paths

i

- Data imported from a DXF file can consist of **open paths**. To avoid problems during processing, open paths need to be **converted** to **closed paths**.
- 1. Select the entire layout.

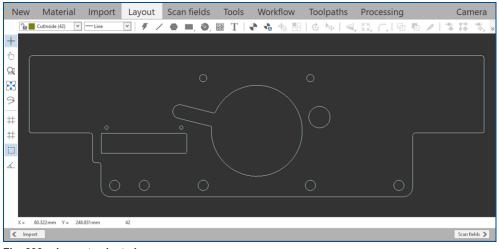


Fig. 298: Layout selected

2. Click on the right mouse button and in the context menu select the entry *Combine open paths*.

8	Cut	Strg+	
b	Сору	Strg+0	C
	Invert select	tion	
0	Group		
▶	Move object	t Strg+N	٨
Ġ	Rotate		
	horizontally	1	
	vertically		
5 A 2 3	Scale		
	Step and rep	peat	
B ii	Create insta	ince type	
3	Expand		
5	Merge		
1	Split		
⇒	Convert to	polygon	
1	Close open	path	
	Combine of	pen paths	
	Convert to f	flash	
	Combine to	flash	
	Assign to la	iyer	•
	Copy to lay	er	٠
	Create aper	ture	
	Select by ge	eometry	
·	Measure		
X	Delete		

The open paths have been combined.

i

Modifying the workflow settings

Before processing the DXF file, the workflow settings need to be modified.

1. Switch to the user guidance step *Workflow*.

Top Side						-	-	
√ Mount Mate	rial	\rightarrow	Drill Fiducials	Drill Through	→ Holes	Structure Top	Cut Inside	ur
Type Instruction	options Mount Material Mount material Mount the base material onto the processing area.							
Tools				scard Ap	Ormpute toolpat			Toolpaths 🔉

Fig. 300: User guidance step Workflow

2. Select the work package Cut Inside.

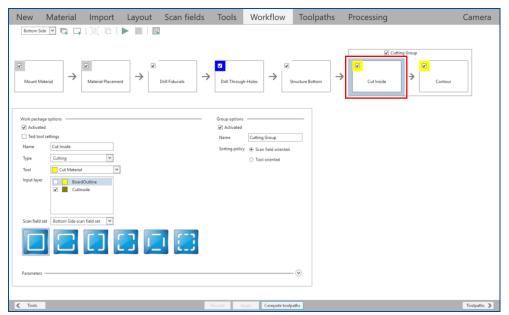


Fig. 301: Selecting the work package *Cut Inside*

3. Expand the group Parameters.

Parameters	
Position	Inside 🗸
Channel width	100 µm
Target beam overlap	10 %
Actual beam overlap	19 %
Sorting	Outside in 🔽 🗌 Unidirectional
Tab positions	NoGaps
Tab width	0 mm
Distance	0 mm



4. Select the entry Center from the drop-down list Position.

Parameters	®
Position	Center
Channel width	100 µm
Target beam overlap	10 %
Actual beam overlap	19 %
Sorting	Outside in 🔽 🗌 Unidirectional
Tab positions	NoGaps 🗸
Tab width	0 mm
Distance	0 mm

Fig. 303: Entry Center selected

- 5. Click on [Apply].
- The workflow settings have been modified.



Modifying the workflow settings

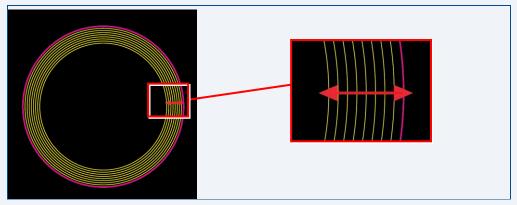
The most important parameters when modifying the workflow settings before processing DXF files are *Position* and *Channel width*.

Parameters		
Fordificters		
Position	Inside	~
Channel width	100 µm	
Target beam overlap	10 %	
Actual beam overlap	19 %	
Sorting	Outside in	V Unidirectional
Tab positions	NoGaps	~
Tab width	0 mm	
Distance	0 mm	

Option	Definition	Example of toolpaths
Inside	The toolpaths are located inside the DXF object.	
Outside	The toolpaths are located outside the DXF object.	
Center	The toolpaths are located outside and inside of the DXF object.	

The parameter *Position* defines the position of the toolpaths (in yellow) in comparison to the DXF object (in magenta). There are three options:

The parameter *Channel width* defines the width of the cutting channel on the material.



Computing toolpaths

- 1. Switch to the user guidance step *Toolpaths*.
- 2. Click on [Compute toolpaths] or on 🕅.
- □ The dialog *Computing toolpaths* is displayed.
- 3. Click on [Continue].
- □ The toolpaths are being calculated. The message *Computation results* is displayed.
- 4. Check the computation results for any possible warnings or errors and make corrections, if needed.
- 5. Click on [Close].
- 6. Save your project.
- The toolpaths have been computed.

The DXF file has successfully been imported in and converted for CircuitPro PL.

5.2 Processing Gerber and Ecxellon files in CircuitPro PL

This chapter describes how to process Gerber and Excellon files in CircuitPro PL. It also shows typical errors during the import and suitable measures for troubleshooting.

The following steps are performed in this tutorial:

- Importing DXF files
- Selecting the file format
- Assigning the desired target layer during import
- Assigning the desired target layer after import
- Setting/Correcting the size and format
- Checking/Modifying aperture properties
- Using layer names from the Gerber file for import
- Using layer names from the Gerber file as default

Selecting Gerber and Excellon files

1. In the user guidance step *New*, select the template that suits the type of PCB you want to process.

In this example the template Single-sided, Top, ProtoLaser U4 is used.

- 2. Click on [Load template] or double-click on the template.
- □ The user guidance step *Material* is displayed.
- 3. Select the suitable material.

In this example the material *FR4*, *Sigle-sided*, *1.5 mm*, *18* μ *m* is used.

Al2O3_Au22 Single-sided, 0.5 mm, 22 µm	â
Al2O3_Cu25 Single-sided, 0.5 mm, 25 µm	â
Al2O3_Cu300 Single-sided, 1,5 mm, 300 µm	â
Al2O3_Cu9 Single-sided, 0.5 mm, 9 μm	â
Chrome mask on glass_fast ablation Single-sided, 2.3 mm, 0,1 µm	â
Chrome mask on glass_high quality ablation Single-sided, 2.3 mm, 0,1 µm	a
Focus determination Single-sided, 1.5 mm, 18 µm	â
FR4 Single-sided, 1,5 mm, 18 µm	8
FR4 Single-sided, 1.5 mm, 35 µm	â
ITO_fast ablation Single-sided, 0.2 mm, 0,1 μm	â
ITO_high quality ablation Single-sided, 0.2 mm, 0,1 μm	ô
ML104	

Fig. 304: Selecting the material

- 4. Click on [Select material] or double-click on the material.
- □ The user guidance step *Import* is displayed.
- 5. Click on [.]

