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

Phone: +386 592 08 800
Fax: +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:

⚠ DANGER
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.

⚠ WARNING
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.

⚠ CAUTION
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

Introduction 8

1 Basics 10

1.1 Project placement 10
1.2 Recognizing fiducials 15
1.3 Measuring the material thickness 20

1.3.1 Material thickness measurement settings 20
1.3.2 Measurement strategies 22
1.3.3 Measuring the material thickness automatically 23
1.3.4 Measuring the material thickness manually (at a single point) 25
1.3.5 Referencing the material thickness measurement sensor 27

1.4 Working with mixed PCB layouts in CircuitPro PL 29
1.5 Testing and tuning the tool settings 39
1.6 Processing multi layers 48
1.7 Processing methods of the ProtoLaser 51

1.7.1 Delamination method 52
1.7.2 Hatching method 54

1.8 Design guidelines for the production of multi-layer PCBs 56

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 58
1.8.4 Annular rings 58
1.8.5 Inner layers fill 59
1.8.6 Staggered vias 60
1.8.7 Stacked vias 61

2 Producing double-sided PCBs 63

2.1 How to produce a double-sided PCB without through-hole plating 63
2.2 How to produce a double-sided PCB with galvanic through-hole plating 83
2.3 How to produce a double-sided PCB with non-galvanic through-hole plating 104
2.4 Producing a double-sided flexible PCB with galvanic through-hole plating 110

3 Producing multi-layer PCBs 124

3.1 How to produce a multi-layer PCB with galvanic through-hole plating by using a laser system only 124
3.2 How to produce a multi-layer PCB with galvanic through-hole plating 140
3.3 How to produce a multi-layer PCB with non-galvanic through-hole plating 153
3.4 How to produce a multi-layer PCB with blind vias and buried vias 165

4 Other how-to examples 188

4.1 Producing a polyimide stencil 188
4.2 Producing a flex-rigid PCB 193
4.3 Structuring the solder mask with the laser system 224
4.4 Producing an RF PCB 230

5 Processing files 238

5.1 Processing DXF files in CircuitPro PL 238
5.2 Processing Gerber and Ecxellon files in CircuitPro PL 245

6 Appendix 266

6.1 List of figures 266
6.2 List of tables 272

Introduction

This document describes the most common applications for the production of double-sided 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.

Basics

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.

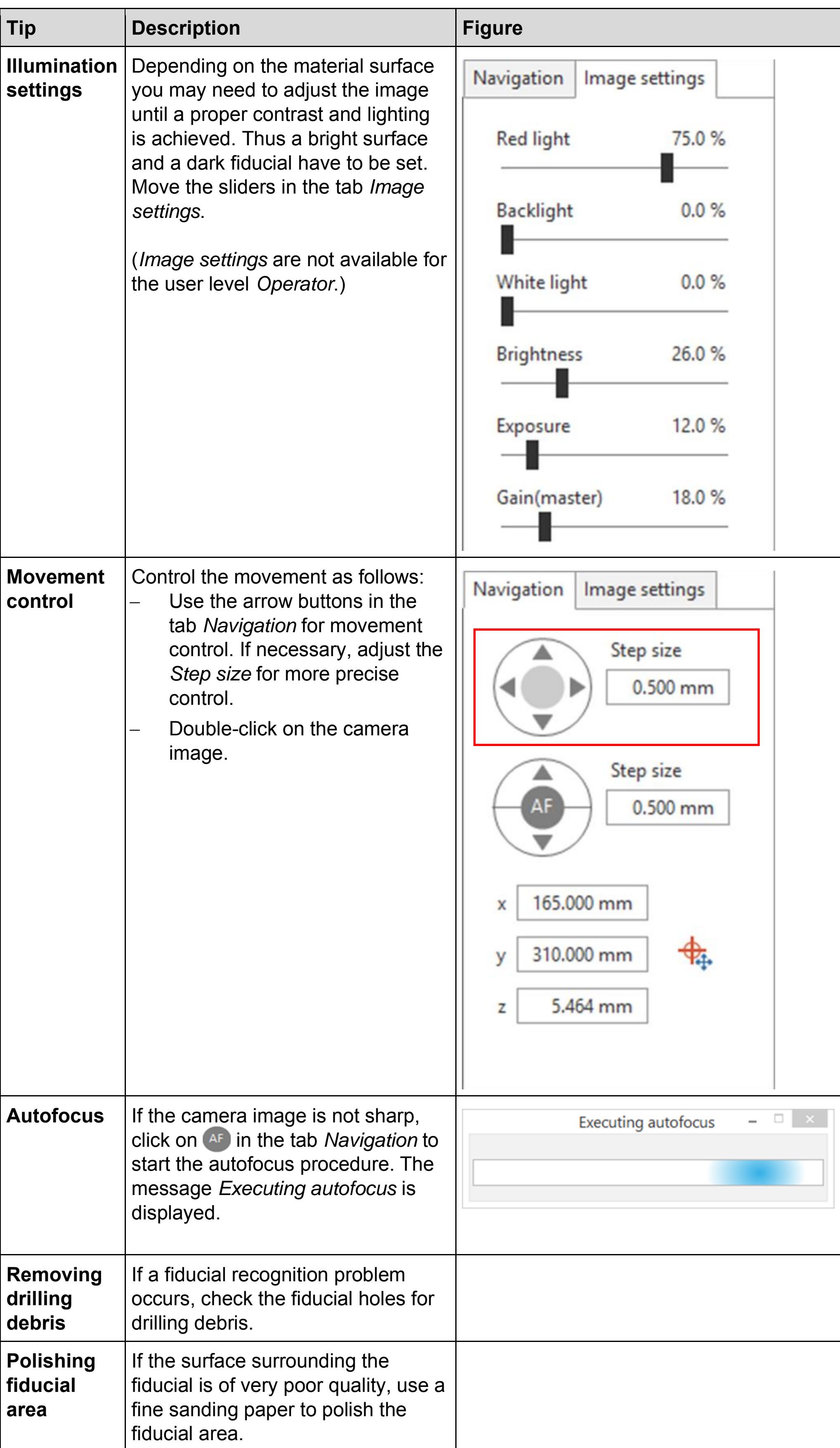
Ensure that the following prerequisites are fulfilled before performing the described tasks:

Prerequisites

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

The following table provides tips for positioning and fiducial recognition:

Tip	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 ⁎ to switch the pilot laser on/off.	
Zooming in	Zoom in the processing area by using the mouse wheel for more precise positioning.	



Tip	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 *Image settings*. (*Image settings* are not available for the user level *Operator*.)	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 *Navigation* for movement control. If necessary, adjust the *Step size* for more precise control. – Double-click on the camera image.	Navigation Image settings Step size 0.500 mm AF 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 AF in the tab *Navigation* to start the autofocus procedure. The message *Executing autofocus* 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.

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.

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.
7. Switch to the view *Camera*.

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

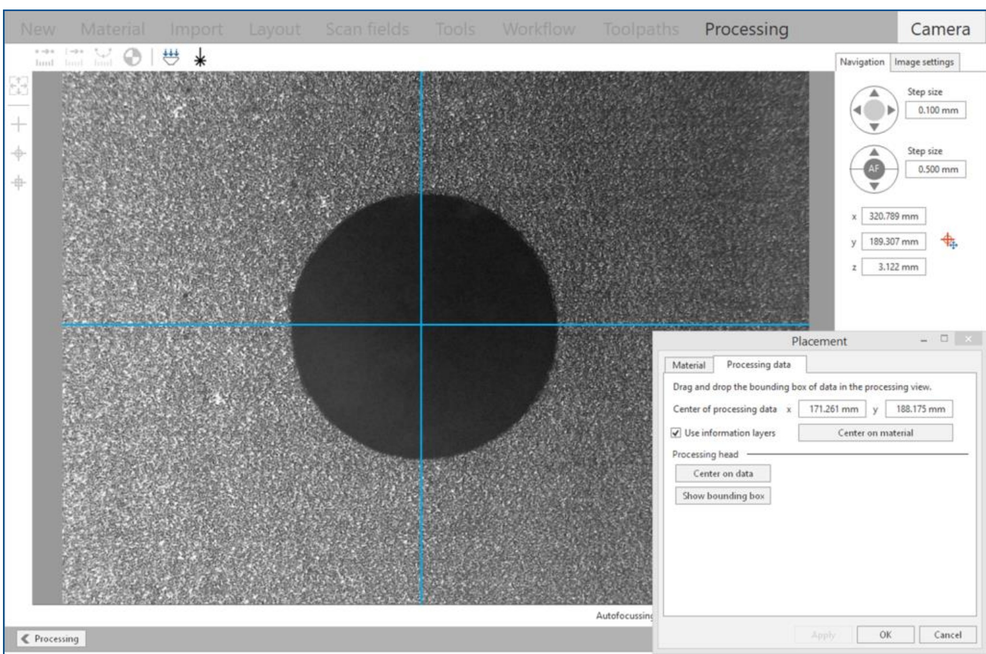


Fig. 1: Fiducial in the view *Camera*

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:

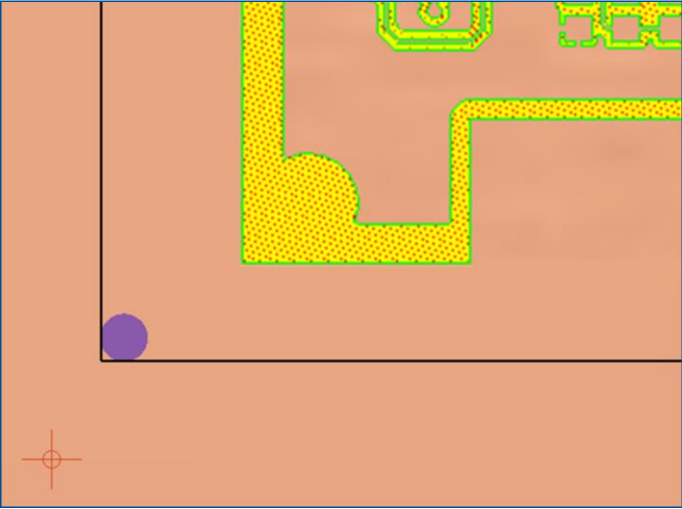


Fig. 2: Inaccurate position of the layout (example image)

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 (●) and the red crosshairs (⌖) overlap.

9. Move the mouse cursor over the layout.

☐ The mouse cursor turns into the hand symbol (☝).

10. Move the layout with Drag & Drop to match the location of the fiducial and the red crosshairs:

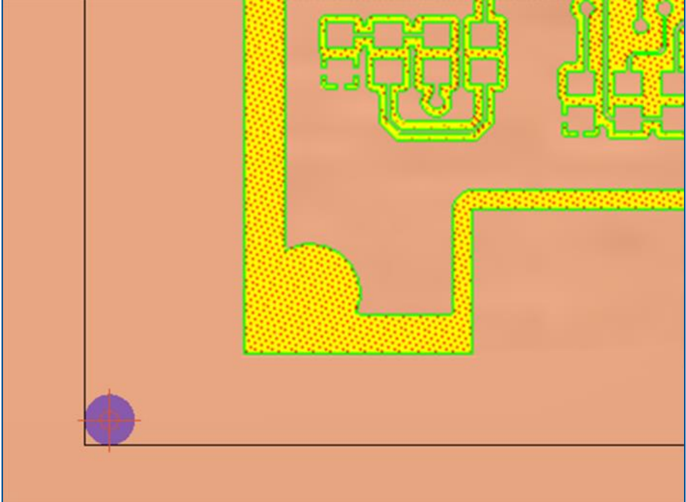


Fig. 3: Matching processing head and fiducial position

☐ 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.

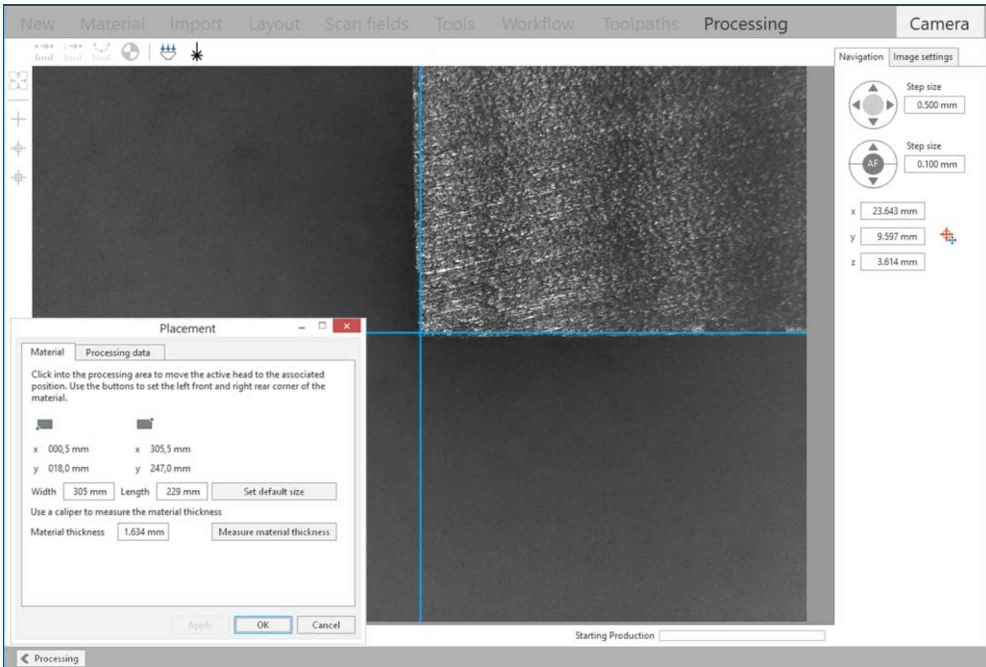


Fig. 4: Matching position of crosshair to base material edge

7. Confirm the position by clicking on [icon] 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 [icon] 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:

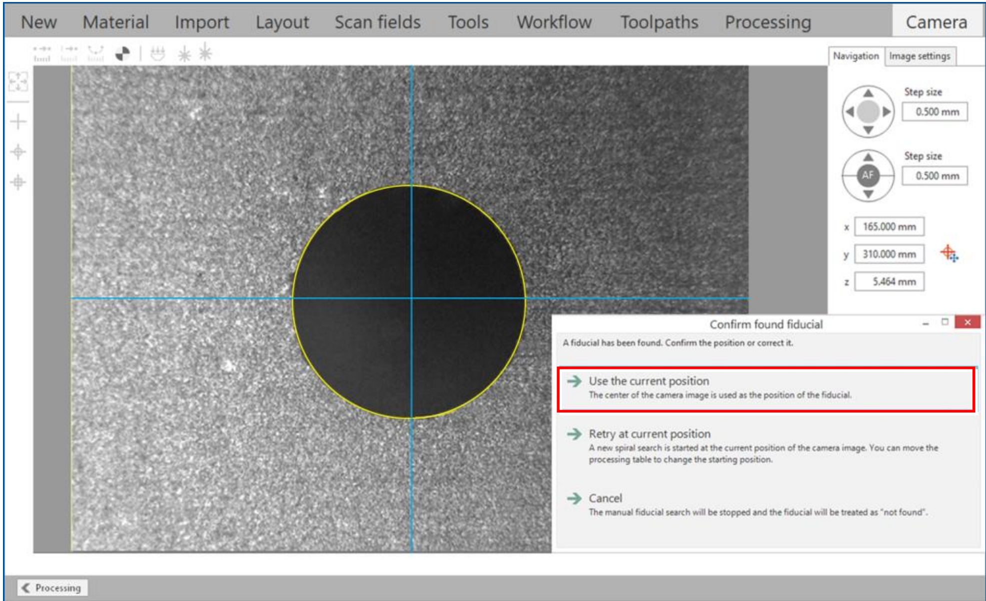


Fig. 5: Recognized fiducial

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:

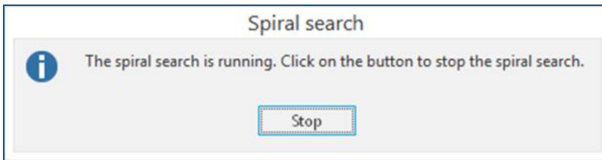


Fig. 6: Message *Spiral search active*

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:

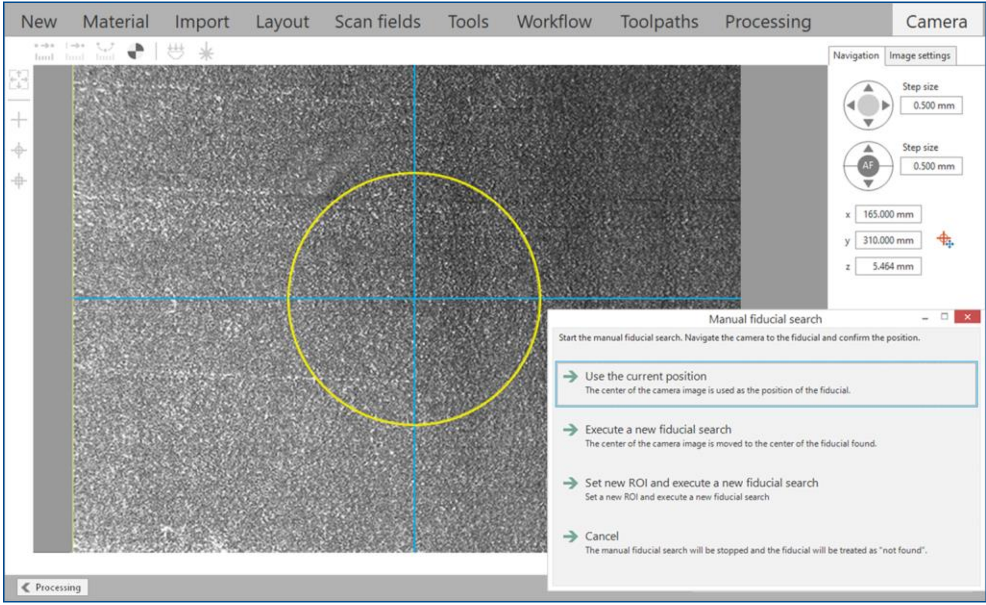


Fig. 7: ***Manual fiducial search***

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:

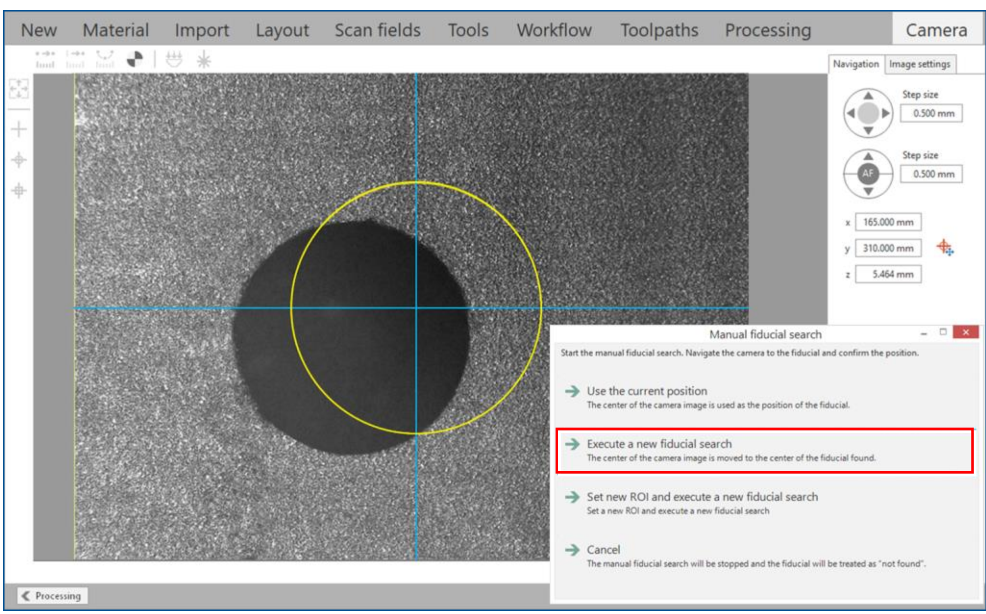


Fig. 8: **Visible fiducial**

4. Click on *Execute a new fiducial search*.

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

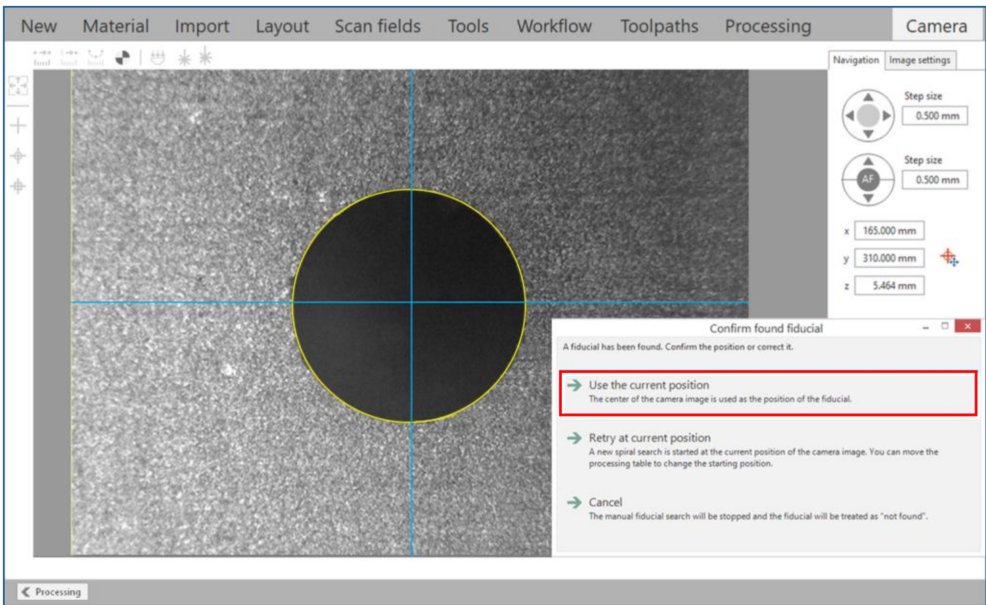


Fig. 9: Recognized fiducial

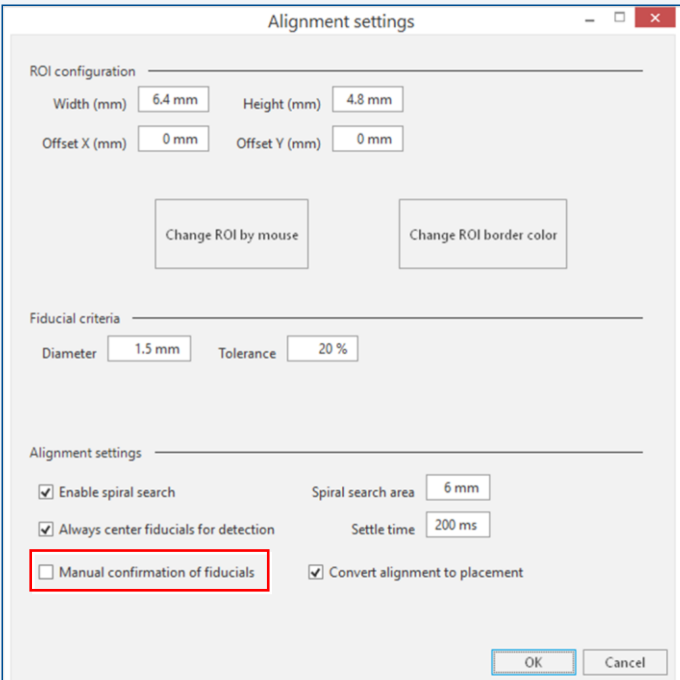
5. Click on *Use the current position*.