- □ The dialog *Open* is displayed.
- 6. Navigate to the folder that contains the data you want to import. The example data used for this sequence are located in the folder: C:\Program Files\LPKF Laser & Electronics AG\ LPKF CircuitPro PL\Example Data\UseCase GerberExcellonFiles.
- 7. Select the files you want to import (in this example: *Tutor.BOA, Tutor.BOT, Tutor.DRL, Tutor.SMB, Tutor.SMT, Tutor.SPT, Tutor.TOP*).
- 8. Click on [Open].
- □ The user guidance step *Import* is displayed.

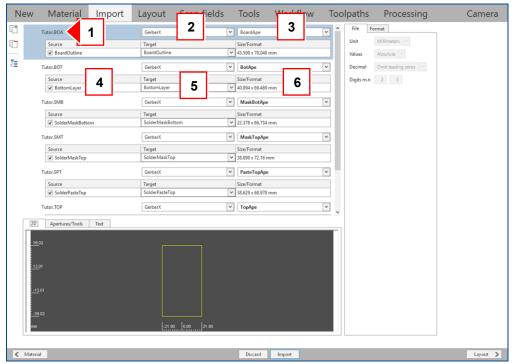


Fig. 305: User guidance step Import



The following information on the imported files is displayed:

ltem	Information	Description
		•
1	File name	Displays the name of the imported file.
2	Format	Displays the format of the selected file. If CircuitPro PL has not recognized the file format correctly, you can assign the correct format in the corresponding drop-down list.
3	Aperture/Tool list	Displays the aperture list/tool list that is usually a part of the Gerber or Excellon file. The apertures contained are displayed in the tab <i>Apertures/Tools</i> .
4	Source	Indicates the names of the paths/3D elements within the imported file. Activate the check boxes of the files that you want to import.
5	Target	Displays the layer to be assigned.
6	Size/Format	Displays the size/format of the imported layer.

- 9. Deactivate the check boxes of the files you do not want to import (in this example: *Tutor.BOT*, *Tutor.SMB*, *Tutor.SMT* and *Tutor.SPT*).
- The files have been deselected.
 The user guidance step *Import* changes as follows:

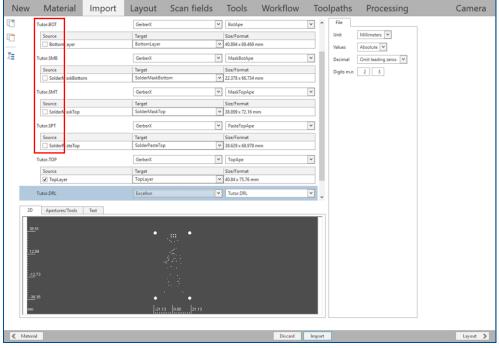


Fig. 306: Deselected files

The Gerber and Excellon files have been selected.

Selecting the file format

In some cases it is possible that the format of the selected files is not recognized correctly.



Note that the file problem described in this procedure was created intentionally. The files in the folder *Example Data* do not contain these problems.

- 1. In the dialog *Open*, select the files you want to import (in this example: *Tutor.BOA*, *Tutor.DRL* and *Tutor.TOP*).
- 2. Click on [Open].
- □ In the user guidance step *Import*, the file format as well as other file-specific information are missing (= *Undefined*). A yellow warning symbol is displayed next to the file name:

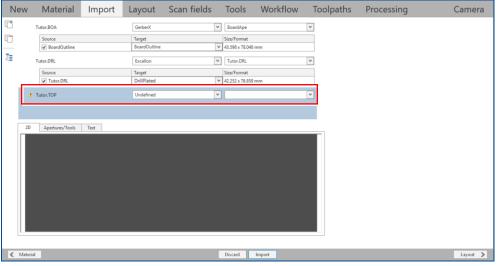


Fig. 307: Format undefined

3. Select the appropriate file format from the drop-down list (in this example *GerberX*):

N	ew	Material	Import	Layout	Scan fields	Tools	Workflow	Toolpaths	Processing	Camera
[]		Tutor.BOA		GerberX	×	BoardApe		*		
		Source BoardOutline		Target BoardOutline	v	Size/Format 43.598 x 78.048 m	ım			
Æ		Tutor.DRL		Excellon	v	Tutor.DRL		v		
		Source Tutor.DRL		Target DrillPlated	v	Size/Format 42.252 x 76.859 m	ım			
		Tutor.TOP		Undefined	~			~		
				GerberX	Î					
	2D	Apertures/Tools	Text	Excellon S&M Dxf format						
				HP-GL data fo ODB++						
				LPKF_Mill_Dril Circuit CAM E IPC format						
				JOB BoardMas	ter 🗸					
		_								
<	Material					Discard	mport			Layout 🔰

Fig. 308: Selecting the appropriate file format

□ The missing file-specific information is displayed automatically.

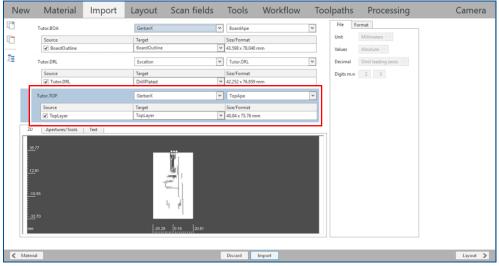


Fig. 309: Appropriate file format selected

The file format has been selected.

If CircuitPro PL does not assign the file to the desired layer automatically, there are two ways to assign it manually. This takes place either during or after import of the files. Both possibilities are described as follows.

Assigning the desired target layer during import

- 1. In the dialog *Open*, select the files you want to import (in this example: *Tutor.BOA*, *Tutor.DRL* and *Tutor.TOP*).
- 2. Click on [Open].
- □ In the user guidance step *Import*, the file *Tutor.DRL* has not been automatically assigned to an existing layer (the column *Target* is active and contains the name of the file):

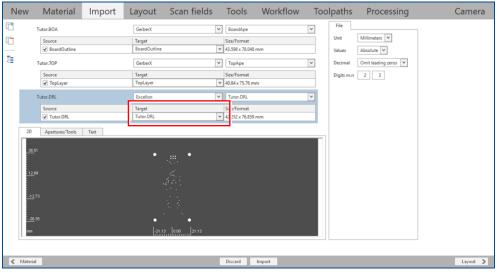


Fig. 310: Unassigned layer

3. Select the desired target layer from the drop-down list (in this example DrillPlated):

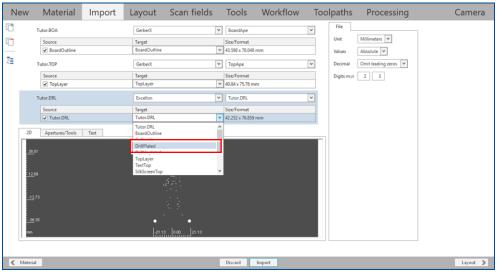
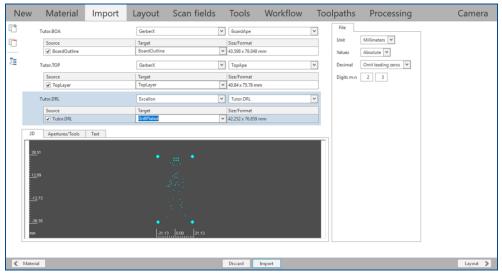
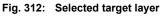


Fig. 311: Selecting the target layer



□ The user guidance step *Import* changes as follows:



4. Click on [Import].

□ The processing data are displayed in the user guidance step *Layout*.

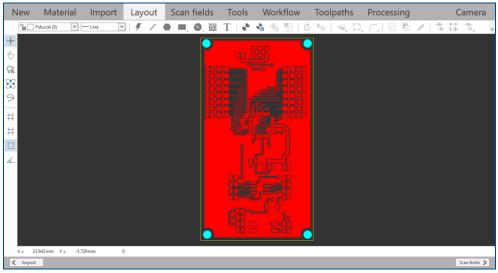


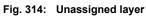
Fig. 313: User guidance step Layout

The desired target layer has been assigned during import.

Assigning the desired target layer after import

- 1. In the dialog *Open*, select the files you want to import (in this example: *Tutor.BOA*, *Tutor.DRL* and *Tutor.TOP*).
- 2. Click on [Open].
- □ In the user guidance step *Import*, the file *Tutor.DRL* has not been automatically assigned to an existing layer:

New	Material	Import	Layout	Scan fields	Tools	Workflow	Тоо	lpaths	Processing	Camera
[] [] [注]	Tutor.BOA Source		GerberX Target BoardOutline GerberX Target TopLayer Excellon Target Tutor.DRL	v v v	BoardApe Size/Format 43.598 x 78.048	mm	V	File Unit Values Decimal Digits m.n	Millimeters V Absolute V Omit leading zeros V	
2D		Text								
Material					Discard	Import				Layout 🔉



- 3. Click on [Import].
- □ The following message is displayed:

Info ×
During import, these new layers have been created: Tutor.DRL
Assign the correct PCB layer and technology to them.
Close

Fig. 315: Info about new layer created

4. Click on [Close].

□ A new layer *Tutor.DRL* has been created and the processing data are displayed in the user guidance step *Layout*.

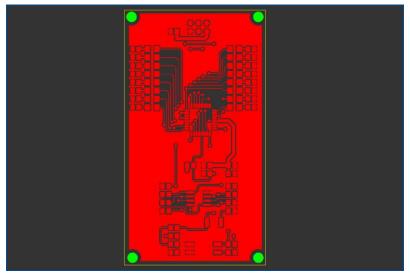


Fig. 316: User guidance step Layout | processing data

- 5. In the pane *Workflow setup*, expand the group *Layout* and expand the node *Layers*.
- □ A list of layers is displayed.
- 6. Right-click on the layer *Tutor.DRL (64)* and click on the context menu item *Select by layer.*

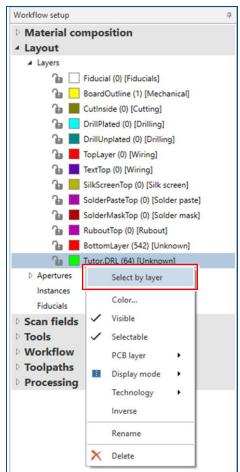
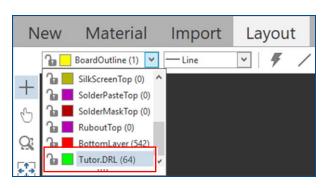


Fig. 317: List of layers | Select by layer

i

Alternatively, you can select the layer from the drop-down list:



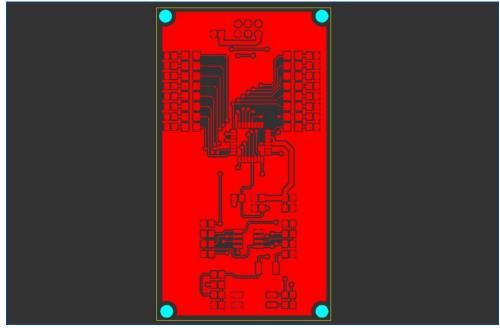
Right-click on the black background and select the context menu item Select by layer.

- □ All objects on this layer are selected and highlighted.
- 7. Right-click on the black background.
- 8. In the context menu, click on Assign to layer > DrillPlated:

Ж	Cut	Ctrl+X	
	Сору	Ctrl+C	
	Invert selection		
D.	Group		-
▶	Move object	Ctrl+M	
Ċ	Rotate		
	horizontally		
	vertically		
к X К У	Scale		
	Step and repeat		
	Create instance ty	pe	_
	Expand		
5	Round		
(Chamfer		
5	Merge		
0	Split		
¢	Convert to closed	path	
	Combine to flash		
	Explode flash		Fiducial
₽3	Create fiducial wit	th defaults	BoardOutline
	Assign to layer	•	DrillPlated
	Copy to layer		DrillUnplated
	Create aperture		TopLayer
	Select by aperture		TextTop
	Select by geometr	ry	SilkScreenTop
·	Measure		SolderPasteTop
×	Delete		SolderMaskTop
			RuboutTop

Fig. 318: Assigning objects to target layer

9. In the user guidance step *Layout*, click anywhere on the black background or press *Esc* to deselect the highlighted objects.



□ The color of the holes changes to turquoise.

Fig. 319: Assigned holes

The desired target layer has been assigned after import.

Setting import assignments

During file import, some files are automatically assigned to target layers and the column *Target* is inactive. This is a result of *Import assignments* settings. If you frequently use file names with the same suffixes, you can adapt the *Import assignments* to your requirements.

Perform the following steps:

- 1. Click on File > Options...
- □ The dialog *Options* is displayed.
- 2. Click on Import assignments.

□ The dialog changes as follows:

All settings								_
General CDisplay Grid	C:\Users\User name\D	locuments\LPKF Laser & Electro	onics\LPKF Cir	cuitPro PL\ApertureTemp	ates.xml		***	
Snap Snap Import / Export Import assignments	C:\Users\User name\D	locuments\LPKF Laser & Electro	onics\LPKF Cir	rcuitPro PL\Import Assignr	nents\Ea	igle.xml		
Formats Gerber Excellon						New S	iave As.	
LMD	File name	Format		Layer/Template		Aperture/Tool list		1
HPGL DXF	*.TOP	GerberX	~	TopLayer	~	ТорАре	¥	1
CP2D	*.BOT	GerberX	~	BottomLayer	~	BotApe	~	1
Miscellaneous	*.BOA	GerberX	~	BoardOutline	~	BoardApe	~	1
Machine	*.SST	GerberX	~	SilkScreenTop	~	SilkTopApe	~	1
🖌 🗀 Logging	*.SSB	GerberX	~	SilkScreenBottom	*	SilkBotApe	~	1
Logging	*.SMT	GerberX	~	SolderMaskTop	~	MaskTopApe	~	ł
General	*.SMB	GerberX	~	SolderMaskBottom	~	MaskBotApe	~	1
	*.SPT	GerberX	~	SolderPasteTop	~	PasteTopApe	~	ŀ
	Adjust drill files					Add	Remove	e

- 3. Click on [New].
- □ The dialog *Save As* is displayed.
- 4. Enter the new name of the .xml file and click on [Save].
- □ The dialog changes as follows:

		Options		- 🗆 🗙
Call settings General Display Grid	C:\Users\User name\Do	ocuments\LPKF Laser & Electroni	ics\LPKF CircuitPro PL\ApertureTemplat	tes.xml
Snap Snap Import / Export Import assignments	C:\Users\User name\D	ocuments\LPKF Laser & Electron	ics\LPKF CircuitPro PL\Import Assignme	ents\New.xml
Import assignments Formats				New Save As
Gerber Kxcellon LMD HPGL DXF CP2D Miscellaneous Machine Logging General	File name	Format	Layer/Template	Aperture/Tool list Add Remove
				OK Cancel

- 5. Click on [Add].
- 6. Enter the suffix of the file name in the column *File Name*.