- 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.

Disabling manual confirmation of fiducials

1. Click on *Processing > Alignment settings...*
2. Deactivate the check box *Manual confirmation of fiducials*.
3. Click on [OK].



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:

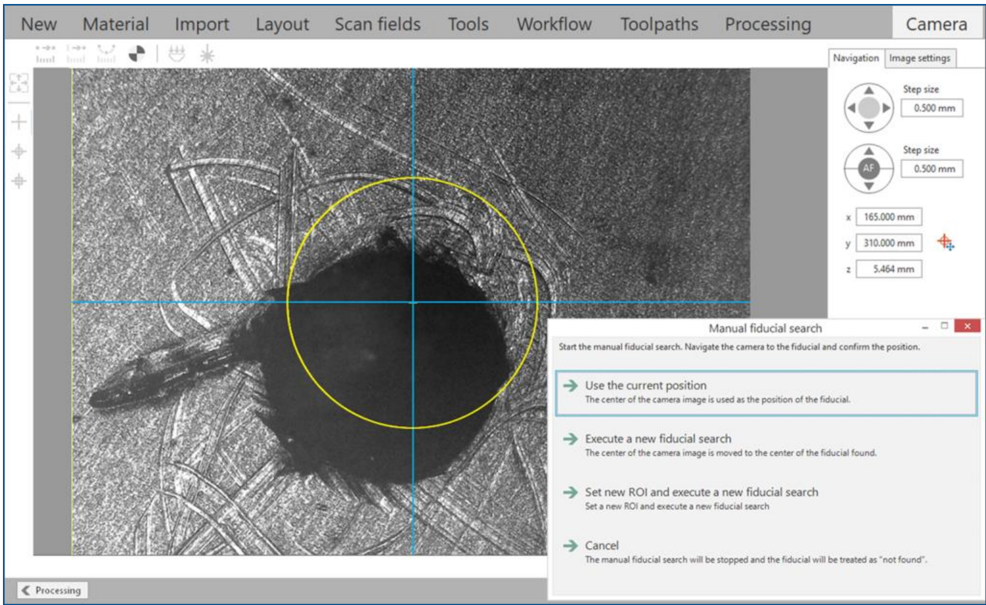


Fig. 10: Fiducial of poor quality

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

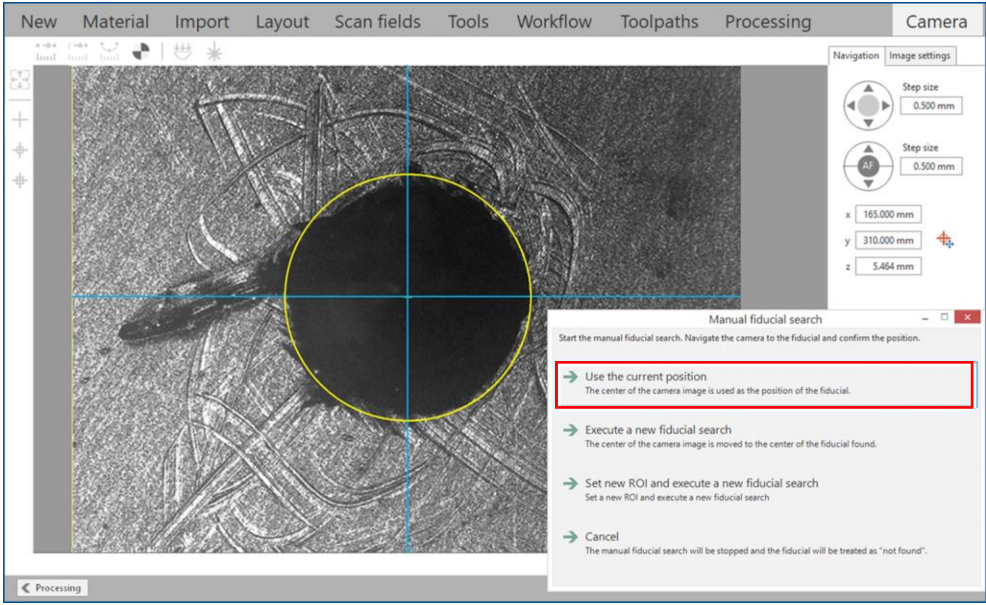


Fig. 11: Matched position of poor quality fiducial

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*.

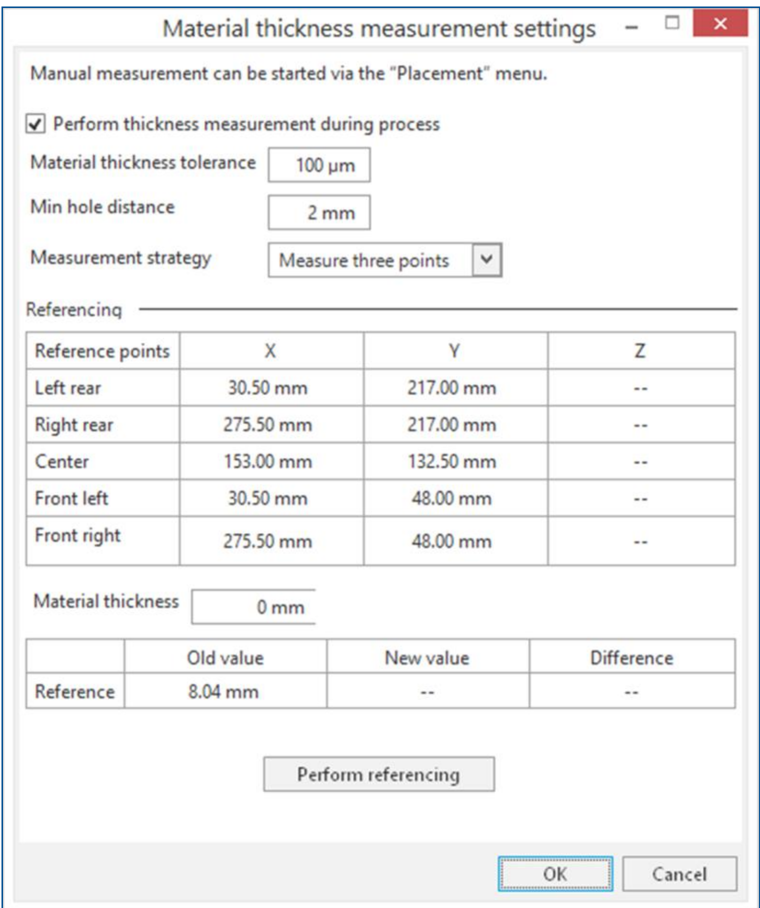


Fig. 12: Dialog *Material thickness measurement settings* (default settings)

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 (*Old value* and *New value*). The *Difference* 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:

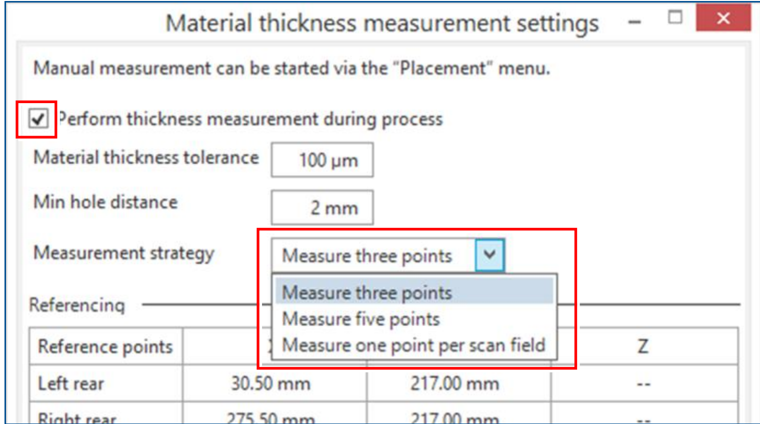


Fig. 13: Drop-down list *Measurement strategy*

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].

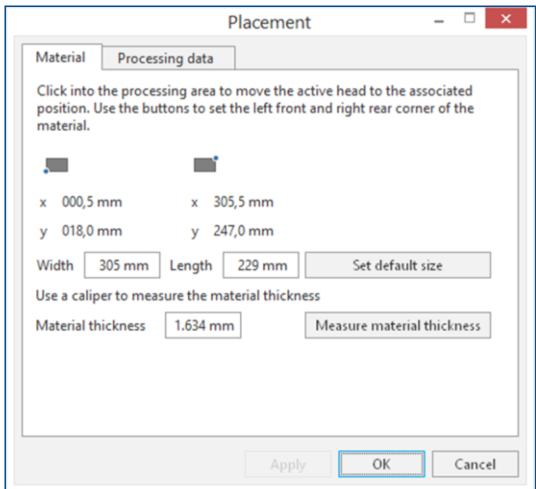


Fig. 14: Dialog *Placement*

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

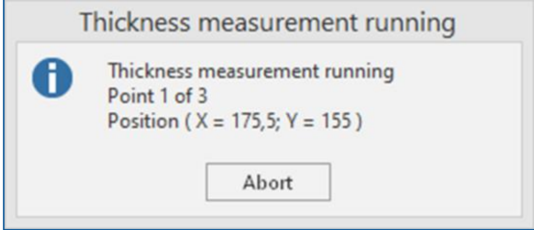


Fig. 15: Message *Thickness measurement running*

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:

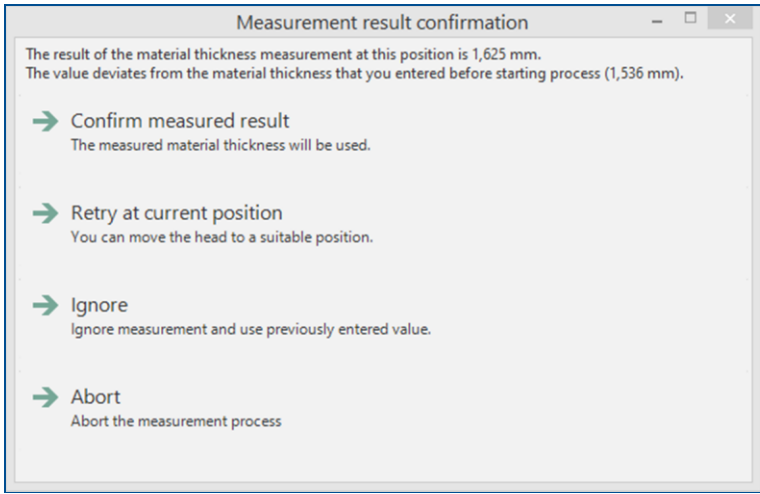


Fig. 16: Dialog *Measurement result confirmation*

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:

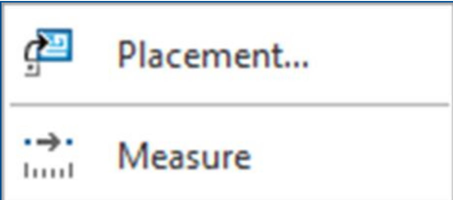


Fig. 17: Context menu

2. Select the menu item *Placement…*

☐ The following dialog is displayed:

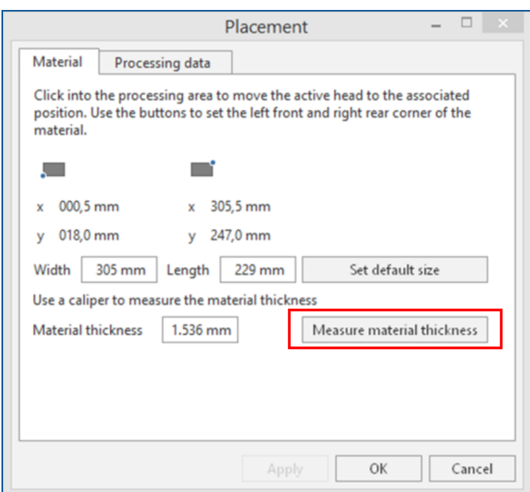


Fig. 18: Dialog *Placement*

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:

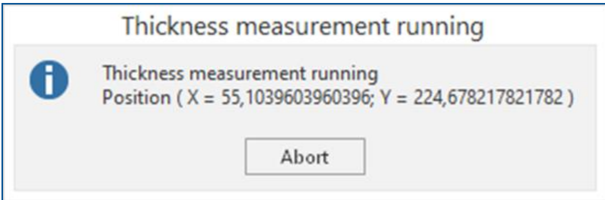


Fig. 19: Message *Thickness measurement running*

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

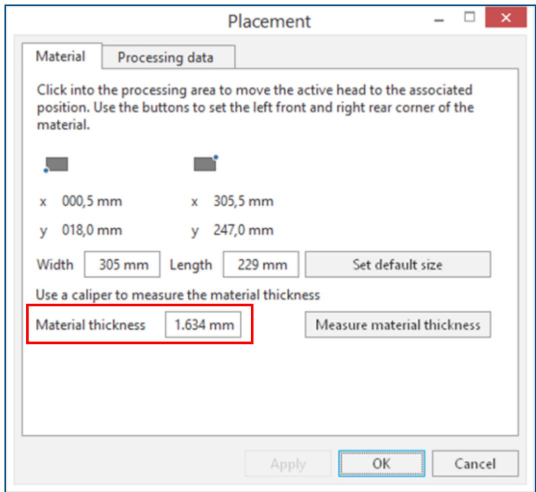


Fig. 20: Measured value

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

- The user guidance step *Processing* is active.

Auxiliaries

- 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*).

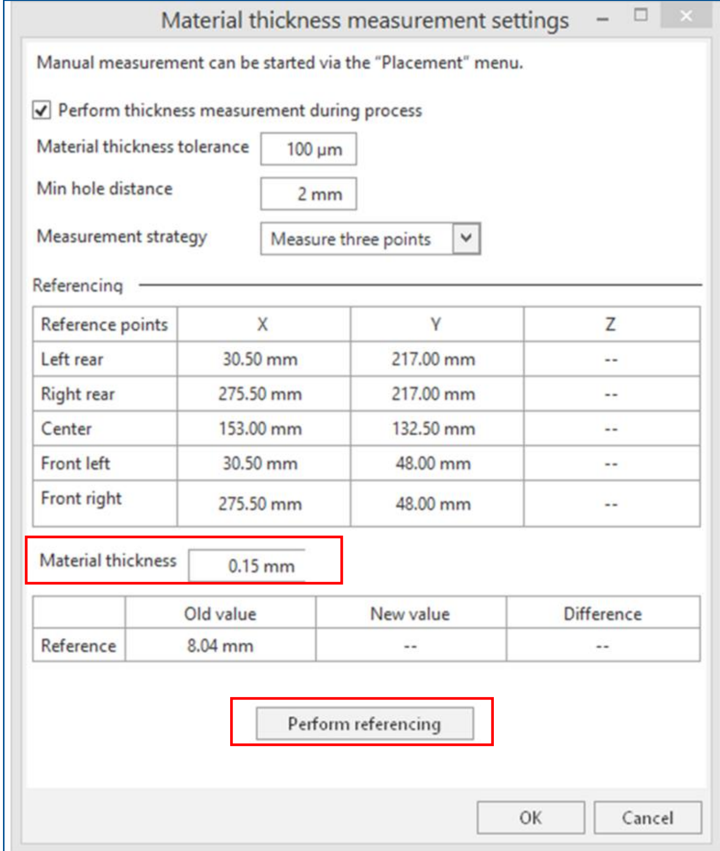


Fig. 21: Entering *Material thickness*

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*.

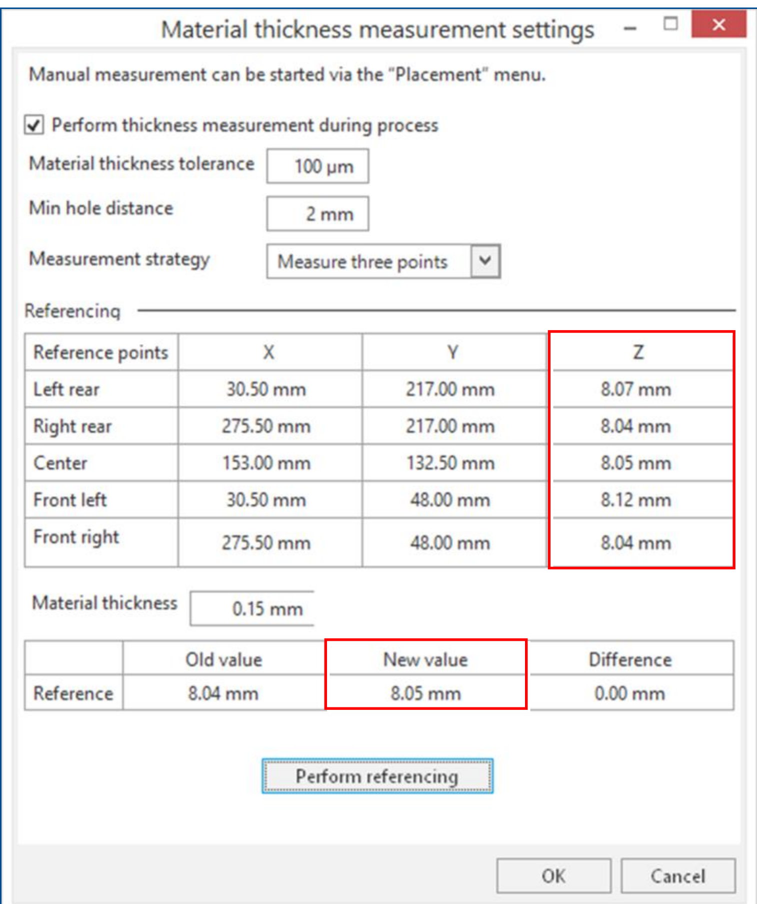


Fig. 22: Measured values and average value of the reference points

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.

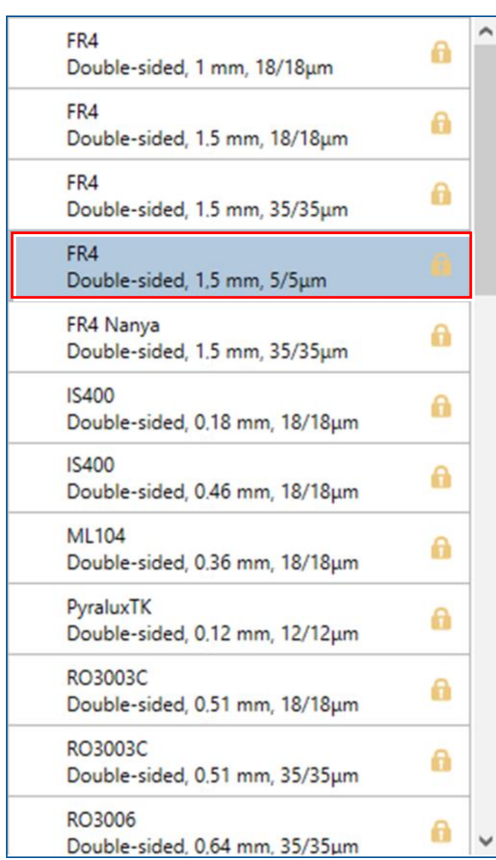


Fig. 23: Select material

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*.