- 7. Select the corresponding values from the drop-down lists in the columns *Format*, *Layer/Template* and *Aperture/Tool* list.
- 8. Repeat the steps 6 and 7 for creating further Import assignments.
- 9. When you have finished creating the desired Import assignments, click on [OK].
- □ The following figure displays an example of an import assignment setting for the file names with the suffix *.*CUT*:

File Name /	Format	Layer/Template	Aperture/Tool list
*.CUT	GerberX 🗸	BoardOutline 💌	BoardApe 🗸

Setting/Correcting the size and format

In some cases, the layout of the file is not displayed correctly in CircuitPro PL. There are four possible causes for this:

- Incorrect measurement unit: A wrong measurement unit was selected when importing the file (inch instead of mm).
- Incorrect number of decimal digits: The number of decimal digits entered does not match the file's contents.
- Incorrect declaration of the value (relative/absolute).
- Incorrect zero suppression (decimal).

Note that the file problem described in this procedure was created intentionally. The files in the folder *Example Data* do not contain these problems.

- 1. In the dialog *Open*, select the files you want to import (in this example: *Tutor.BOA*, *Tutor.DRL* and *Tutor.TOP*).
- 2. Click on [Open].
- □ In the user guidance step *Import*, the file *Tutor.DRL* displays a peculiarly large size of the layout:

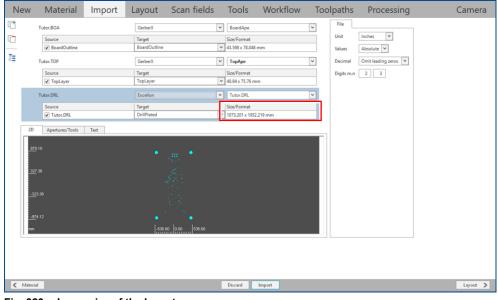


Fig. 320: Large size of the layout



The measurement unit for GerberX files is preset as default and cannot be modified. The measurement unit for all other file types (Excellon, Gerber etc.) can be modified.

- 3. Check the *Unit* in the sub-tab *File* of the user guidance step *Import* (in this example *Inches* are selected).
- New Material Import Layout Scan fields Tools Workflow Toolpaths Processing Camera C GerberX Tutor.BOA ♥ BoardApe ~ Unit 1 Source BoardOutline Target BoardOutline Size/Format 43.598 x 78.048 mm Value 阳 neros Y Tutor,TOP GerberX ✓ TopApe ~ Decimal Digits m.n 2 3 Source Size/Format 40.84 x 75.76 mm Target TopLayer ▼ Tutor.DRL Tutor.DRL Source Target DrillPlated Size/Format V 1073.201 x 1952.219 mm 2D Apertures/Tools Text Discard Import Layout 🔰
- 4. Select Millimeters from the drop-down list.

Fig. 321: Selecting the appropriate unit

□ The dimensions of the layout change automatically:

lew	v Material Impor	t Layout Scan f	ields Tools Workfl		olpaths Processing	Came
	Tutor.BOA	GerberX	✓ BoardApe	×	File	
	Source	Target	Size/Format		Unit Millimeters ¥	
	✓ BoardOutline	BoardOutline	43.598 x 78.048 mm		Values Absolute 💙	
	Tutor.TOP	GerberX	TopApe	~	Decimal Omit leading zeros	
	Source	Target	Size/Format		Digits m.n 2 3	
	✓ TopLayer	TopLayer	✓ 40.84 x 75.76 mm			
	Tutor.DRL	Excellon	Y Tutor.DRL	~		
	Source	Target	Size/Format			
	✓ Tutor.DRL	DrillPlated	✓ 42.252 x 76.859 mm			
	<u>38</u> 51 1 <u>12</u> 89					
	-38.35					
	nn	21.13 0.00 2	1.13			
	terial		Discard Import			Layout

Fig. 322: Changed dimensions of the layout

□ The measurement unit has been checked and corrected.

If the layout is still not displayed correctly in the tab *2D*, you should check the number of decimal digits. An incorrect preview of the layout data is displayed as follows:

Layout 🔰

Fig. 323: Incorrect preview of the layout

- 5. Check the *Digits m.n* in the sub-tab *File* of the user guidance step *Import*.
- 6. Enter 3 in the *n* digit count.
- □ The user guidance step *Import* changes as follows:

ew	Material Imp	ort Layout Scan	neids Tools	Workflow To	oolpaths	Processing	Came
	Tutor.BOA	GerberX	♥ BoardApe	*	File		
	Source	Target	Size/Format		Unit	Millimeters ¥	
	✓ BoardOutline	BoardOutline	✓ 43.598 x 78.0	48 mm	Values	Absolute 💙	
	Tutor.TOP	GerberX	✓ TopApe	~	Decimal	Omit leading zeros	
	Source	Target	Size/Format		Digits m.n	2 3	
	✓ TopLayer	TopLayer	✓ 40.84 x 75.76	mm			
	Tutor.DRL	Excellon	✓ Tutor.DRL	×			
	Source	Target	Size/Format				
	✓ Tutor.DRL	DrillPlated	✓ 42.252 x 76.8	59 mm			
<u>-12</u> 8 - <u>12</u> 7 rm		21.13 000	21.13				

Fig. 324: Correct preview of the layout

- □ The number of decimal digits has been checked and corrected.
- The size and format have been set/corrected.

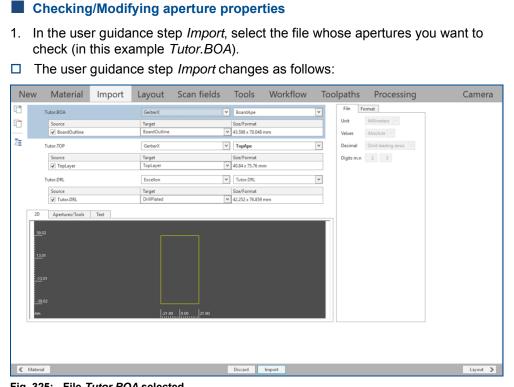


Fig. 325: File Tutor.BOA selected

- 2. Click on the tab Apertures/Tools and then on the sub-tab Attributes.
- □ The user guidance step *Import* changes as follows:

New	Material	Import	Layout	Scan fields	Tools	Workflow	Тос	olpaths Processing	Camera
	Tutor.BOA		GerberX		♥ BoardApe		~	File Format Attributes	
	Source		Target		Size/Format			Type: Circle 💙	
	✓ BoardOutline		BoardOutline		✓ 43.598 x 78.04	8 mm		Rotation: 0*	
_	Tutor.TOP		GerberX		TopApe		¥	a: 0.1 mm	
	Source		Target		Size/Format			b:	
	✓ TopLayer		TopLayer		✓ 40.84 x 75.76 r	nm		c	
	Tutor.DRL		Excellon		✓ Tutor.DRL		~	d:	
	Source		Target		Size/Format				
	✓ Tutor.DRL		DrillPlated		¥ 42.252 x 76.85	9 mm			
2D	Apertures/Tools	Text						a	
Image	e Name	Mode	Rotation	a b	c	d			
	O D10		0 *	0.1 mm					
Ļ									

Fig. 326: Tab Apertures/Tools and sub-tab Attributes



The following aperture properties can be modified in the sub-tab Attributes:

- Shape of the aperture (circle, square, oval etc.) _
- aperture rotation _
- parameters of the aperture geometry

In this example, the aperture's shape is to be changed from a circle to a rectangle.

3. In the sub-tab Attributes under Type, select Rectangle from the drop-down list.

New	1	∕late	erial	Im	port	Layout	Scan fields	;	Tools	Wor	kflow	Тоо	lpat	hs	Processi	ng	Camera
	Tuto S Tuto Tuto D	r.BOA burce] Board r.TOP burce] TopLa r.DRL burce] Tutor. Aperture	Outline yer DRL	Text		Layout GeberX Target BoardOytline GeberX TropLayet Draget DraliPlated DrailPlated 0*		> > > >	BoardApe Size/Format 43.598 x 78.0 TopApe Size/Format 40.84 x 75.7 Tutor.DRL Size/Format 42.252 x 76.0	:)48 mm : 5 mm	d d	V V	Ipat File Type Rotai a: b: c: d:	F tion:	Processi ormat Attributes Circle V Rectangle Regular polygon Octagon Bevel rectangle Bevel rectangle Bevel rectangle	5	Camera
K Mat	erial								Discard	Import							Layout 🔰

Fig. 327: Selecting the type of aperture

4. Enter 1.5 in the input field a:.

A preview of the modified aperture is displayed in the sub-tab *Attributes*:

Ne	W	Materi	al In	nport	Layout	Scan field	s	Tools	Workf	flow	То	olpaths	Processing	 Camera
ľ	Т	futor.BOA			GerberX			BoardApe			~	File F	Rectangle	
Ū		Source BoardOut	line		Target BoardOutline			Size/Formal 43.598 x 78.				Rotation:	0*	
足	Т	futor.TOP			GerberX			ТорАре			~	a: b:	1.5 mm	
		Source TopLayer			Target TopLayer			Size/Format 40.84 x 75.7				c		
	Т	Source			Excellon Target		_	Tutor.DRL Size/Format			~	d:		
		✓ Tutor.DRL			DrillPlated			42.252 x 76.					a i	
]	2D Image	Apertures/Te		Mode	Rotation	a	b		c	d			b	
	1	D10			0*	1.5 mm	0.0001	mm						
<	Material							Discard	Import					Layout 🔰

Fig. 328: Preview of the modified aperture

☑ The aperture properties have been checked/modified.

The Gerber file format has an input field for the layer name. There are two ways to use the layer name for an import. Either you define for each import whether the layer name from the Gerber file is to be used or you set this function as default in the options. Both possibilities are described as follows.

The layer name defined in the Gerber file is displayed in the tab *Text* of the user guidance step *Import* (in this example the file *Tutor.BOA* which contains the layer name *BoardOutline* is used):

New	Material	Import	Layout	Scan fields	Tools	Workflow	Тоо	lpaths	Processing	Camera
rt 👘	Tutor.BOA		GerberX	۷	BoardApe		~	_	ormat	
1	Source		Target BoardOutline	~	Size/Format 43.598 x 78.048	mm		Unit Values	Millimeters V Absolute V	
ξ	Tutor.TOP		GerberX	~	ТорАре		~	Decimal	Omit leading zeros	
	Source		Target TopLayer		Size/Format 40.84 x 75.76 m			Digits m.n	2 3	
	Tutor.DRL		Excellon	v	Tutor.DRL		~			
	Source		Target DrillPlated		Size/Format 42.252 x 76.859					
2D	Apertures/Tools	Text	Dhiiriated		42-232 X /0.839	mm				
P6FSL	X23Y23*% MM*%									
G54D	oardOutline*%									
X-217 X-217	9Y-38974D02* 49Y-38974D01* 49Y38974D01*									
X2174 X2174 M02*	9Y38974D01* 9Y-38974D01*									
K Materi	4				Discard	Import				Layout 🔰

Fig. 329: Layer name in the file Tutor.BOA

Using layer names from the Gerber file for import

- 1. In the dialog *Open*, select the files you want to import (in this example: *Tutor.BOA*, *Tutor.DRL* and *Tutor.TOP*).
- 2. Click on [Open].
- □ The user guidance step *Import* is displayed. The layer names defined in the Gerber file are not used and not displayed in the column *Target* and in the column *Source*:

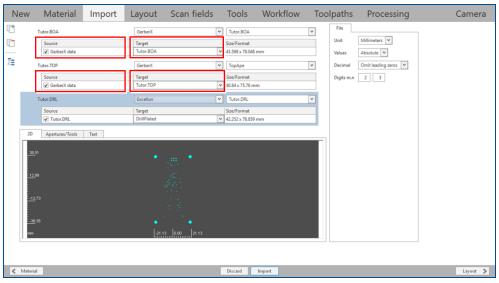


Fig. 330: Layer names not used in the Gerber file

3. Select one of the Gerber files (in this example *Tutor.BOA*) and switch to the sub-tab *Format*.

New Material Impor	rt Layout Scan fields Tools Workflow Toolpaths Processing	Camera
New Material Impor	rt Layout Scan fields Tools Workflow Toolpaths Processing Gerbex Gerbex Target Target Gerbex	Camera
nei	22.000	
▲ Material	Discard Import	Layout 义

Fig. 331: Gerber file selected

4. Activate the check boxes Use layer name and Apply to all Gerber files.

File Format				
✓ Use layer name				
✓ Use layer polarity				
Rotate square in movement direction				
Step and repeat to flash				
Rotate AM-Octagon				
Set 360 interpolation as default				
✓ Apply to all Gerber files				

Fig. 332: Activated check boxes

□ The layer names defined in the Gerber files are displayed in the column *Target* and in the column *Source*:

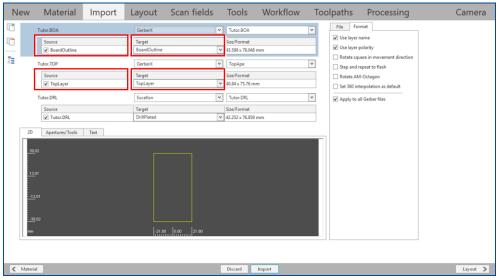


Fig. 333: Layer names defined in Gerber files displayed

The layer names from the Gerber file have been used for import.

Using layer names from the Gerber file as default

- 1. Click on File > Options...
- □ The dialog *Options* is displayed.
- 2. Click on Formats > Gerber.