You can add the fiducials to each individual PCB layout or add them after you have imported the individual PCB layouts into a single project. When you add the fiducials to each individual layout a more precise positioning is achieved.
In this example the fiducials are added to each individual PCB 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.

12. Click on [Compute toolpaths] or on .

- 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 .
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.

1. 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.

3. 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 µm* is used.
4. Click on [Select material] or double-click on the material.

☐ The user guidance step *Import* is displayed.

5. Click on .
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.

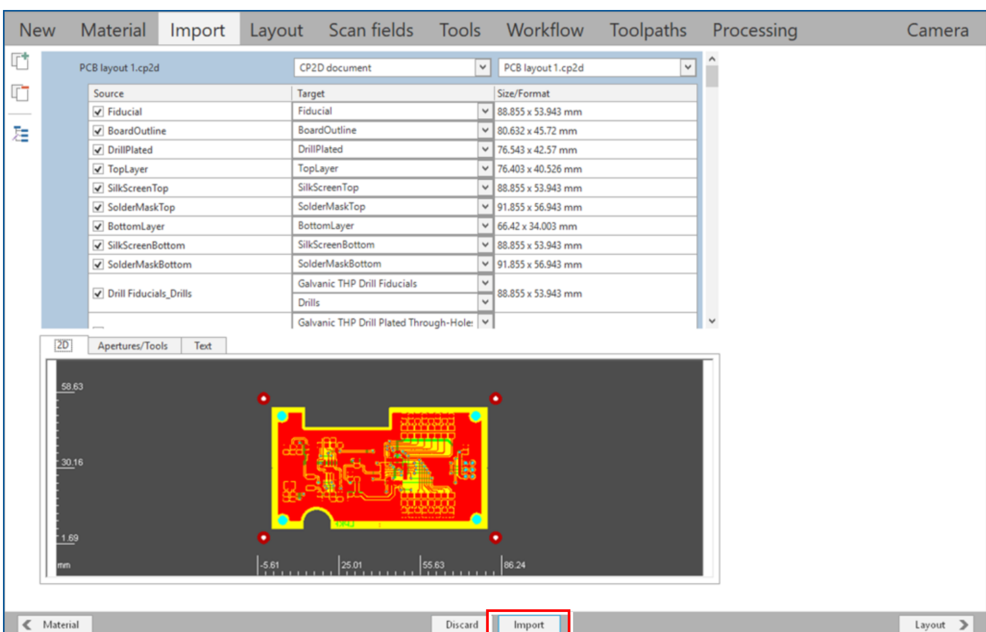


Fig. 24: **User guidance step *Import***

9. Click on [Import].

☐ The imported PCB layouts are displayed in the user guidance step *Layout*:

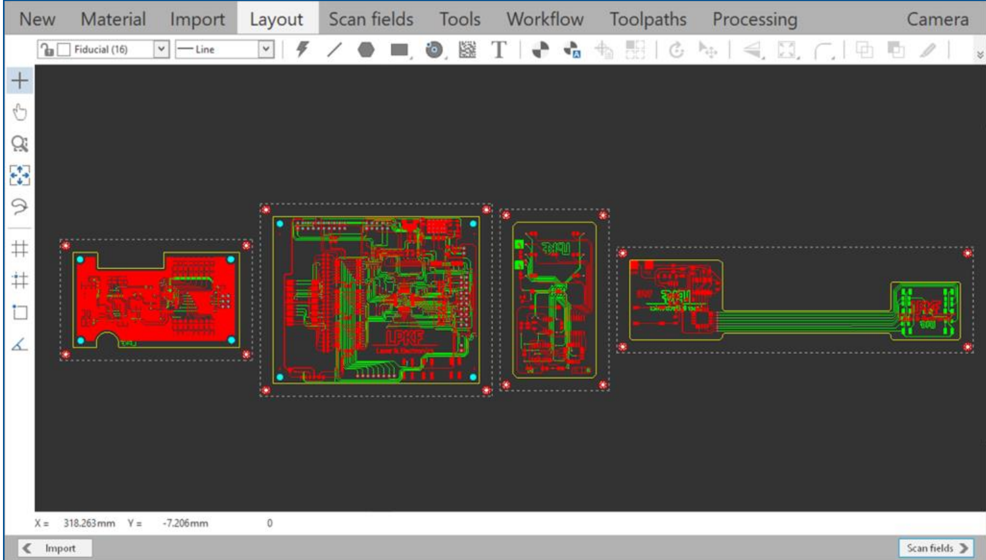


Fig. 25: Imported PCB layouts

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.

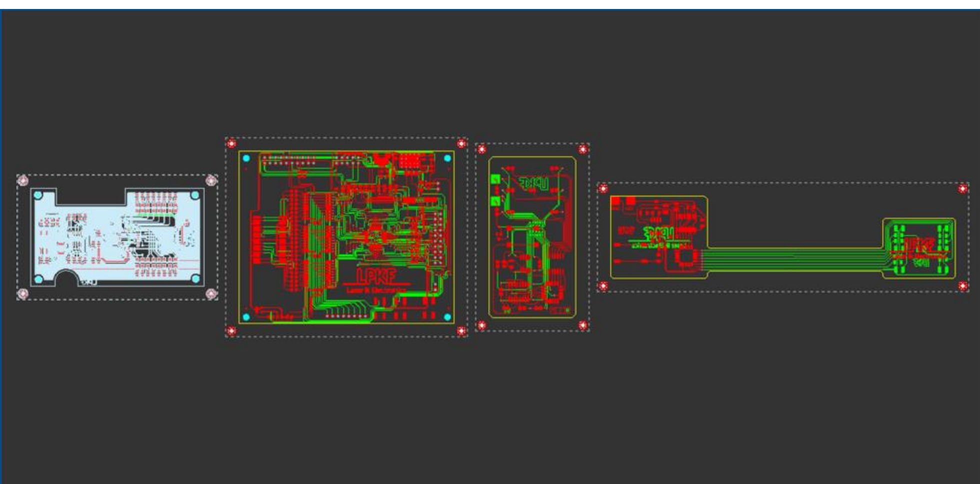


Fig. 26: One PCB layout selected

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

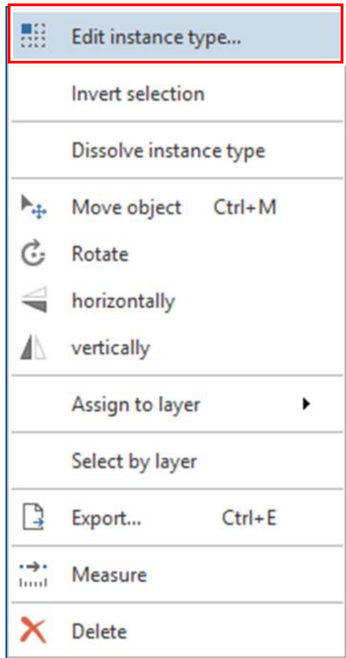


Fig. 27: Menu *Edit instance type*

- The following dialog is displayed:

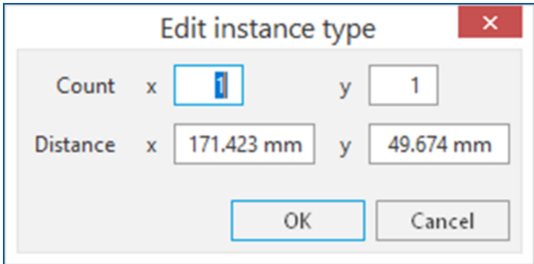


Fig. 28: Dialog *Edit instance type*

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:

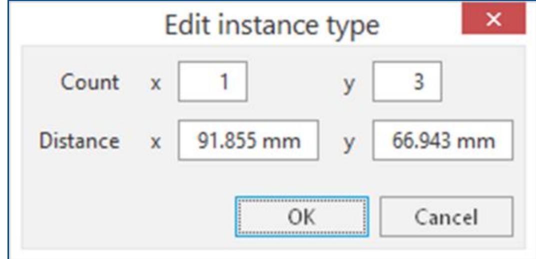


Fig. 29: Dialog *Edit instance type* | Values

14. Click on [OK].

☐ The selected layout is multiplied:

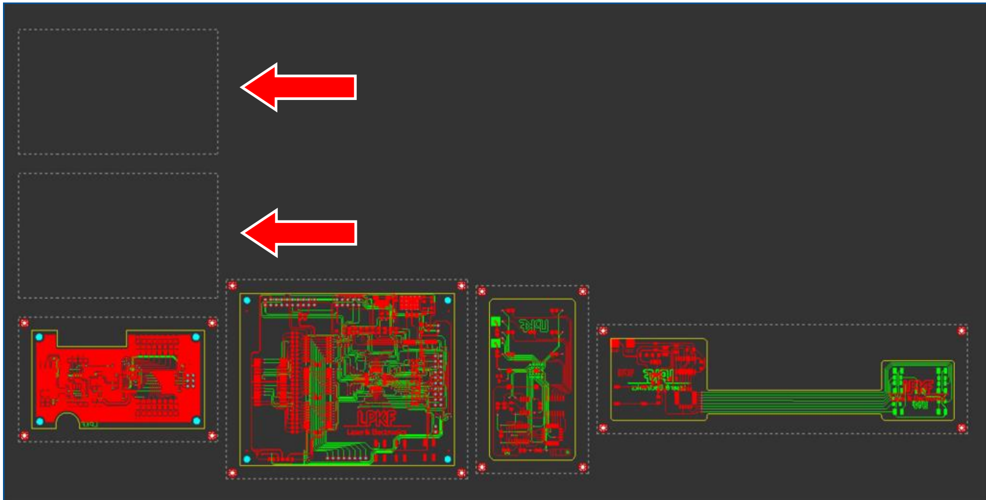


Fig. 30: Layout multiplied

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:

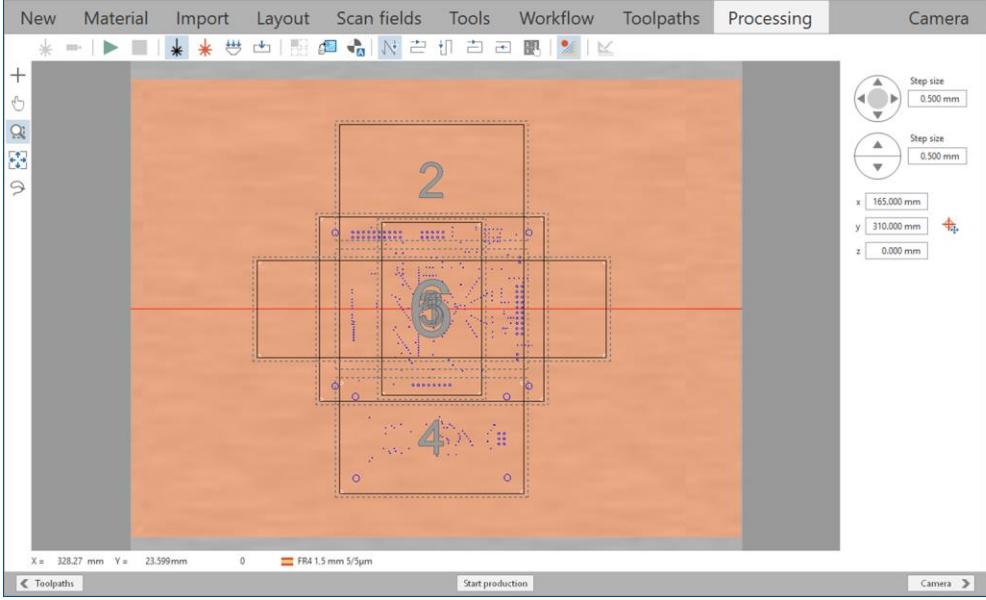


Fig. 31: User guidance step *Processing*

17. Click on [icon] to arrange the PCB layouts automatically.

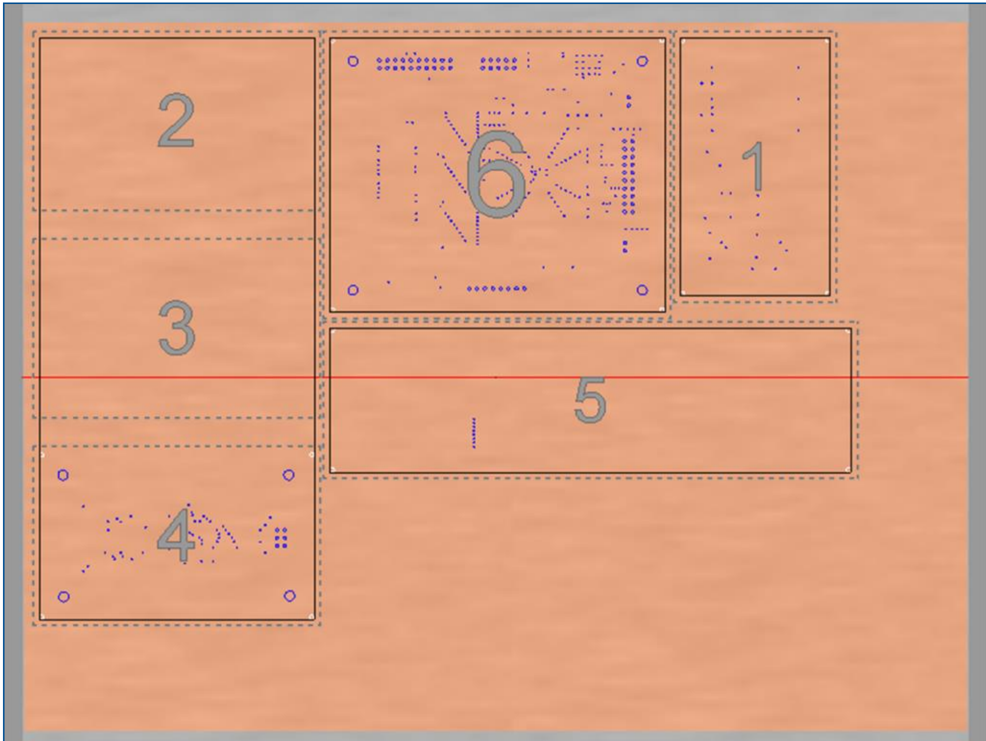


Fig. 32: Automatic arrangement of PCB layouts

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

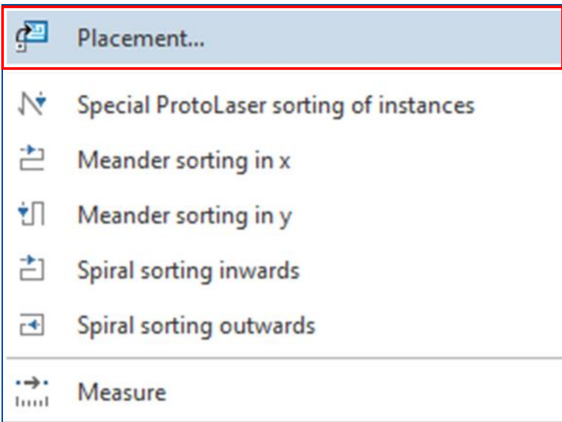


Fig. 33: Menu item *Placement*

☐ The dialog *Placement* is displayed.

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

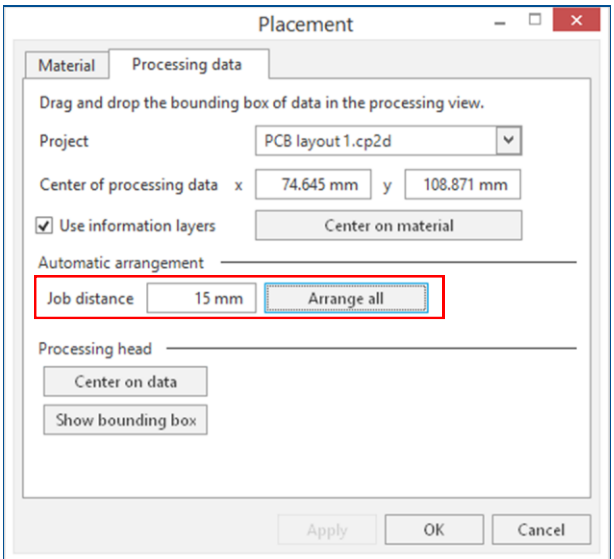


Fig. 34: Dialog *Placement* | *Processing data*

22. Click on [Arrange all].
23. Click on [OK].

☐ The distance between individual PCB layouts is increased.

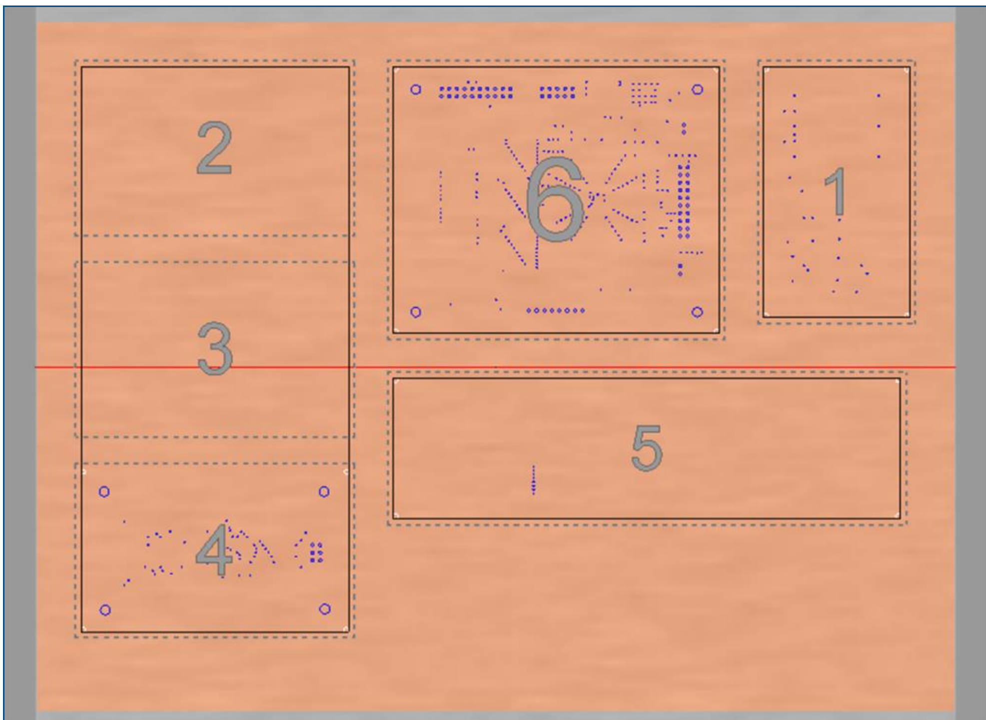
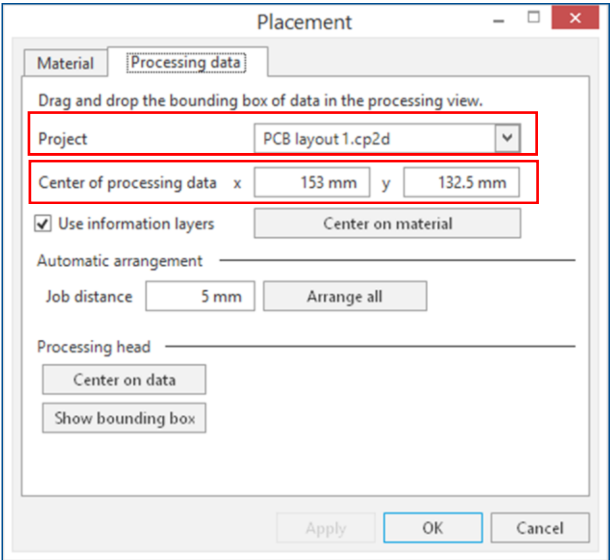


Fig. 35: Distance between PCB layouts increased

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*.