		Options			-		×
All settings General Display Grid Snap Import / Export Import assignments Formats Gerber	~		True False False False False False		-		×
Gerber Excellon LMD HPGL DXF CP2D Miscellaneous Machine Logging							
Cogging		. Use layer polarity tivate/Deactivate analysis of layer pola	rrity during Gerber				
				OK		Canc	el

Fig. 334: Dialog Options | Gerber

- Options Import 🖌 🗀 All settings General 1.1. Use layer polarity True 1.2. Rotate square False 🔺 🗀 Display 1.3. Step and repeat to flash False 🗋 Grid 1.4. Rotate AM-octagon False 🗋 Snap 1.5. Use layer name False Import / Export 1.6. 360-degree interpolation as d Import assignments True False Formats 🗋 Gerber T Excellon HPGL DXF CP2D Miscellaneous Machine 🔺 🗀 Logging Logging 1.5. Use laver name Use layer name definition from Gerber file. General Cancel OK
- 3. Set the value in line 1.5 Use layer name to True.

Fig. 335: Selecting value True for Use layer name

If the value *False* is selected, the option is not activated as default. In such cases, when importing Gerber files, you must always define whether you want to use the layer names contained in the files.

- 4. Click on [OK].
- □ The option for the default use of the layer name contained in the Gerber file has been activated.
- 5. In the dialog *Open*, select the files you want to import (in this example: *Tutor.BOA*, *Tutor.DRL* and *Tutor.TOP*).
- 6. Click on [Open].
- □ The user guidance step *Import* is displayed. The layer names defined in the Gerber file are displayed in the column *Target* and in the column *Source*:

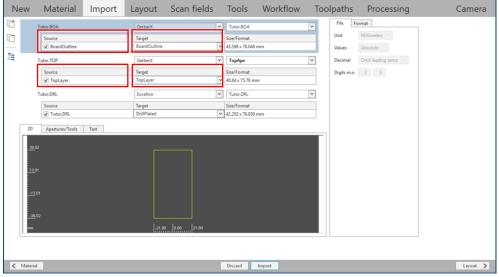


Fig. 336: Layer names defined in Gerber files displayed

The layer names from the Gerber file have been used as default.

The Gerber and the Excellon files have been processed in CircuitPro PL.

6 Appendix

This chapter contains navigation elements of the document.

6.1 List of figures

Fig. 1:	Fiducial in the view Camera	
Fig. 2:	Inaccurate position of the layout (example image)	13
Fig. 3:	Matching processing head and fiducial position	13
Fig. 4:	Matching position of crosshair to base material edge	14
Fig. 5:	Recognized fiducial	15
Fig. 6:	Message Spiral search active	15
Fig. 7:	Manual fiducial search	16
Fig. 8:	Visible fiducial	16
Fig. 9:	Recognized fiducial	
Fig. 10:	Fiducial of poor quality	19
Fig. 11:	Matched position of poor quality fiducial	
Fig. 12:	Dialog Material thickness measurement settings (default settings)	
Fig. 13:	Drop-down list Measurement strategy	
Fig. 14:	Dialog Placement	
Fig. 15:	Message Thickness measurement running	
Fig. 16:	Dialog Measurement result confirmation	
Fig. 17:	Context menu	
Fig. 18:	Dialog Placement	
Fig. 19:	Message Thickness measurement running	
Fig. 20:	Measured value	
Fig. 21:	Entering Material thickness	
Fig. 22:	Measured values and average value of the reference points	
Fig. 23:	Select material	
Fig. 24:	User guidance step Import	
Fig. 25:	Imported PCB layouts	
Fig. 26:	One PCB layout selected	
Fig. 27:	Menu Edit instance type	
Fig. 28:	Dialog Edit instance type	
Fig. 20:	Dialog Edit instance type Values	
Fig. 30:	Layout multiplied	
Fig. 31:	User guidance step Processing	
Fig. 32:	Automatic arrangement of PCB layouts	
Fig. 33:	Menu item Placement	
Fig. 34:	Dialog Placement Processing data	
Fig. 35:	Distance between PCB layouts increased	
Fig. 36:	Place and process "Small test sample 7x7 mm"	
Fig. 37:	Specifying the position	
Fig. 38:	Dialog Test tool settings	
Fig. 39:	Dialog Check test tool finished	
Fig. 40:	Tune tool settings	
Fig. 41:	View Tool Settings	
Fig. 42:	Tool selected	
Fig. 43:	Modifying the tool settings	
Fig. 44:	Copy of the tool with the modified settings	
Fig. 45:	Saving the tool	
•	•	
Fig. 46: Fig. 47:	Tool saved Resuming the processing procedure	
Fig. 47. Fig. 48:	Phase Bottom Side selected	
Fig. 48. Fig. 49:	Work package Structure Bottom selected	
-		
Fig. 50:	Test tool settings disabled Assembly of the press mold and the materials	
Fig. 51:	הספרווטוע טו נווב אובפא וווטוע מווע נוופ ווומנפוומוא	49

Fig. 52:	Example of a laminated PCB material	
Fig. 53:	Example of a non-laminated PCB material	. 51
Fig. 54:	PCB layout	
Fig. 55:	PCB layout isolation channel	
Fig. 56:	PCB layout creating strips	
-		
Fig. 57:	PCB layout delaminating strips	
Fig. 58:	RF PCB layout RF geometry	
Fig. 59:	PCB layout Isolation channel	. 54
Fig. 60:	PCB layout Hatching	. 55
Fig. 61:	A 4-layer PCB structure	. 56
Fig. 62:	A 4-layer PCB structure with blind vias and buried vias	
Fig. 63:	A 6-layer PCB structure	
-	,	
Fig. 64:	A 6-layer PCB structure with blind vias and buried vias	
Fig. 65:	Annular ring	
Fig. 66:	A through hole with external and internal annular rings	
Fig. 67:	Incorrect inner layer fill in the user guidance step Layout	
Fig. 68:	Correct inner layer fill in the user guidance step Layout	. 60
Fig. 69:	Staggered vias and minimum distance	
Fig. 70:	Stacked vias	
Fig. 71:	Desktop icon CircuitPro PL	
•		
Fig. 72:	Message System connection	
Fig. 73:	Dialog Signal light check	
Fig. 74:	User guidance step New	
Fig. 75:	Drop-down list material	. 65
Fig. 76:	Drop-down list system	. 65
Fig. 77:	List of templates	. 65
Fig. 78:	User guidance step Material	
Fig. 79:	Dialog Open	
-		
Fig. 80:	User guidance step Import	
Fig. 81:	Assigning holes to layer	
Fig. 82:	User guidance step Layout	
Fig. 83:	Context menu Step and repeat	. 70
Fig. 84:	Dialog Step and repeat	. 71
Fig. 85:	Dialog Step and repeat after entering values	
Fig. 86:	Multiplied layout	
Fig. 87:	Create fiducials automatically	
0		
Fig. 88:	Dialog Automatic fiducial creation	
Fig. 89:	Fiducials created	
Fig. 90:	Compute scan fields	
Fig. 91:	Dialog Compute scan fields	. 76
Fig. 92:	Scan fields computed	. 77
Fig. 93:	Computing toolpaths	
Fig. 94:	Message Toolpaths calculation	
Fig. 95:	Message Computation results	
•		
Fig. 96:	User guidance step <i>Processing</i>	
Fig. 97:	Dialog Processing phase: Mount material	
Fig. 98:	Dialog Placement	
Fig. 99:	Dialog Placement Processing data	. 80
Fig. 100:	Message Thickness measurement running	. 80
•	Message Laser warm-up phase in progress	
	Dialog Test tool settings	
	Message Processing phase: Flip material	
	Message Board production finished	
-	Desktop icon CircuitPro PM	
	Dialog Connection steps	
Fig. 107:	Dialog New document	. 84
Fig. 108:	Dialog Open	. 85
	Dialog Import Assigned layers	