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

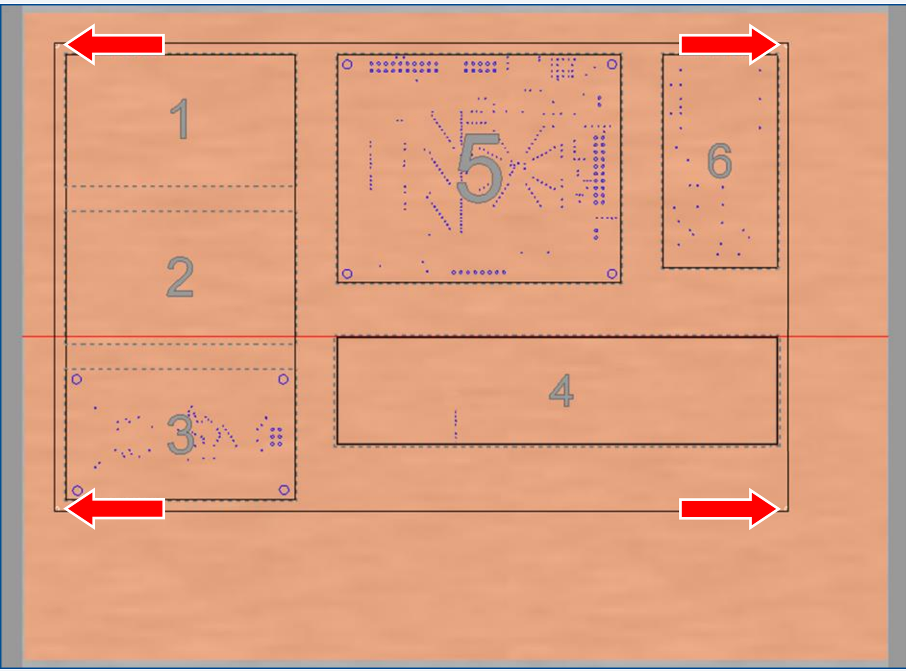
If you have not created fiducials in the individual PCB layouts, you can create them after importing the .cp2d files into a single project.

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

- The user guidance step *Processing* is active.
- The dialog *Test tool settings* is displayed.

Spare parts and auxiliaries

- Portable hand-held microscope

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)

Placing and processing the test sample

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

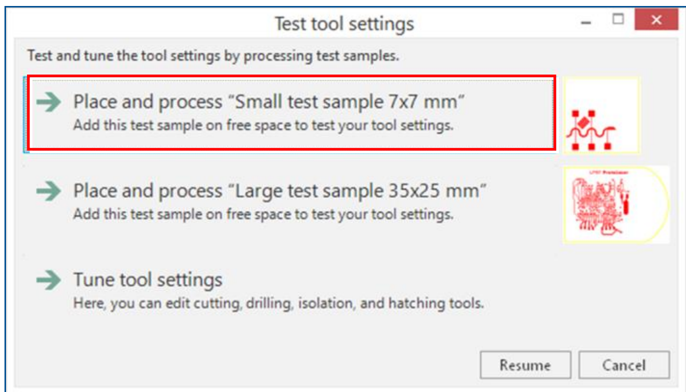


Fig. 36: ***Place and process "Small test sample 7x7 mm"***

☐ The input fields for specifying the position of the test sample are displayed:

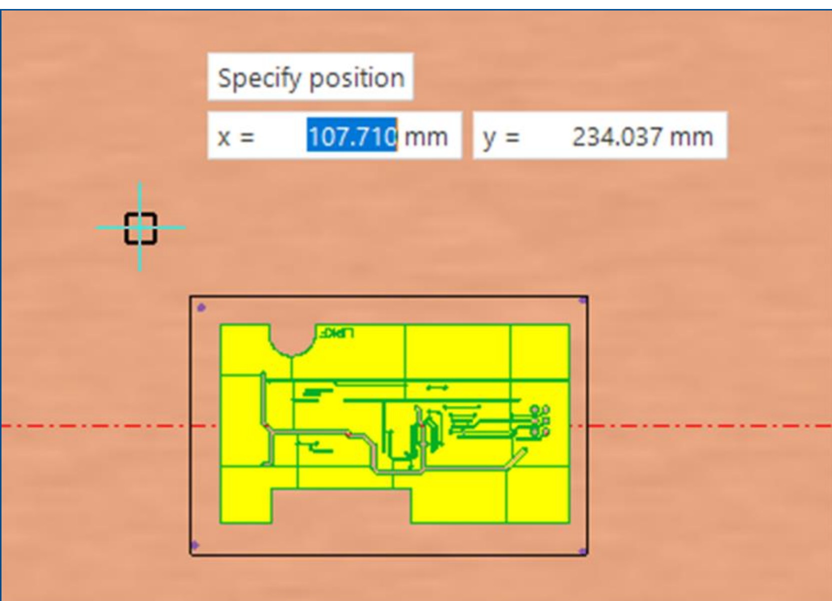


Fig. 37: Specifying the position

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:

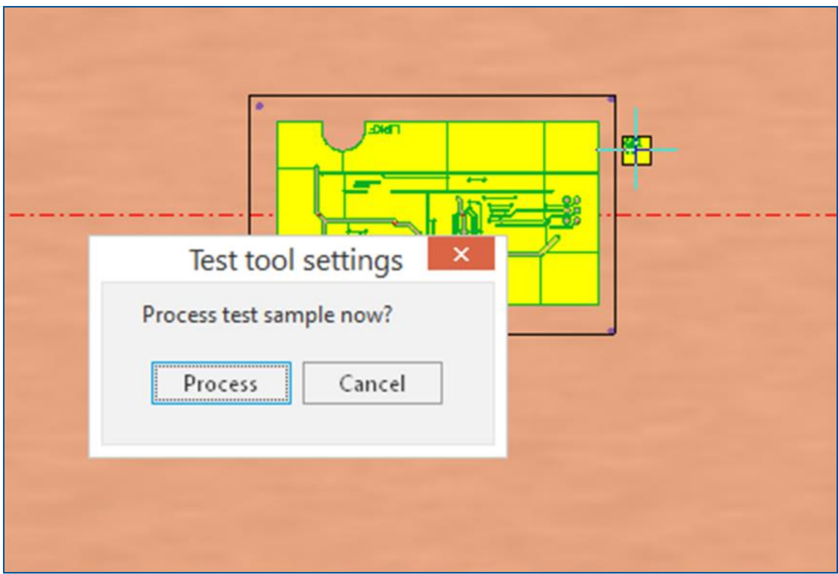


Fig. 38: Dialog *Test tool settings*

4. Click on [Process].

- ☐ Processing the test sample is started.
  When processing the test sample is finished, the following dialog is displayed:

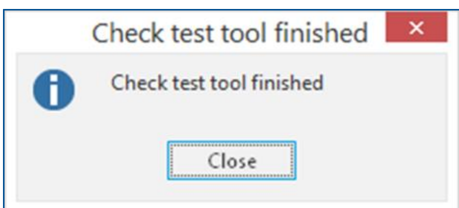


Fig. 39: Dialog *Check test tool finished*

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*.

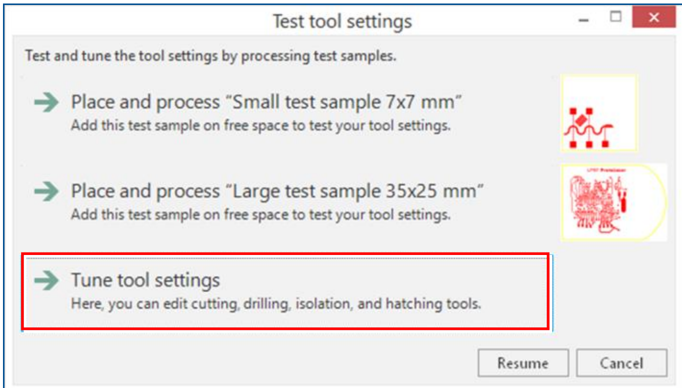


Fig. 40: ***Tune tool settings***

☐ The view *Tool Settings* is displayed:

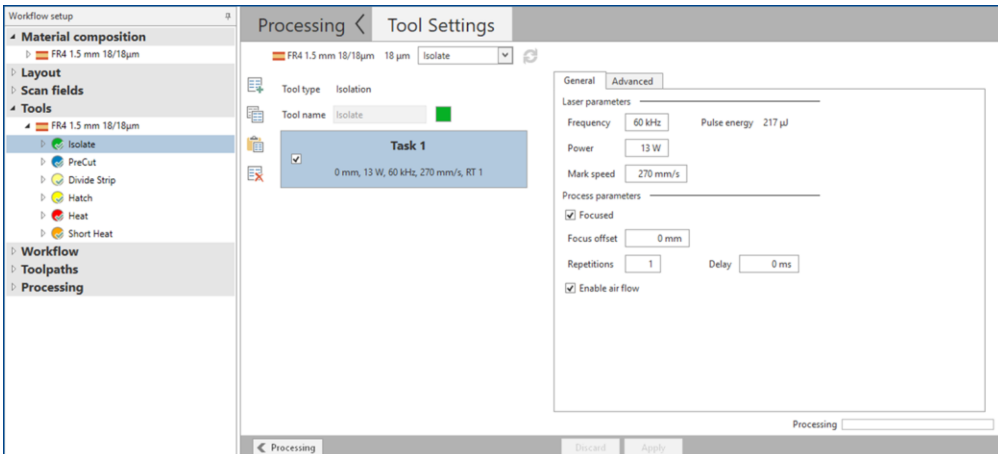


Fig. 41: ***View Tool Settings***

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

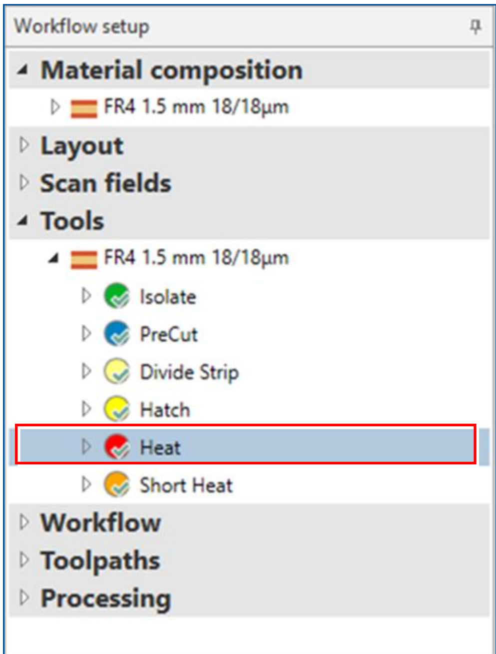
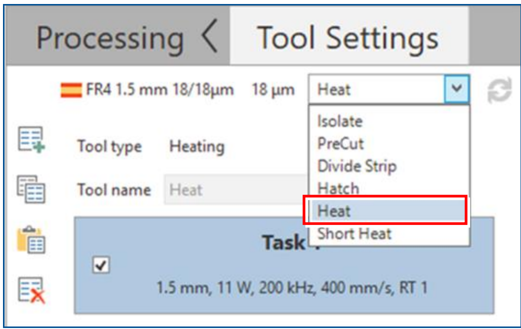


Fig. 42: Tool selected

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



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

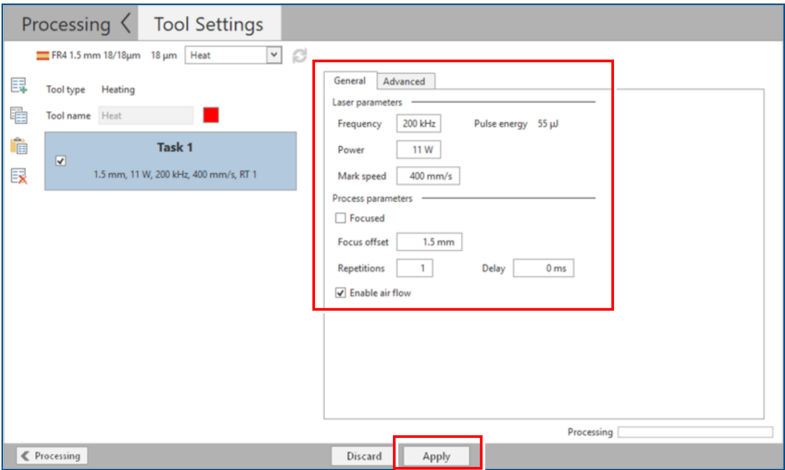


Fig. 43: Modifying the tool settings

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.

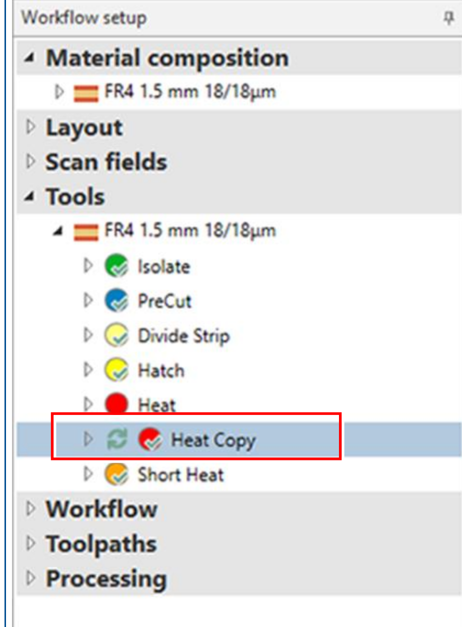


Fig. 44: Copy of the tool with the modified settings

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 🔄 next to the tool name with the modified settings.

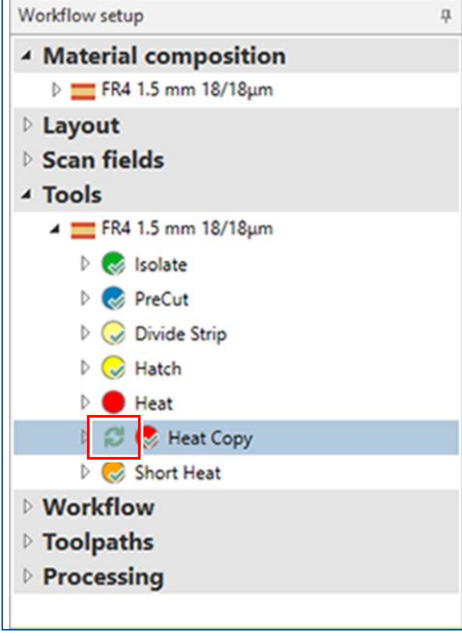


Fig. 45: Saving the tool

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

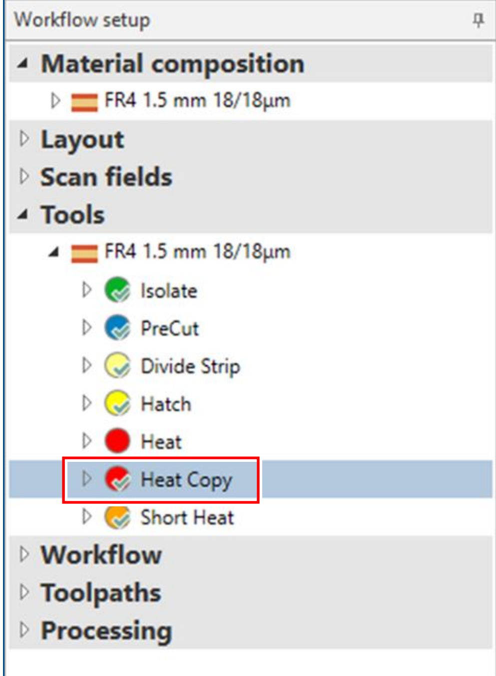


Fig. 46: Tool saved

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].

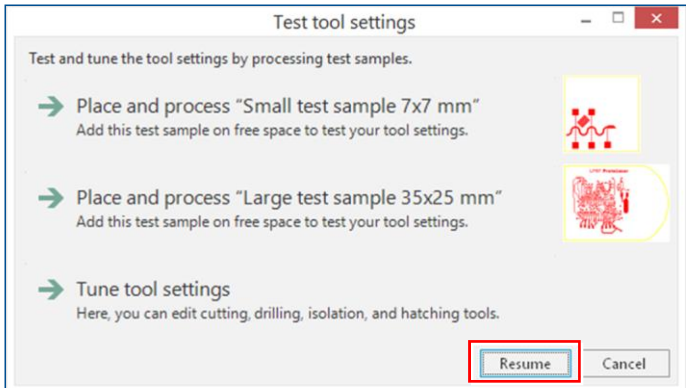


Fig. 47: Resuming the processing procedure

☑ 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.

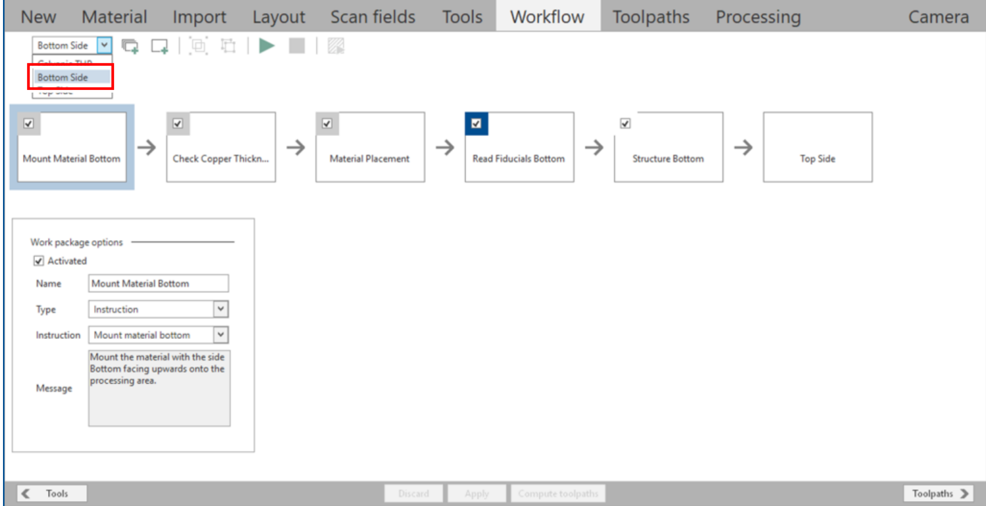


Fig. 48: Phase *Bottom Side* selected

3. Select the work package *Structure.*

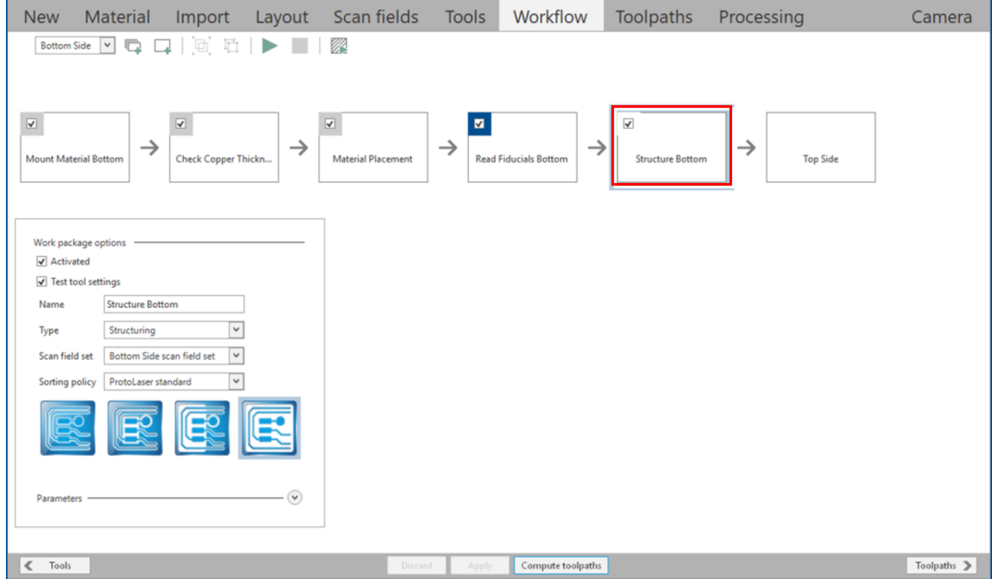


Fig. 49: Work package *Structure Bottom* selected

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

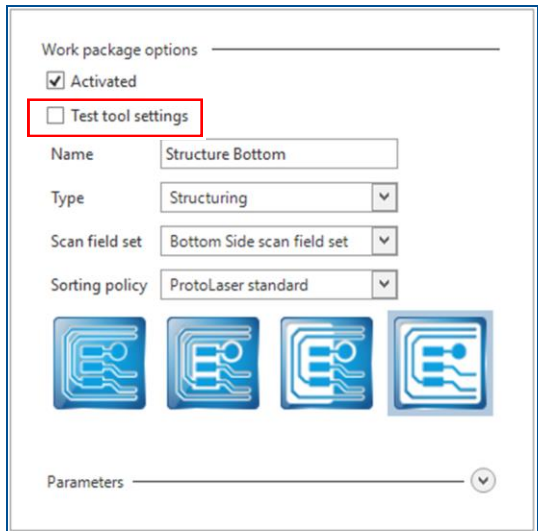


Fig. 50: *Test tool settings* disabled

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

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.
3. 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

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:

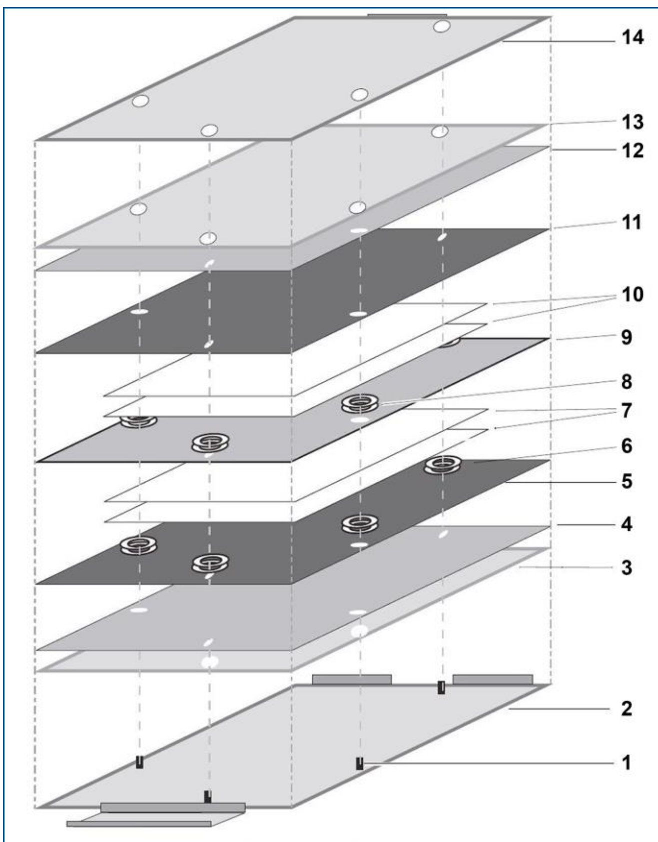


Fig. 51: Assembly of the press mold and the materials

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.

Pressing and curing the multi-layer stack

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

After preheating, the MultiPress S will prompt you by an acoustic signal and a display message to insert the multi-layer stack.

1. Press the button *ENT* and wait for the press jaws to open.
2. Open the sliding door and insert the assembled press mold together with the press cardboards into the system.
3. Close the sliding door.
4. Press the button *ENT* to start the pressing process.

☐ The pressing jaws close. The cooling phase starts automatically.

Prepressing and main pressing combined last 70 minutes. The cooling phase runs until the temperature drops below 50 °C (~122 °F).

☐ An acoustic signal indicates that the cooling phase has finished. The multi-layer stack has been pressed.

5. When the message *finished* appears on the display, open the sliding door and remove the press mold and the press cardboards from the system.
6. Leave the assembled multi-layer stack in the press mold to rest at ambient temperature in horizontal position for at least 12 to 18 hours. The resin needs to cure completely.

Alternatively, you can **accelerate the curing** cycle. Heat the pressed multi-layer stack in a convection oven for **50 minutes at 100 °C** (~212 °F) for the epoxy resin to fully cure. This way the long curing at ambient temperature can be skipped. After heating, cool the multi-layer stack at ambient temperature for 5 minutes before proceeding with production.

7. Carefully remove the multi-layer stack from the press mold.

☑ The multi-layer stack has been pressed and cured.

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):

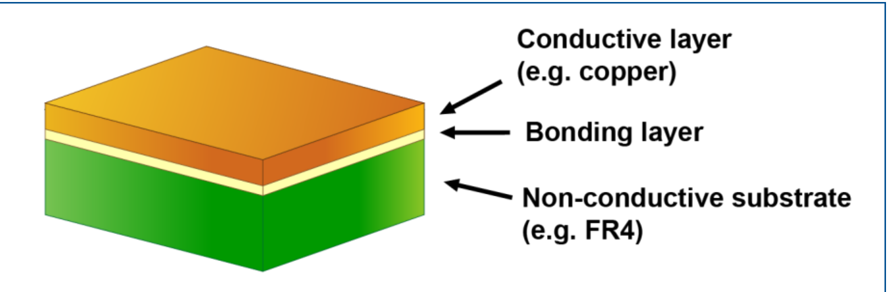


Fig. 52: Example of a laminated PCB material

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):

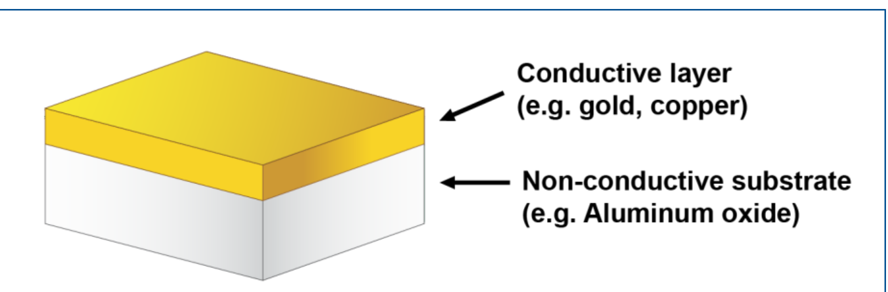


Fig. 53: Example of a non-laminated PCB material

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:

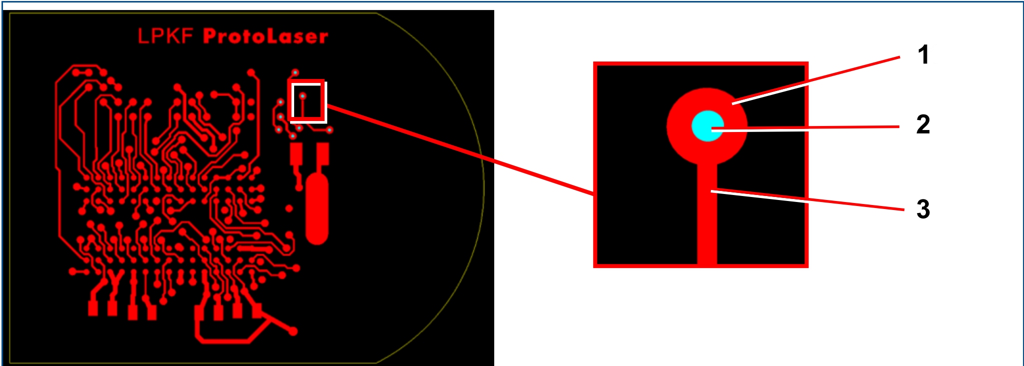


Fig. 54: PCB layout

1 Annular ring

2 Hole

3 Conductive track

The delamination method consists of the following stages:

1. Creating the isolation channels
2. Creating strips
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*:

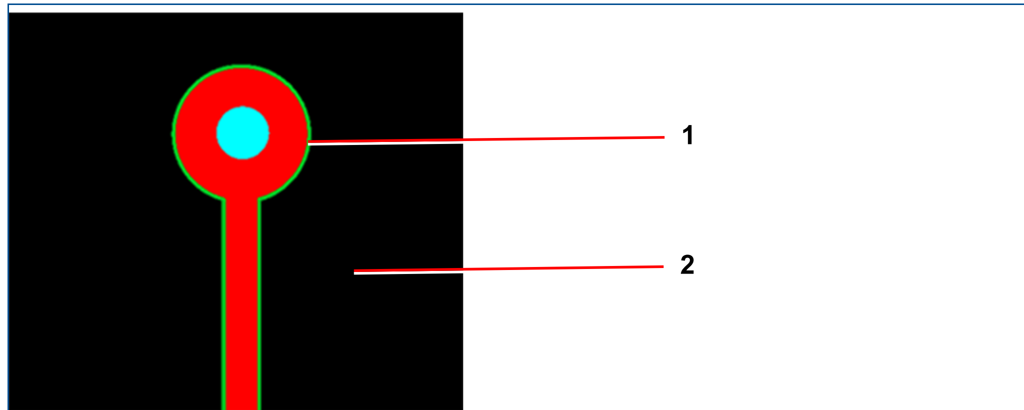


Fig. 55: PCB layout | isolation channel

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*:

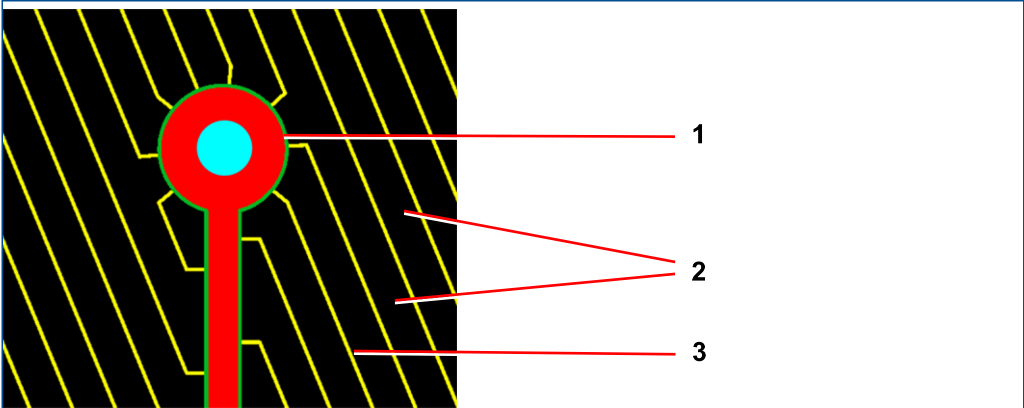


Fig. 56: PCB layout | creating strips

1 Isolation channel

2 Strips of the rubout area

3 Hatching line

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:

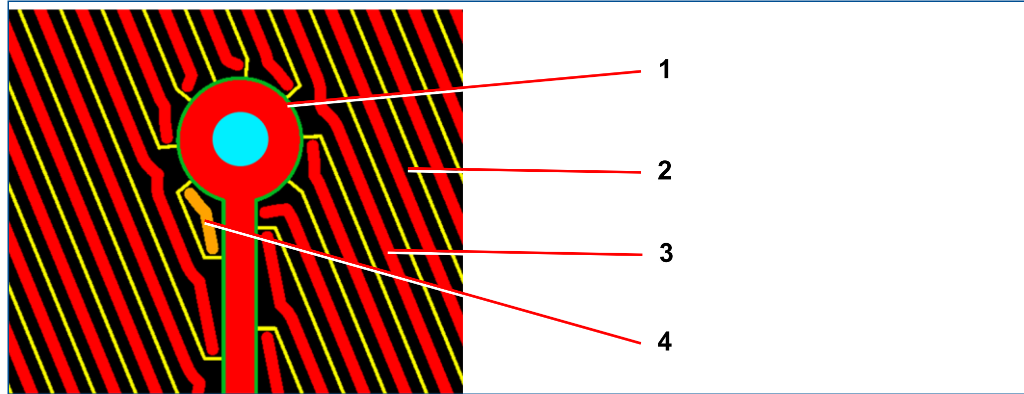


Fig. 57: PCB layout | delaminating strips

1 Isolation channel

2 Strip of the rubout area

3 Heating line

4 Short heating line

1.7.2 Hatching method

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

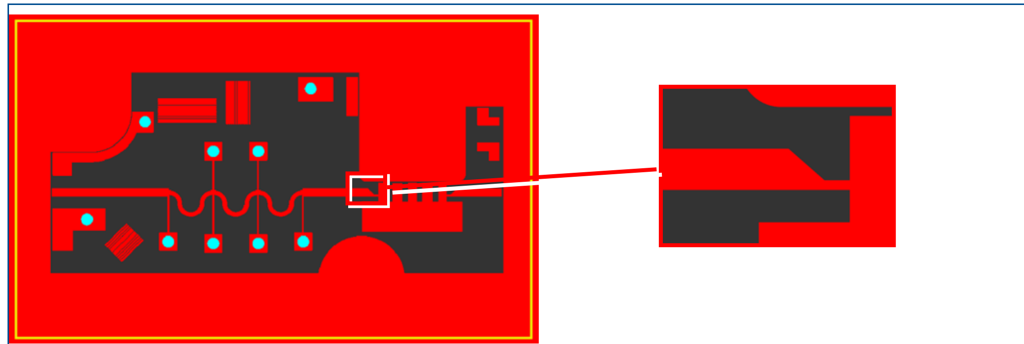


Fig. 58: RF PCB layout | RF geometry

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*:

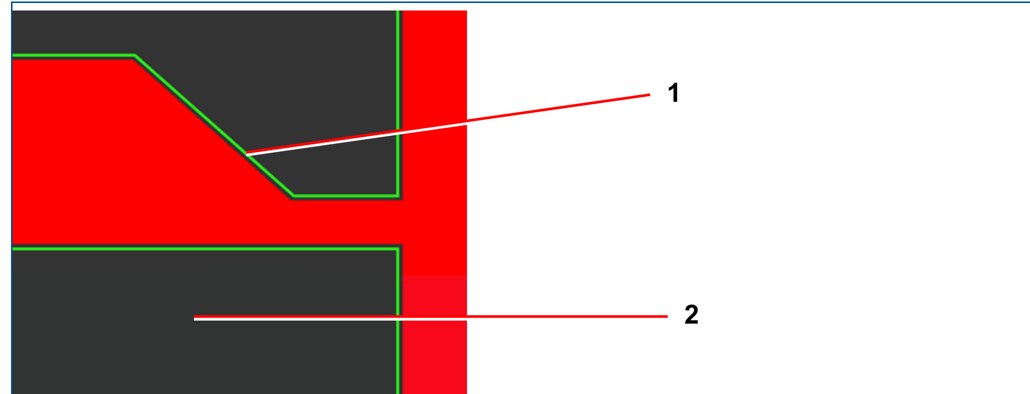


Fig. 59: PCB layout | Isolation channel

1 Isolation channel

2 Rubout area

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*:

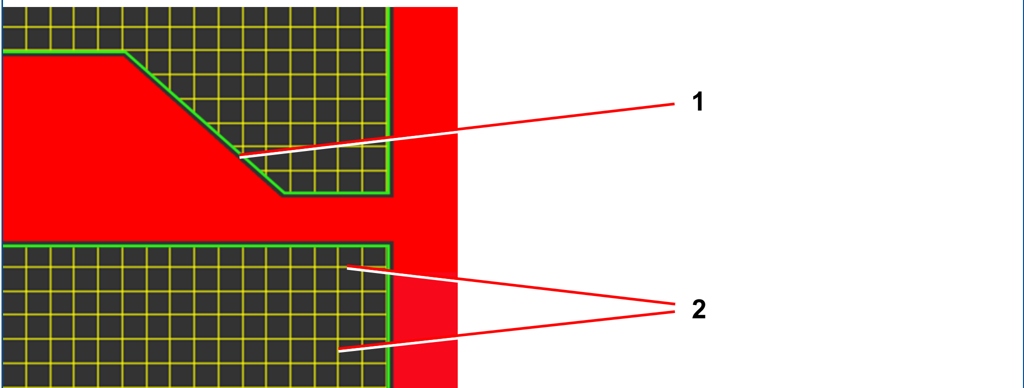


Fig. 60: PCB layout | Hatching

1 Isolation channel **2** Hatching line

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.

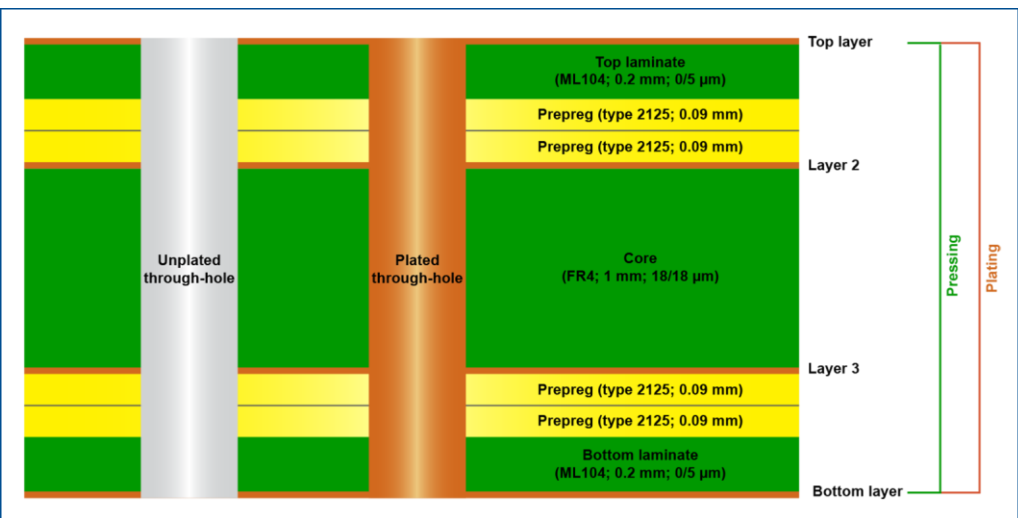


Fig. 61: A 4-layer PCB structure

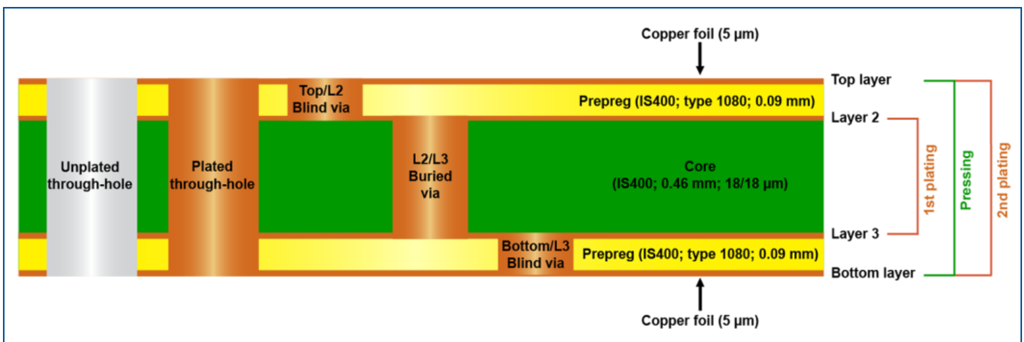


Fig. 62: A 4-layer PCB structure with blind vias and buried vias

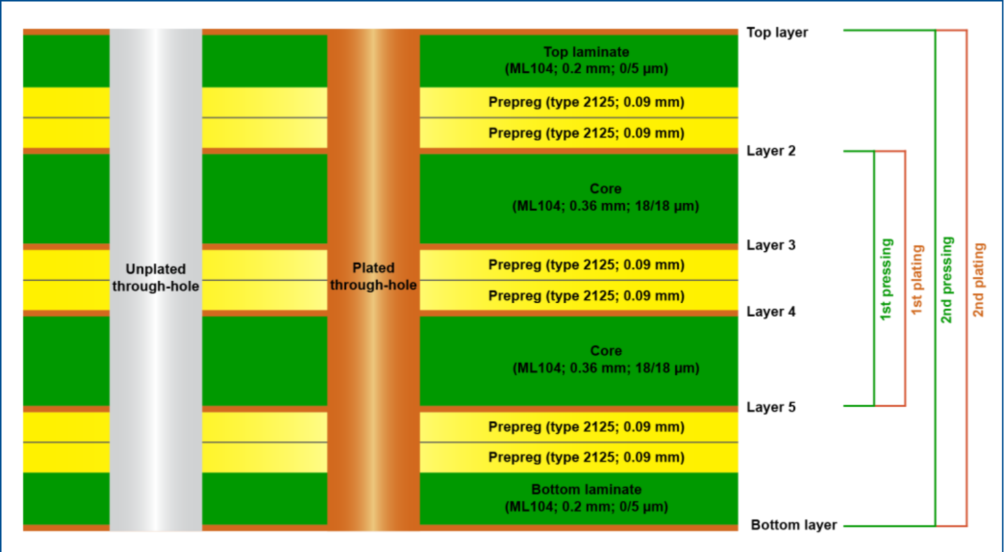


Fig. 63: A 6-layer PCB structure

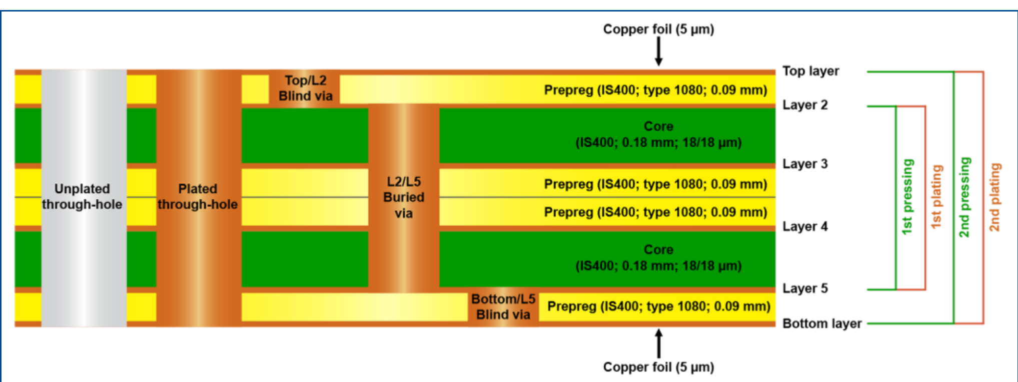


Fig. 64: A 6-layer PCB structure with blind vias and buried vias

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.