Fig. 110:	Imported data in the CAM view	86
Fig. 111:	CAM view after selecting holes	87
Fig. 112:	Context menu Assign objects to DrillUnplated layer	87
	Context menu Step & Repeat	
	Dialog Step & Repeat	
	Dialog Step & Repeat after entering values	
	CAM view of the multiplied layout	
	Dialog Create fiducial	
	CAM view after creating fiducials	
	Tab Isolate	
	Tab Contour routing	
	Tab Pockets	
0	Message Computation Results	
	Open project	
	Dialog Upgrade project	
	Upgrade selected	
	Material selection	
0	Opened project in the user guidance step <i>Toolpaths</i>	
	Dialog Computing toolpaths	
•	Message Processing phase: Process drilling on a ProtoMat	
	Message Processing phase: Galvanic through-hole plating	
•	Message Processing phase: Mount material bottom	
	Dialog Copper thickness	
	Dialog Placement	
	Message Processing phase: Flip material	
	Dialog Open CAM view of the opened project	
	Machining view of the opened project	
	Drop-down list of the processing phases	
	Machining view of drilling and contour-routing data	
	Upgrade selected	
•	List of templates	
	Material selection.	
	User guidance step Import	
	User guidance step <i>Layout</i>	
	Dialog Compute scan fields	
	Scan fields computed Overlap displayed	
	Computing toolpaths	
	Flexible material fastened to the processing table	
	Dialog Placement	
Fig. 150:	Message Processing phase: Galvanic through-hole plating	117
Fig. 151:	PCB holder for flexible PCBs	117
Fig. 152:	Inserting the flexible PCB into tank 3	118
	Pane Workflow setup	
	Context menu Process from here	
-	Dialog Copper thickness	
-	Dialog Test tool settings	
	List of multi-layer templates	
	Material selection	
	User guidance step Import	
	Layout of a multi-layer PCB	
	Context menu Move object	
	Specify reference point	
	Specify distance	
	Desired position of the copied layout	
	Layout moved to the desired position	
	Opened project	

		404
	User guidance step <i>Processing</i>	
	Message Processing phase: Mount laminate	
	Dialog Placement	
Fig. 171:	Message Processing phase: Mount core	133
Fig. 172:	Message Processing phase: Press all layers	134
Fig. 173:	Pane Workflow setup	135
	Context menu Process from here	
	Message Processing phase: Mount material top	
	Dialog Placement	
-	Message Processing phase: Galvanic through-hole plating	
	Dialog Copper thickness	
Fig. 179:	Dialog Import Assigned layers	141
Fig. 180:	CAM view multi-layer PCB	141
Fig. 181:	Upgrade selected	144
	Material selection	
-	Opened project	
•	Dialog Computing toolpaths	
•	User guidance step <i>Processing</i>	
•	Message Processing phase: Custom instruction	
-	Message Processing phase: Mount core	
	Dialog Placement	
Fig. 189:	Message Processing phase: Press all layers	148
	Context menu Process from here	
Fig. 191:	Dialog Copper thickness	150
	Message Processing phase: Process drilling on a ProtoMat	
	Dialog <i>Import</i> Assigned layers	
	CAM view multi-layer PCB	
•		
•	Upgrade selected	
0	Material selection	
	User guidance step <i>Processing</i>	
	Message Processing phase: Custom instruction	
	Message Processing phase: Custom instruction Message Processing phase: Mount core	
Fig. 199:	Message Processing phase: Mount core	159
Fig. 199: Fig. 200:	Message Processing phase: Mount core Dialog Placement	159 159
Fig. 199: Fig. 200: Fig. 201:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers	159 159 160
Fig. 199: Fig. 200: Fig. 201: Fig. 202:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here	159 159 160 161
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers	159 159 160 161 166
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 203:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias	159 159 160 161 166 167
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected	159 159 160 161 166 167 169
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection	159 159 160 161 166 167 169 170
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 205: Fig. 206: Fig. 207:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project	159 159 160 161 166 167 169 170 170
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 205: Fig. 206: Fig. 207: Fig. 208:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths	159 159 160 161 166 167 169 170 170 170
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 205: Fig. 206: Fig. 207: Fig. 208:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths	159 159 160 161 166 167 169 170 170 170
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing	159 159 160 161 166 167 169 170 170 170
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 210:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction	159 160 161 166 167 169 170 170 170 171
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 208: Fig. 208: Fig. 209: Fig. 210: Fig. 211:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness	159 160 161 166 167 169 170 170 170 171 171
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 205: Fig. 206: Fig. 207: Fig. 208: Fig. 209: Fig. 210: Fig. 211: Fig. 212:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement	159 160 161 166 167 169 170 170 170 171 171 171
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 205: Fig. 206: Fig. 207: Fig. 208: Fig. 209: Fig. 210: Fig. 211: Fig. 211: Fig. 212: Fig. 213:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers	159 159 160 161 166 167 170 170 170 171 171 171 171 172 173
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 206: Fig. 207: Fig. 208: Fig. 209: Fig. 210: Fig. 211: Fig. 211: Fig. 212: Fig. 213: Fig. 214:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold	159 160 161 166 167 169 170 170 170 171 171 171 172 173 174
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 210: Fig. 211: Fig. 211: Fig. 212: Fig. 213: Fig. 214: Fig. 215:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material	159 159 160 161 166 167 170 170 170 171 171 171 172 173 174 174
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 210: Fig. 211: Fig. 213: Fig. 214: Fig. 215: Fig. 216:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet	159 159 160 161 166 167 170 170 170 170 171 171 172 173 174 174 175
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 210: Fig. 211: Fig. 213: Fig. 214: Fig. 215: Fig. 216: Fig. 217:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing. Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold. Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil	159 159 160 161 166 167 170 170 170 170 171 171 172 173 174 174 175 175
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 210: Fig. 211: Fig. 213: Fig. 214: Fig. 215: Fig. 216: Fig. 217:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet	159 159 160 161 166 167 170 170 170 170 171 171 172 173 174 174 175 175
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 206: Fig. 207: Fig. 208: Fig. 209: Fig. 210: Fig. 211: Fig. 212: Fig. 213: Fig. 214: Fig. 215: Fig. 216: Fig. 217: Fig. 218:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing. Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold. Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil	159 160 161 166 167 169 170 170 170 171 171 171 171 172 173 174 175 175
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 206: Fig. 207: Fig. 208: Fig. 209: Fig. 210: Fig. 211: Fig. 211: Fig. 211: Fig. 213: Fig. 214: Fig. 215: Fig. 216: Fig. 217: Fig. 218: Fig. 219:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil Core material on the dowel pins	159 160 161 166 167 169 170 170 170 171 171 171 171 172 173 174 175 176 176
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 206: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 210: Fig. 211: Fig. 211: Fig. 212: Fig. 213: Fig. 214: Fig. 215: Fig. 216: Fig. 217: Fig. 218: Fig. 219: Fig. 220:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil Core material on the core material Copper foil on the prepreg material	159 160 161 166 167 169 170 170 170 171 171 171 172 173 174 175 176 176 177
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 210: Fig. 210: Fig. 211: Fig. 212: Fig. 213: Fig. 214: Fig. 215: Fig. 216: Fig. 216: Fig. 217: Fig. 218: Fig. 219: Fig. 219: Fig. 220: Fig. 220: Fig. 221:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil Core material on the core material Copper foil on the prepreg material Prepreg material on the core material Prepreg material on the core material Prepreg material on the core material Propreg foil on the prepreg material Pane Workflow setup	159 160 161 166 167 170 170 170 170 171 171 171 172 173 174 175 175 176 177 179
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 209: Fig. 210: Fig. 211: Fig. 213: Fig. 213: Fig. 214: Fig. 215: Fig. 216: Fig. 217: Fig. 217: Fig. 218: Fig. 219: Fig. 219: Fig. 220: Fig. 221: Fig. 222:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil Core material on the core material Copper foil on the prepreg material Pane Workflow setup Context menu Process from here	159 160 161 166 167 170 170 170 170 171 171 171 172 173 174 175 176 177 179 180
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 210: Fig. 210: Fig. 211: Fig. 213: Fig. 214: Fig. 214: Fig. 215: Fig. 216: Fig. 217: Fig. 218: Fig. 219: Fig. 220: Fig. 220: Fig. 220: Fig. 220: Fig. 221: Fig. 222: Fig. 223:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing. Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil Core material on the core material Copper foil on the prepreg material Pane Workflow setup Context menu Process from here Dialog Test tool settings	159 159 160 161 166 167 170 170 170 170 170 177 173 174 175 176 177 179 180 181
Fig. 199: Fig. 200: Fig. 201: Fig. 202: Fig. 203: Fig. 204: Fig. 205: Fig. 206: Fig. 207: Fig. 207: Fig. 208: Fig. 209: Fig. 209: Fig. 209: Fig. 210: Fig. 210: Fig. 211: Fig. 213: Fig. 214: Fig. 214: Fig. 215: Fig. 216: Fig. 217: Fig. 218: Fig. 219: Fig. 220: Fig. 220: Fig. 221: Fig. 222: Fig. 222: Fig. 223: Fig. 224:	Message Processing phase: Mount core Dialog Placement Message Processing phase: Press all layers Context menu Process from here Dialog Import Assigned layers CAM view multi-layer PCB with blind vias and buried vias Upgrade selected Material selection Opened project Dialog Computing toolpaths User guidance step Processing Message Processing phase: Custom instruction Dialog Copper thickness Dialog Placement Message Processing phase: Press all layers Assembly of the materials in the press mold Positioning markers core material Copper foil on the lower steel press sheet Prepreg material on the copper foil Core material on the core material Copper foil on the prepreg material Pane Workflow setup Context menu Process from here	159 159 160 161 166 167 170 170 170 171 171 171 172 173 174 175 176 177 179 180 181 182