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

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

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:

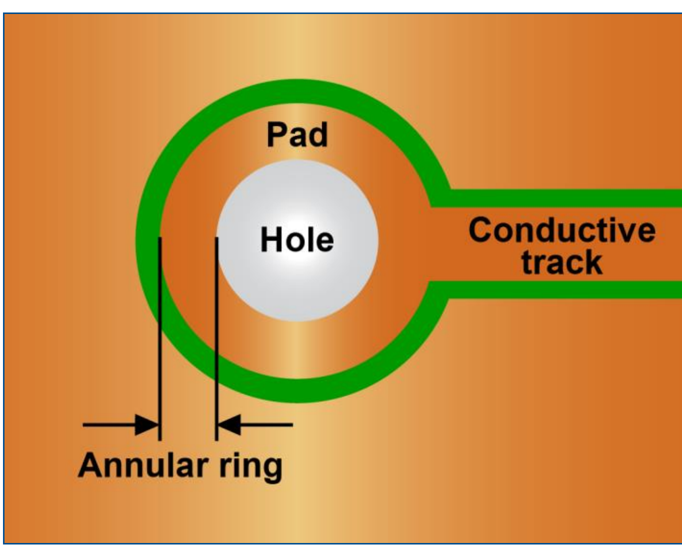


Fig. 65: Annular ring

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 µm**.

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:

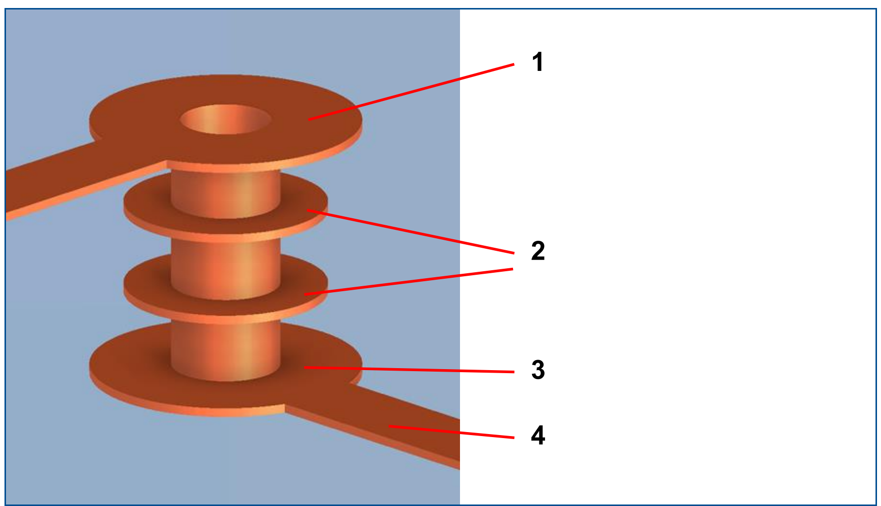


Fig. 66: A through hole with external and internal annular rings

1	External annular ring	**3**	External annular ring
2	Internal annular rings	**4**	Conductive track

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:

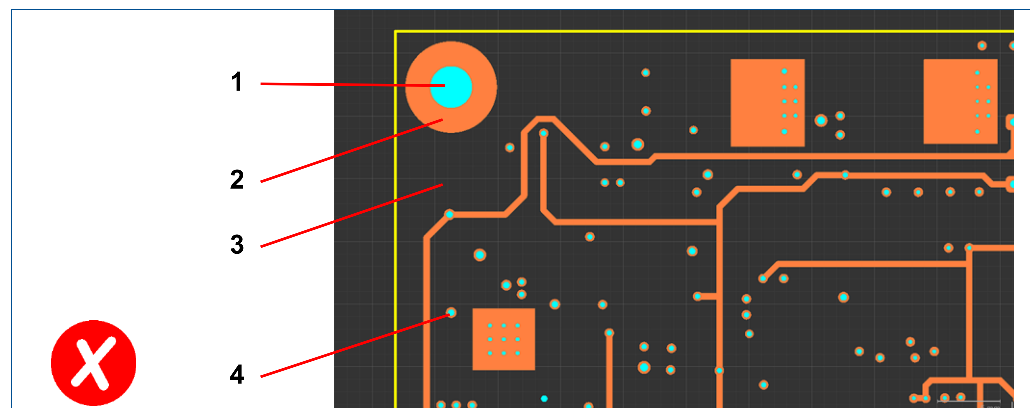


Fig. 67: Incorrect inner layer fill in the user guidance step *Layout*

1	Hole	**3**	Rubout area
2	Annular ring	**4**	Via

The following figure shows a correct inner layer fill:

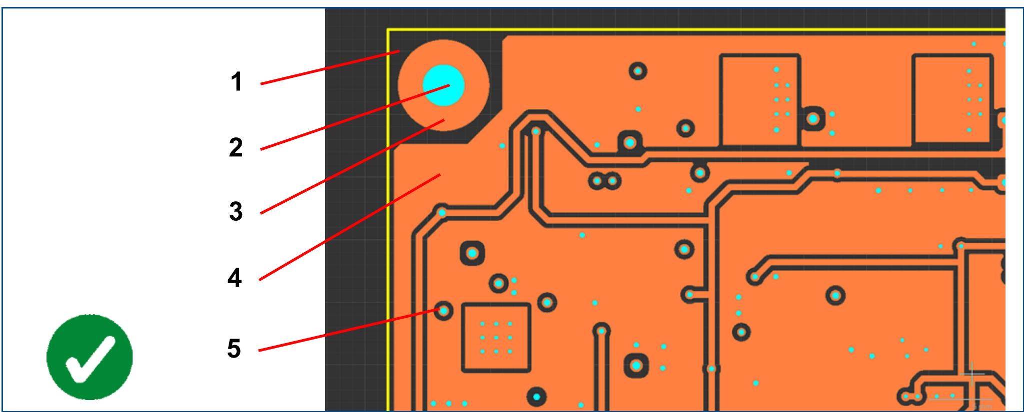


Fig. 68: Correct inner layer fill in the user guidance step *Layout*

1 Rubout area

2 Hole

3 Annular ring (=copper)

4 Copper pour

5 Via

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:

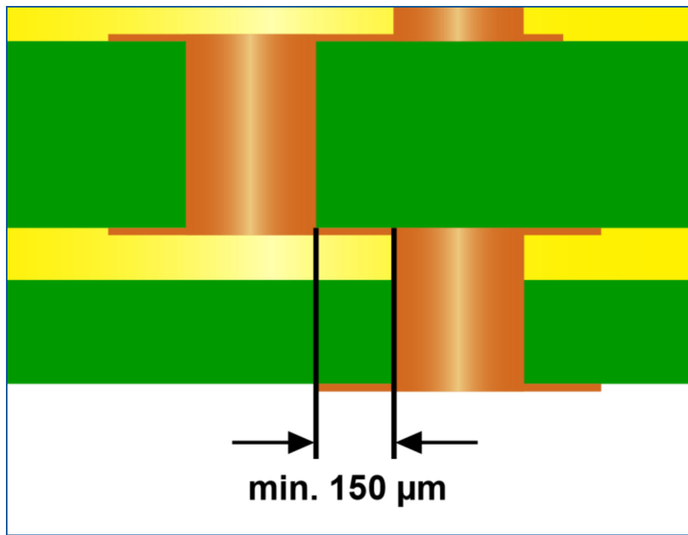


Fig. 69: Staggered vias and minimum distance

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:

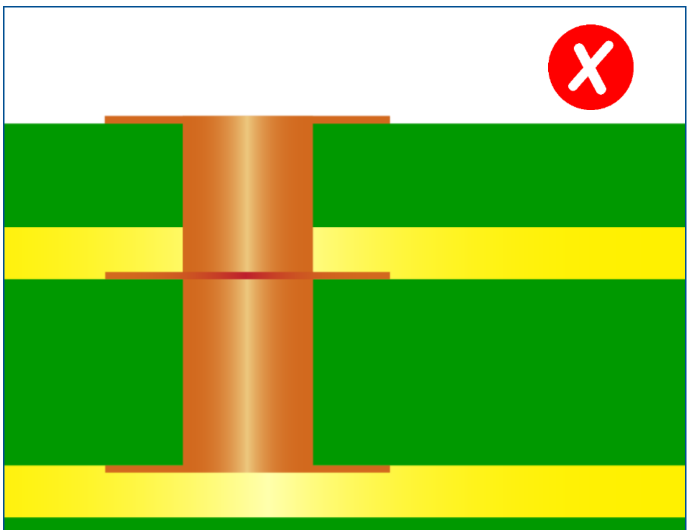


Fig. 70: Stacked vias

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

- Base material FR4, 229 mm × 305 mm × 1.5 mm, 18/18 µm copper-plated (order code: 115967)

Auxiliaries

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

System

- ProtoLaser U4/S4/R4

The following steps are performed in this tutorial:

- Switching on the system
- 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.

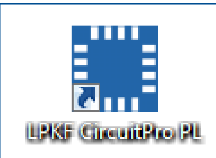


Fig. 71: Desktop icon CircuitPro PL

☐ The following message is displayed:

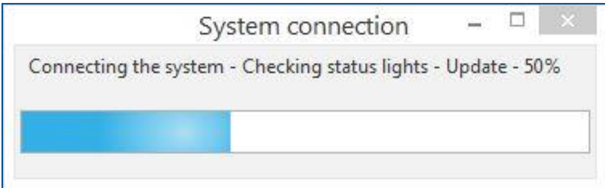


Fig. 72: Message *System connection*

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

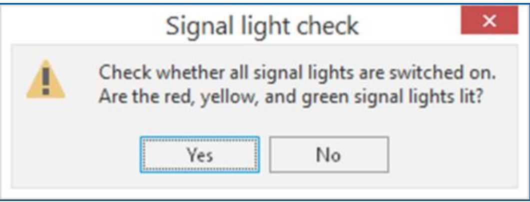


Fig. 73: Dialog *Signal light check*

3. If all lamps of the stack light are lit, click on [Yes].

☐ The user guidance step *New* is displayed:

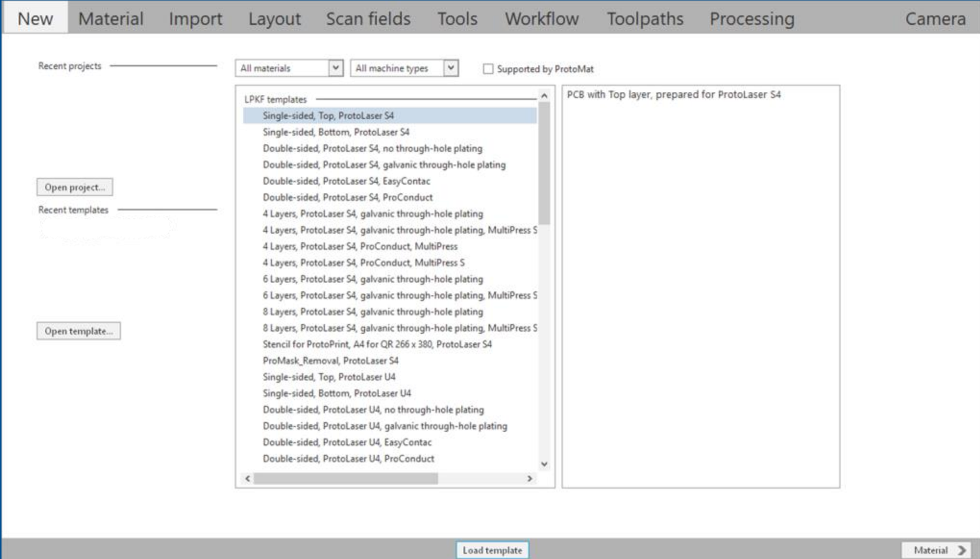


Fig. 74: User guidance step *New*

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

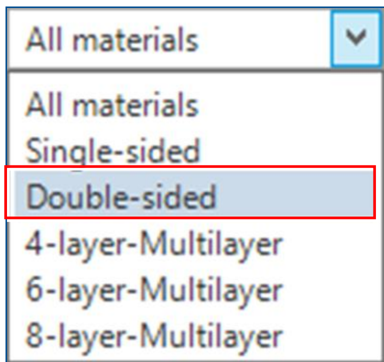


Fig. 75: Drop-down list material

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

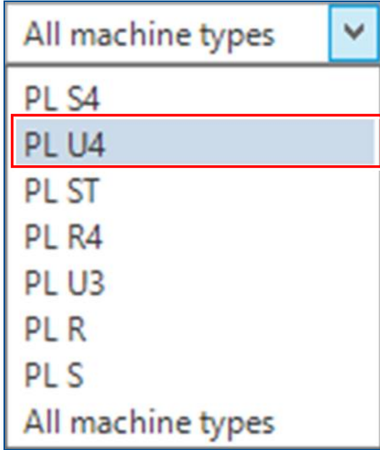


Fig. 76: Drop-down list system

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

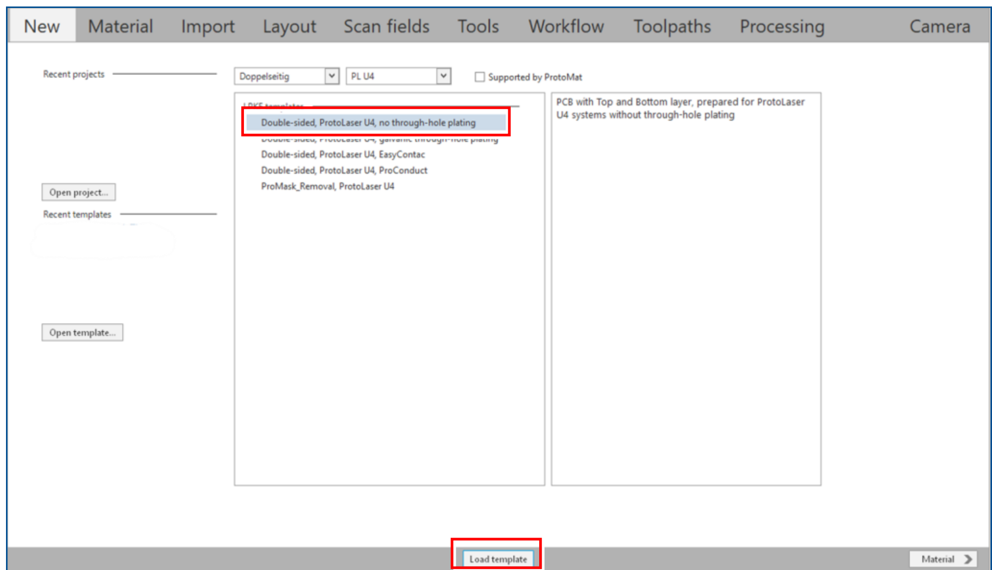


Fig. 77: List of templates

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 . You can continue to work in the user guidance step *Layout* during the warm-up phase.

Tips for selecting a template:

- 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*.

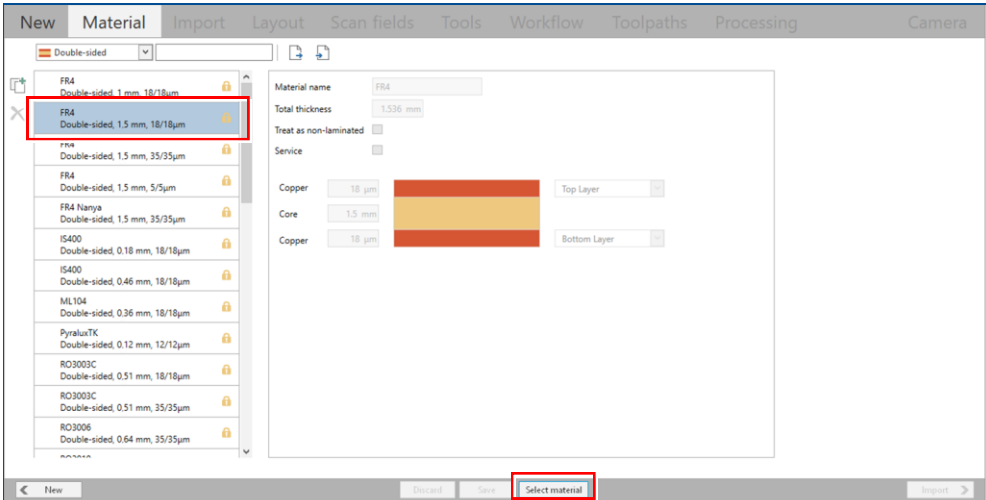


Fig. 78: User guidance step *Material*

2. Click on [Select material] or double-click on the material.

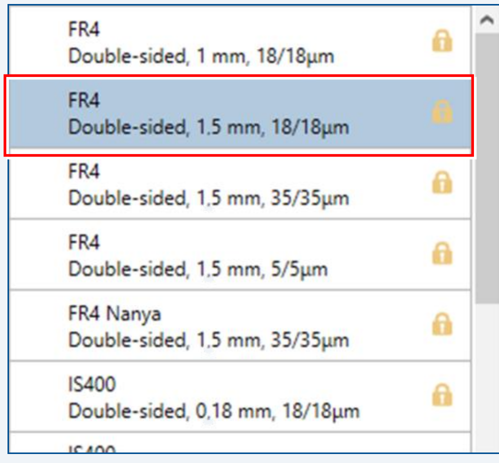
☐ The user guidance step *Import* is displayed.

☑ 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* – material type
- *1.55 mm* – material thickness
- *18/18 µm* – copper thickness

Importing the data

1. In the user guidance step *Import* click on .

☐ The following dialog is displayed:

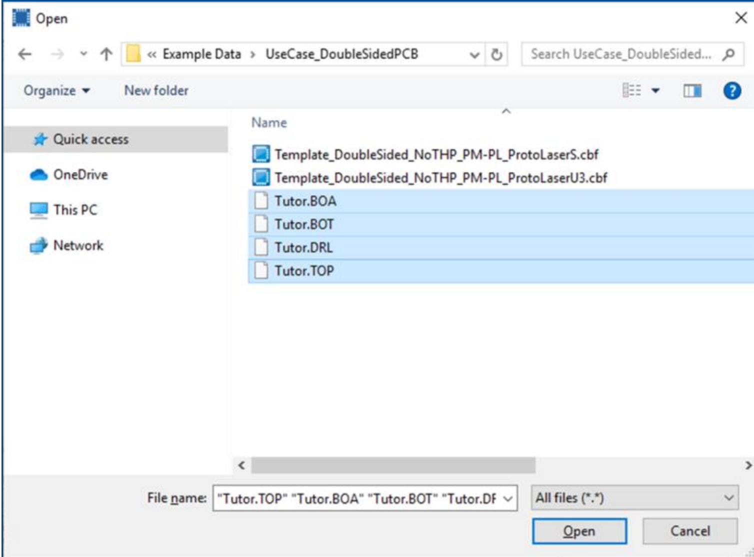


Fig. 79: Dialog *Open*

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:

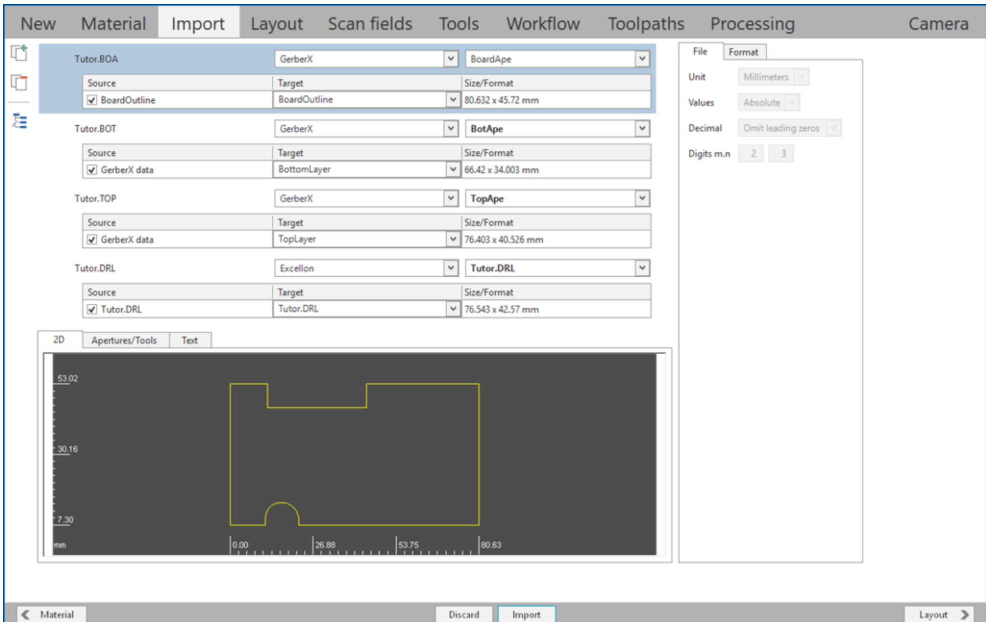


Fig. 80: User guidance step *Import*

5. Assign the holes to the layer *DrillUnplated*.

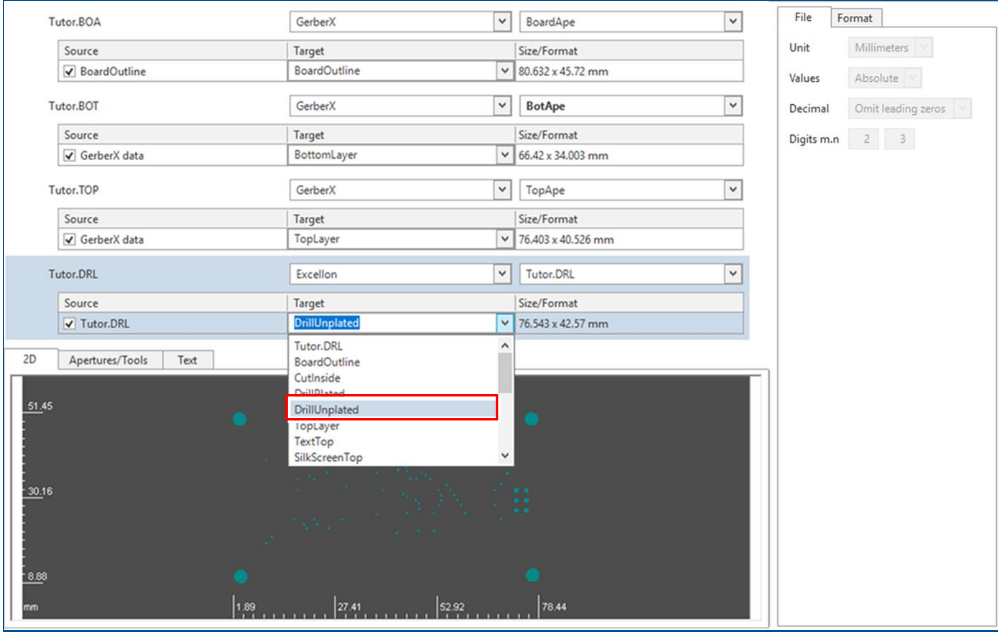


Fig. 81: Assigning holes to layer

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].

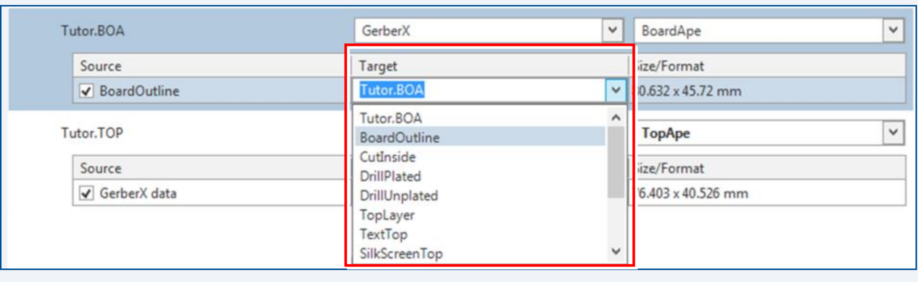
- ☐ The user guidance step *Layout* is displayed:

Fig. 82: User guidance step *Layout*

☑ 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.



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:

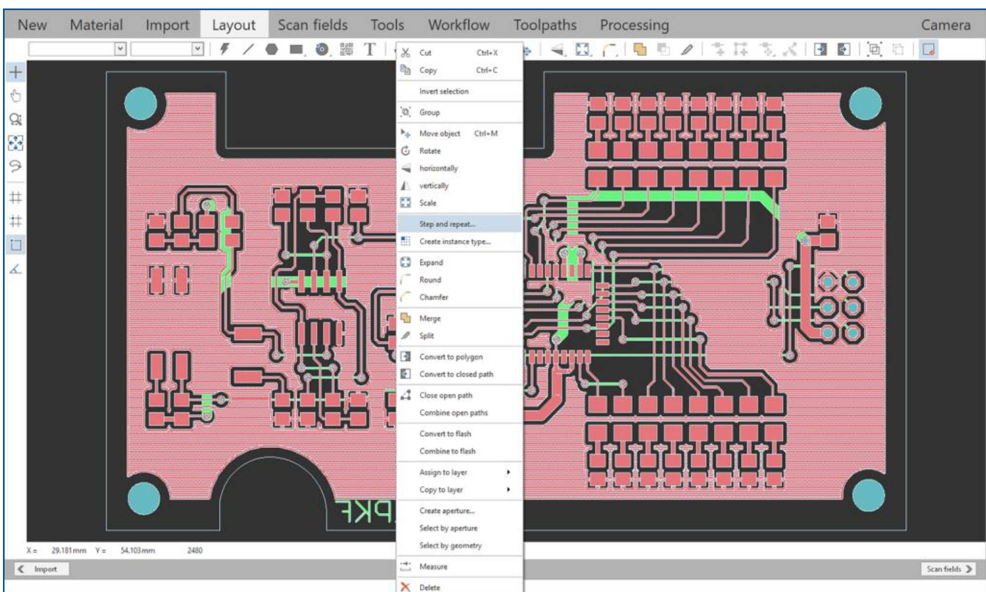


Fig. 83: Context menu *Step and repeat*

3. Click on *Step and repeat...*

☐ The following dialog is displayed:

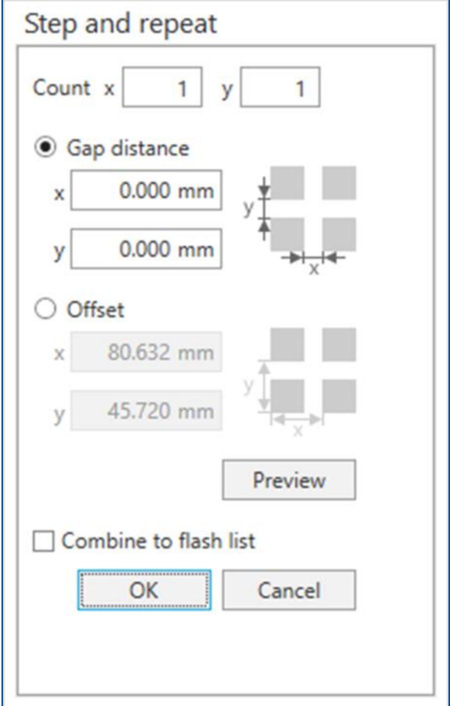


Fig. 84: Dialog *Step and repeat*

4. 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:

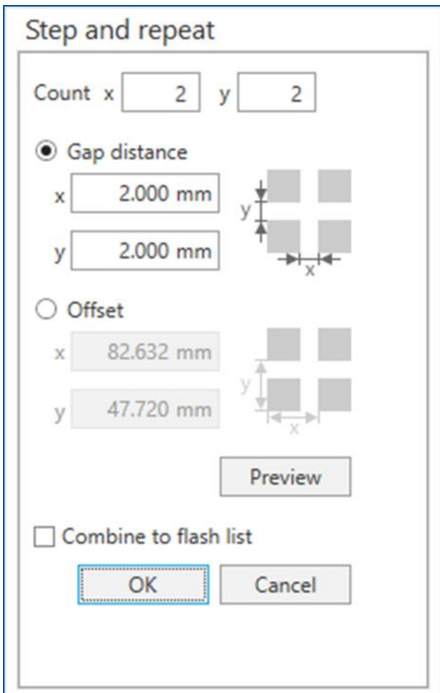


Fig. 85: Dialog *Step and repeat* after entering values

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 [icon].

8. Press [Esc] or click anywhere on the black background to deselect the highlighted layout.

☐ The user guidance step *Layout* changes as follows:

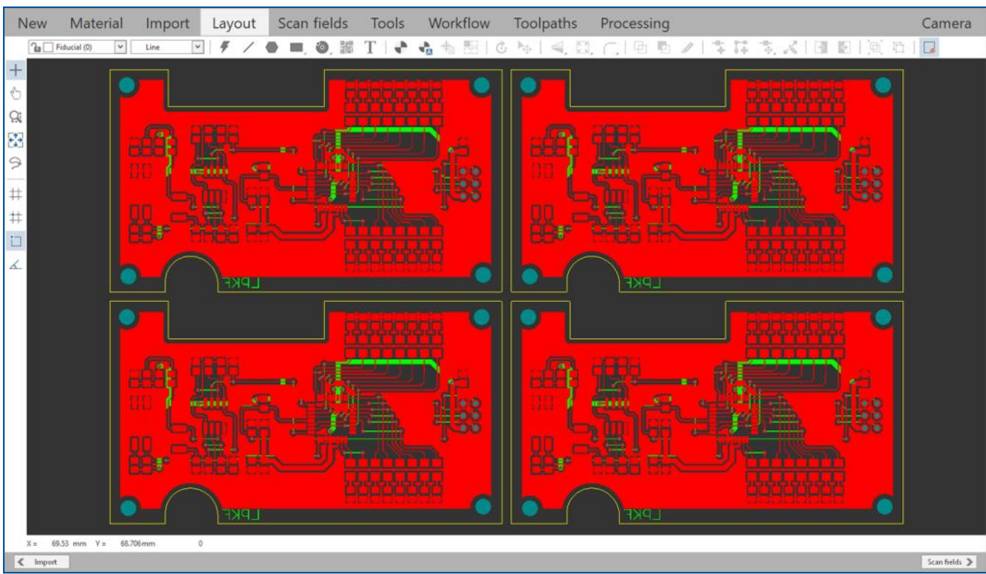


Fig. 86: Multiplied layout

☑ The layout has been multiplied.

Creating an instance type

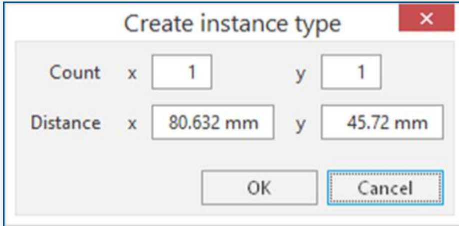
As an alternative to multiplying the layout data, you can use instances. The advantages are:

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

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

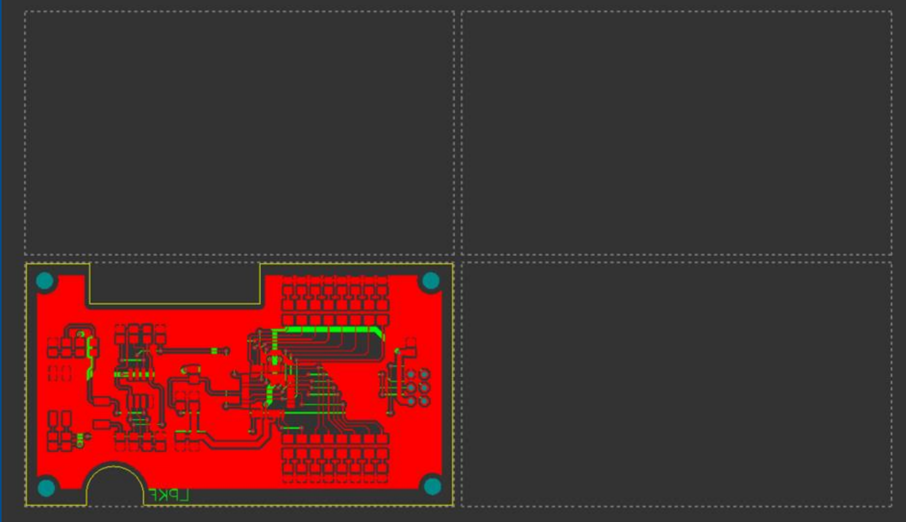
1. Select the entire layout by pressing [Ctrl] + [A].
2. Click on *Insert* > *Create instance type* or on .

☐ The following dialog is displayed:



3. 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

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.

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

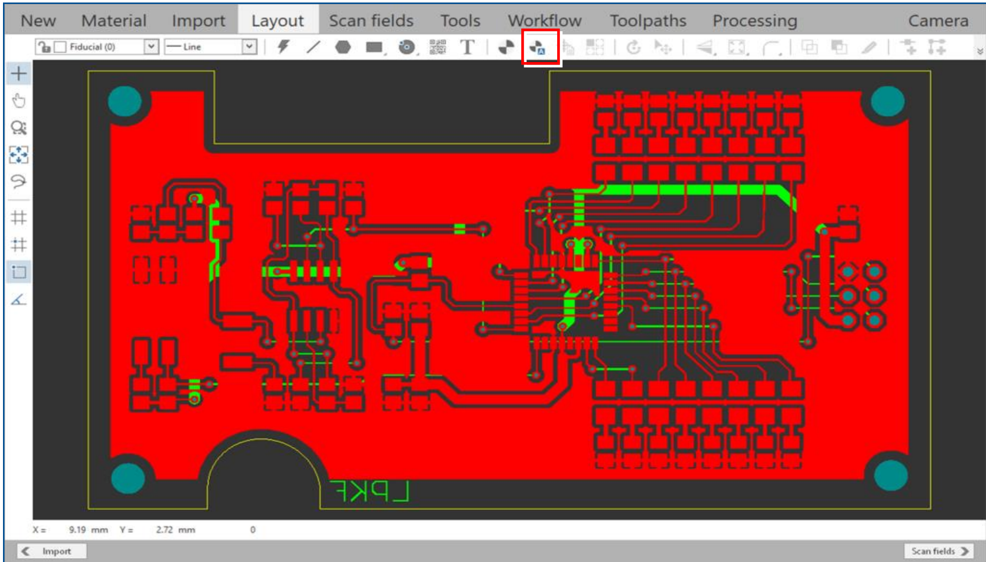


Fig. 87: Create fiducials automatically

- The following dialog is displayed:

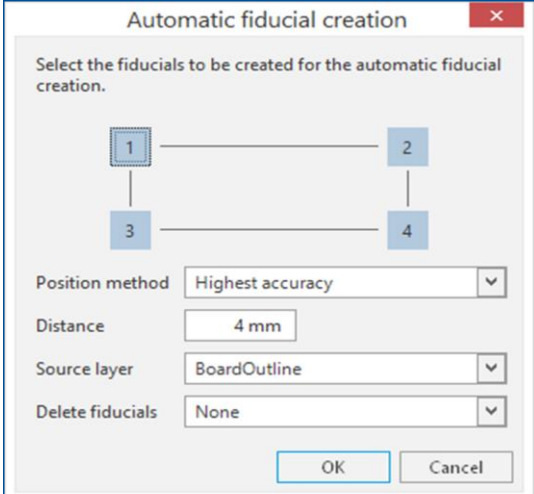


Fig. 88: Dialog *Automatic fiducial creation*

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.

2. 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:

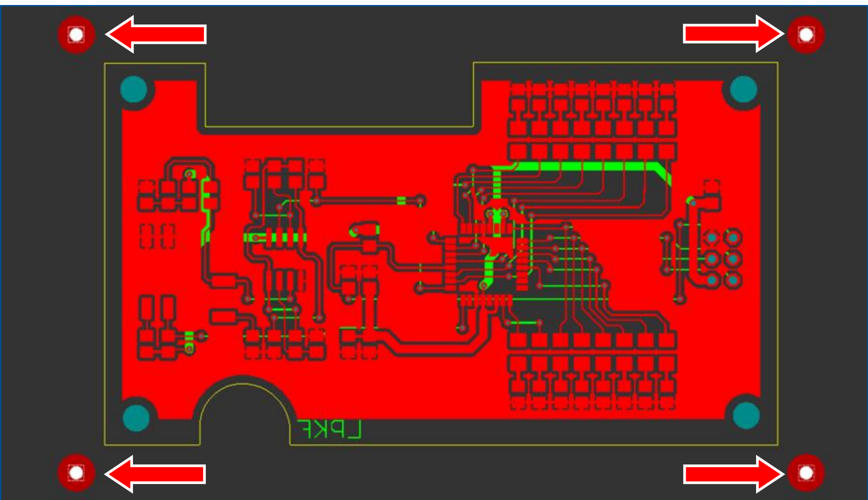


Fig. 89: Fiducials created

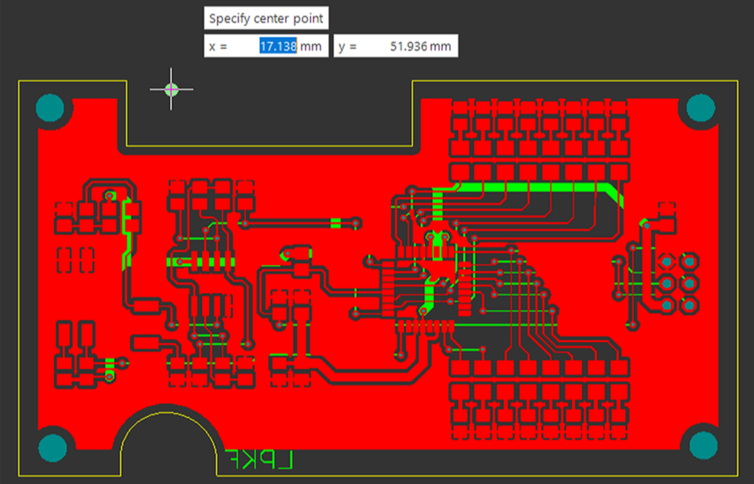
☑ The fiducials have been created.

Creating fiducials manually

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

1. Click on *Insert* > *Fiducial* or click on .

☐ 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*.
2. Click on [Compute scan fields] or on .

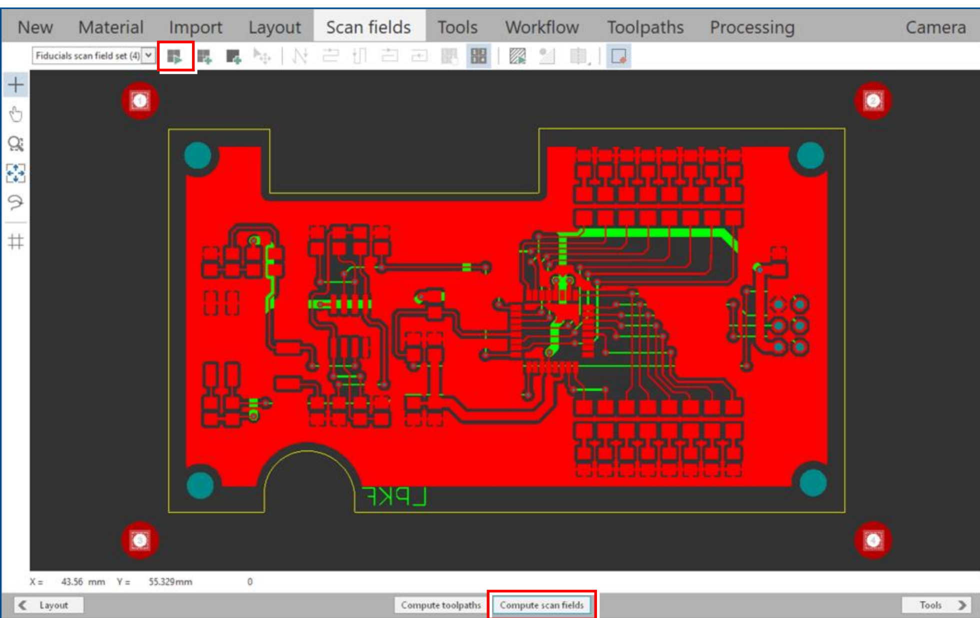


Fig. 90: ***Compute scan fields***

- The following dialog is displayed:

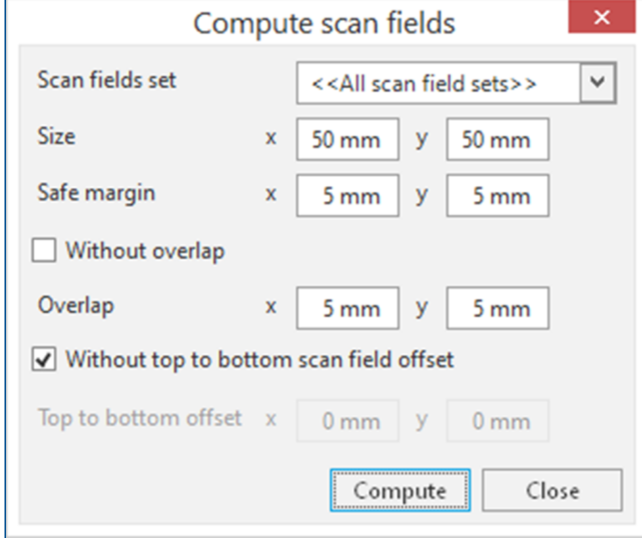


Fig. 91: **Dialog *Compute scan fields***

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.