Fig. 226:	Dialog Copper thickness	184
Fig. 227:	Message Processing phase: Process drilling on a ProtoMat	186
	List of single-sided templates	
Fig. 229:	Material selection	189
Fig. 230:	User guidance step Import	190
Fig. 231:	Layout of a polyimide stencil	190
Fig. 232:	User guidance step Processing	191
	User guidance step Material	
	Copy of selected material	
Fig. 235:	View Modify parameters	195
	Entering the material name	
	Deleting the materials	
	Materials deleted	
	Adding the material	
	Material added to the new set of materials	
	Selecting the material	
	User guidance step Import	
	Assigning board outline to layer BoardOutline	
	Deselecting the source BoardOutline	
	Assigning the holes to layer DrillPlated	
	Message Info	
	User guidance step <i>Layout</i>	
	Highlighted layout	
	Layout moved	
•	Context menu PCB layer > Prepreg 1	
	Context menu Display mode > True Width	
	Context menu Technology > Cutting	
	Menu item Add new layer	
	Context menu item <i>Rename</i>	
•	Renaming layer	
•	Input fields Specify center point	
	Input fields Specify diameter	
	Specifying the next center point	
	Flexible material positioning holes	
	Context menu Copy to layer	
•	Input fields Non-uniform scaling (in percent)	
	Values for Non-uniform scaling (in percent)	
	Preview of the scaled positioning holes	
	Prepreg positioning holes	
-	Rename	
	Work package selected	
-	Modified settings	
	Work package selected	
	Settings of the work package Cut Reference Holes	
	Channel width modified	
0	Work package deselected	
	Work package selected	
-	Modified settings	
	Phase and work package selected	
-	Modified settings	
	User guidance step <i>Processing</i>	
-	Context menu Process from here	
•	Dialog Test tool settings	
-	Dialog Copper thickness	
-	Dialog Test tool settings	
	User guidance step <i>Layout</i>	
	Material selection	
	Specify target point	
· ·g. 200.		

		~~-
	Pasted layout	
Fig. 285:	Material selection	230
Fig. 286:	User guidance step Import	231
	Layout of an RF PCB	
	User guidance step Workflow	
•		
-	Work package Structure Top selected	
•	Group Parameters	
Fig. 291:	Sub-group Hatching	233
Fig. 292:	Tool selected	234
•	Tool moved	
0	User guidance step <i>Processing</i>	
	Select material	
•	Select CutInside	
Fig. 297:	Imported data	239
Fig. 298:	Layout selected	240
•	Combine open paths	
•	User guidance step Workflow	
•		
	Selecting the work package Cut Inside	
•	Group Parameters expanded	
Fig. 303:	Entry Center selected	242
Fig. 304:	Selecting the material	245
	User guidance step Import	
-	Deselected files	
•		
	Format undefined	
	Selecting the appropriate file format	
Fig. 309:	Appropriate file format selected	249
Fig. 310:	Unassigned layer	250
	Selecting the target layer	
	Selected target layer	
	User guidance step <i>Layout</i>	
	Unassigned layer	
	Info about new layer created	
Fig. 316:	User guidance step Layout processing data	253
Fig. 317:	List of layers Select by layer	253
	Assigning objects to target layer	
	Assigned holes	
	Large size of the layout	
	Selecting the appropriate unit	
Fig. 322:	Changed dimensions of the layout	258
Fig. 323:	Incorrect preview of the layout	259
	Correct preview of the layout	
	File Tutor.BOA selected	
-	Tab Apertures/Tools and sub-tab Attributes	
-	Selecting the type of aperture	
Fig. 328:	Preview of the modified aperture	261
Fig. 329:	Layer name in the file Tutor.BOA	262
	Layer names not used in the Gerber file	
•	Gerber file selected	
•		
	Activated check boxes	
	Layer names defined in Gerber files displayed	
	Dialog Options Gerber	
Fig. 335:	Selecting value True for Use layer name	265
	Layer names defined in Gerber files displayed	
5		-

6.2 List of tables

Table 1:	Tips for positioning and fiducial recognition	11
Table 2:	Functions in the dialog Material thickness measurement settings	21
Table 3:	Required files/layers	57
Table 4:	Size range according to hole type	58
Table 5:	Required LPKF systems	63
Table 6:	Required LPKF systems	124