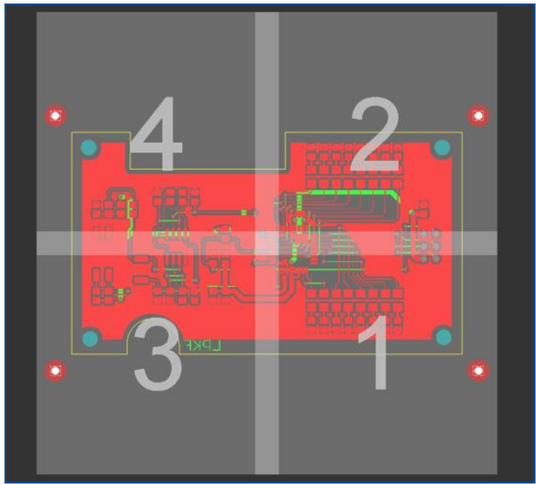


Fig. 92: Scan fields computed

5. Click on [Compute toolpaths] or on .

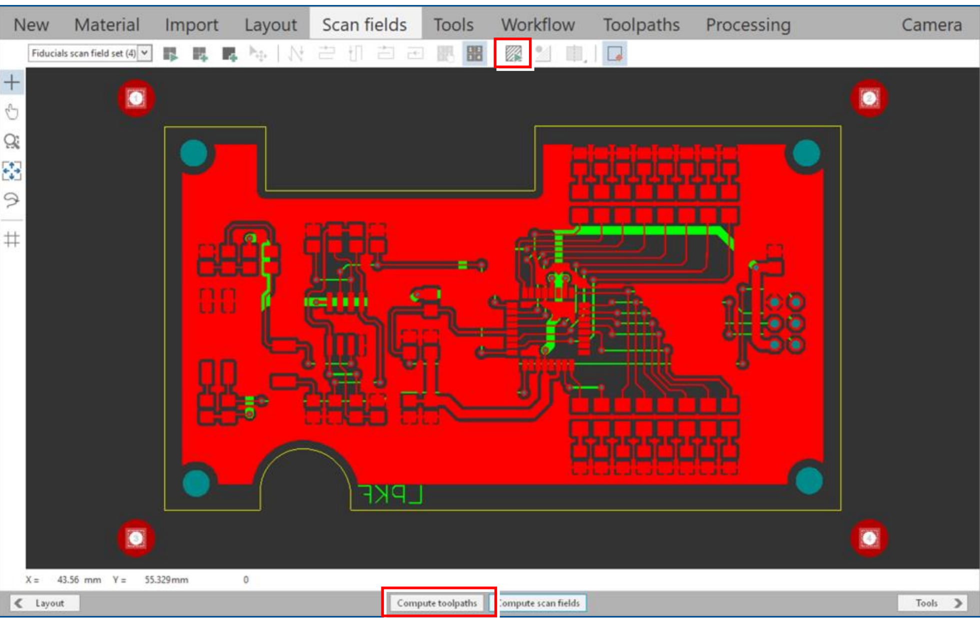


Fig. 93: *Computing toolpaths*

- The following message is displayed:

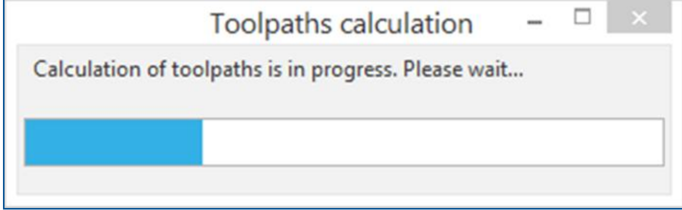


Fig. 94: Message *Toolpaths calculation*

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

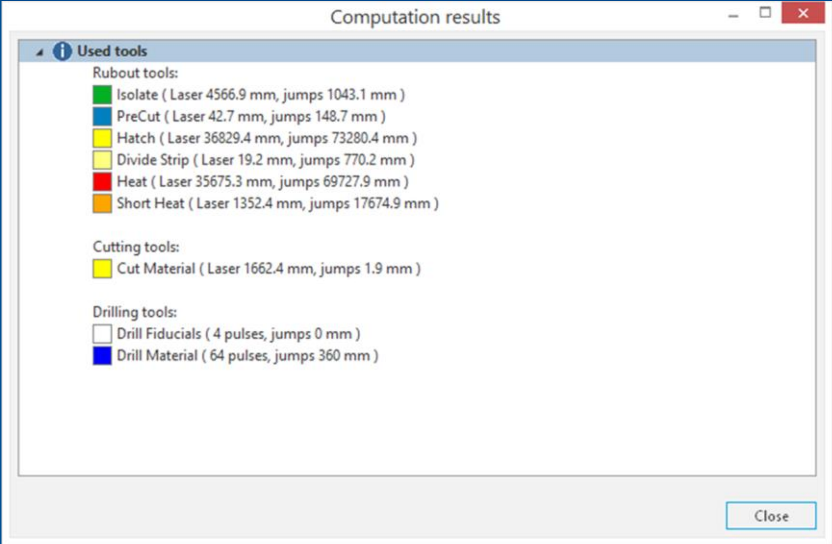


Fig. 95: Message *Computation results*

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*.

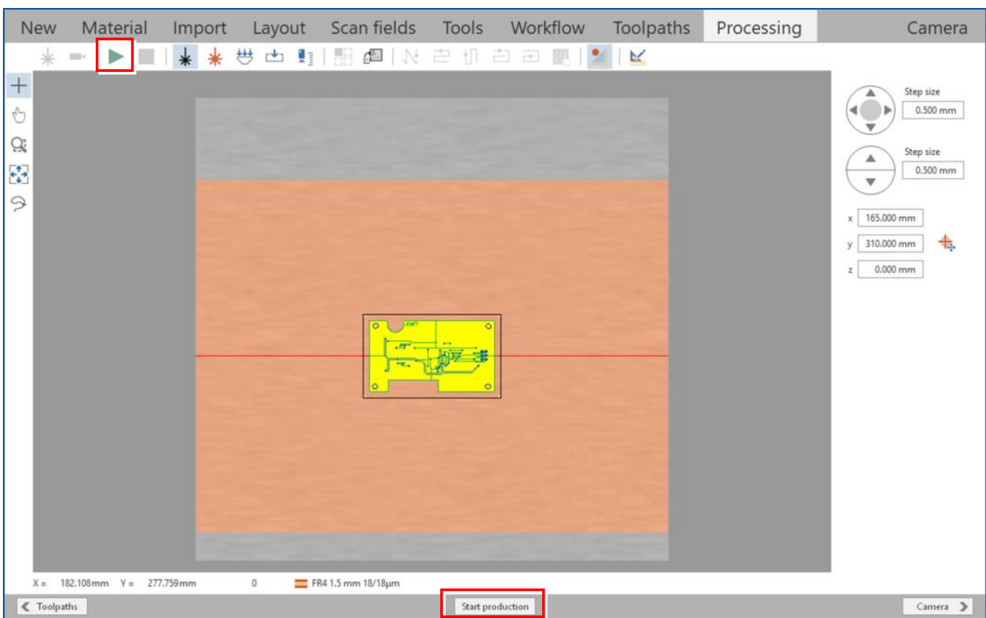


Fig. 96: User guidance step *Processing*

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

- ☐ The following message is displayed:

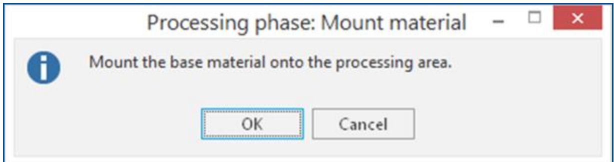


Fig. 97: Dialog *Processing phase: Mount material*

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:

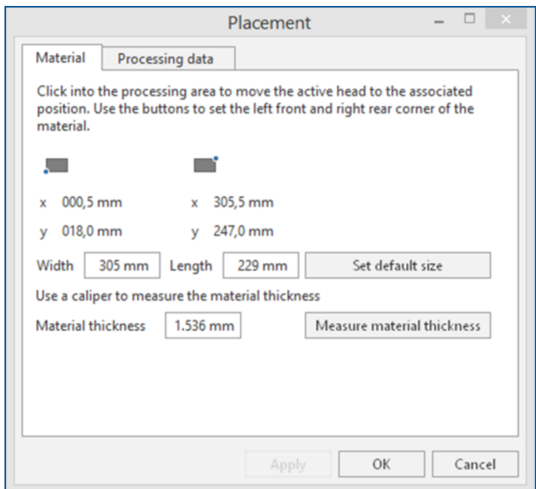


Fig. 98: Dialog *Placement*

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*.

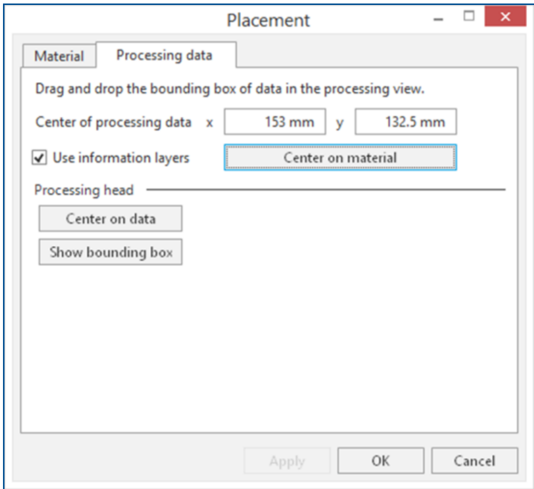


Fig. 99: Dialog *Placement* | *Processing data*

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:

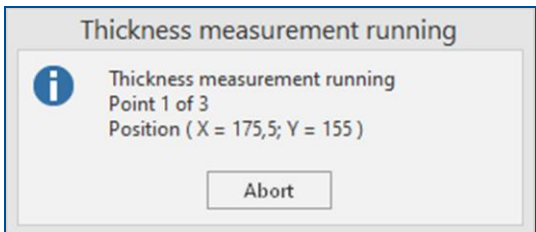


Fig. 100: Message *Thickness measurement running*

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

When finished, the following message is displayed:

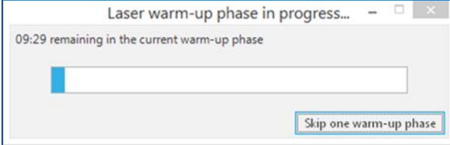


Fig. 101: Message *Laser warm-up phase in progress*

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

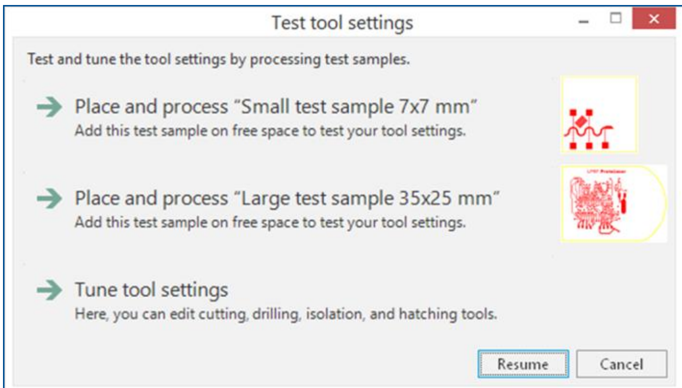


Fig. 102: Dialog *Test tool settings*

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:

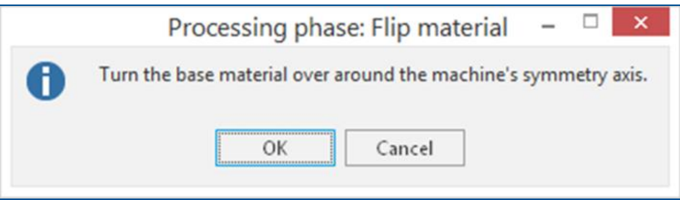


Fig. 103: Message *Processing phase: Flip material*

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*).

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.

18. Click on [Resume].

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

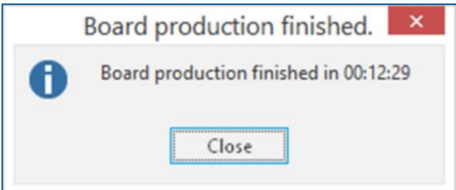


Fig. 104: Message *Board production finished*

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
• Base material FR4, 229 mm × 305 mm × 1.5 mm, 5/5 µm copper-plated with protection foil (order code: SET-10-1053)	• 1 set of tools for ProtoMat • LPKF Cleaner (order code: 115891) • Brush • Oil-free compressed air • Tap water	• ProtoLaser U4/S4/R4 • ProtoMat S or E • Contac S4

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:

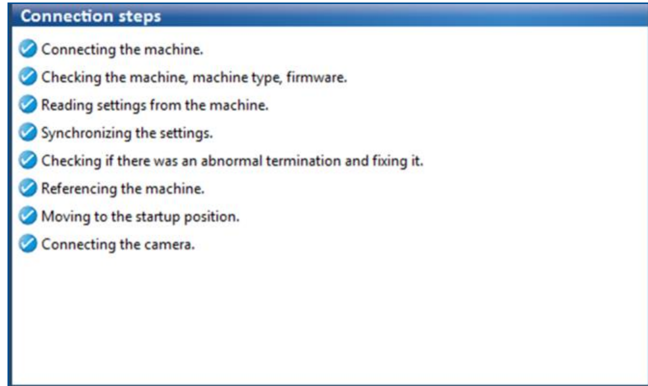


Fig. 106: Dialog *Connection steps*

☐ The following dialog is displayed:

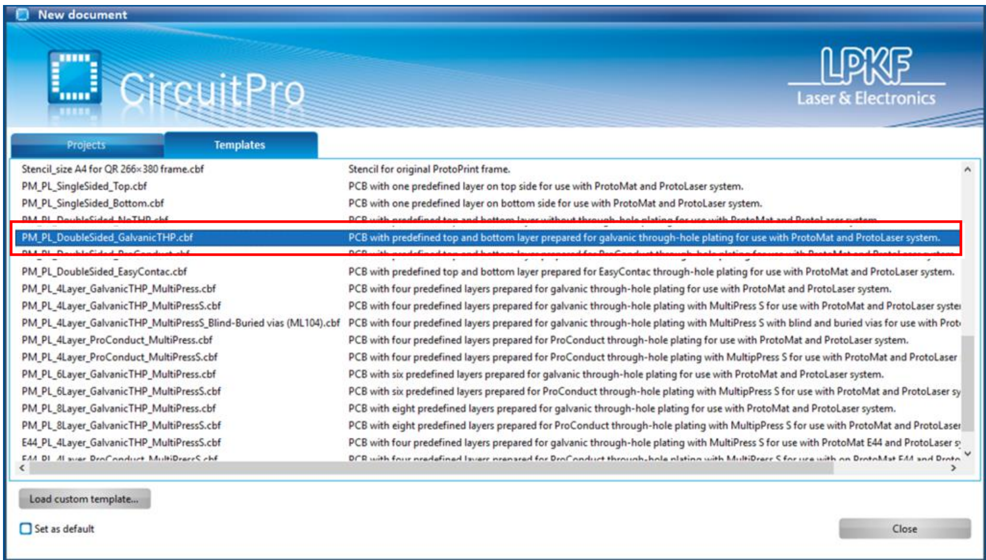


Fig. 107: Dialog *New document*

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

☑ The ProtoMat has been switched on.

Importing the data in CircuitPro PM

1. Click on *File* > *Import* or on .

☐ The following dialog is displayed:

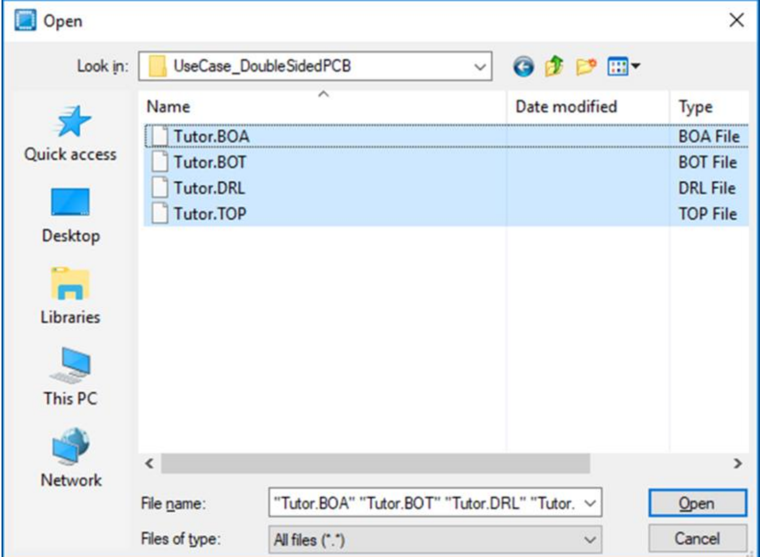


Fig. 108: Dialog *Open*

2. 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:

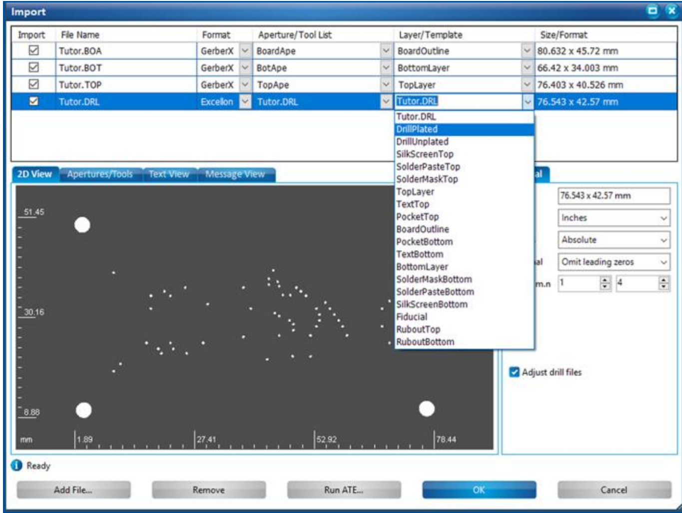


Fig. 109: Dialog *Import* | Assigned layers

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:

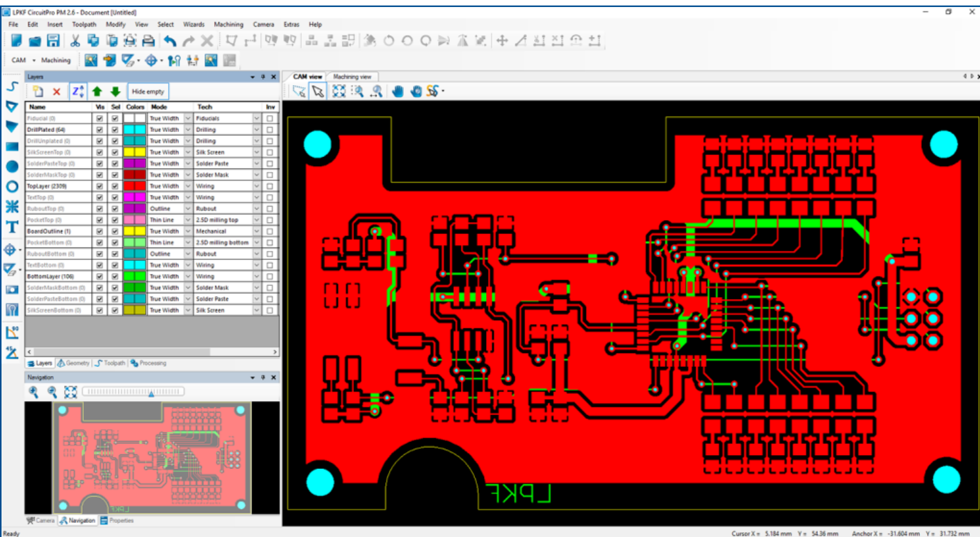


Fig. 110: Imported data in the *CAM view*

☑ 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 `Ctrl` key and click on all holes that are to be assigned to the layer ***DrillUnplated***.

- The holes are highlighted in gray:

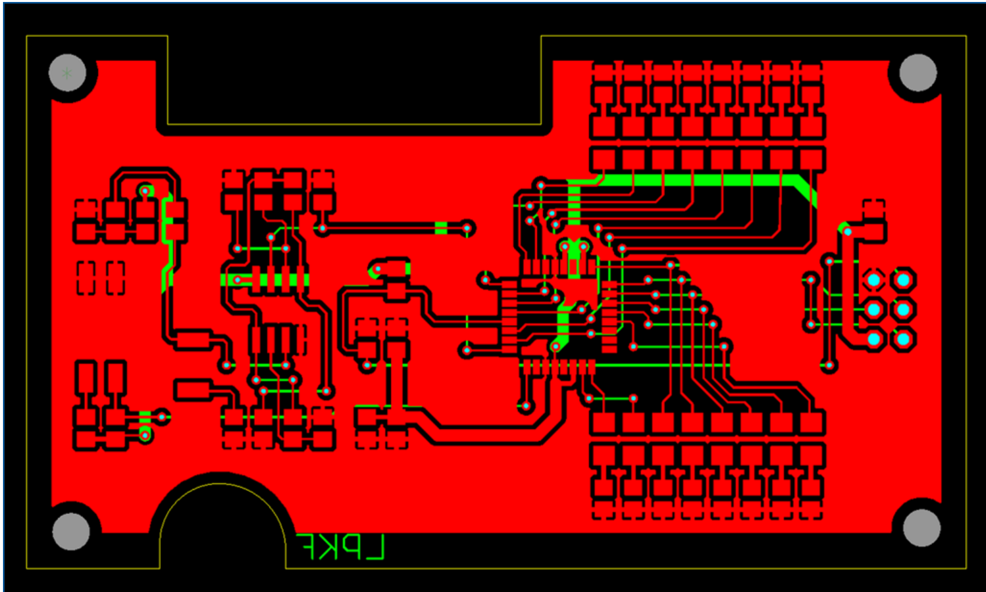


Fig. 111: *CAM view* after selecting holes

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

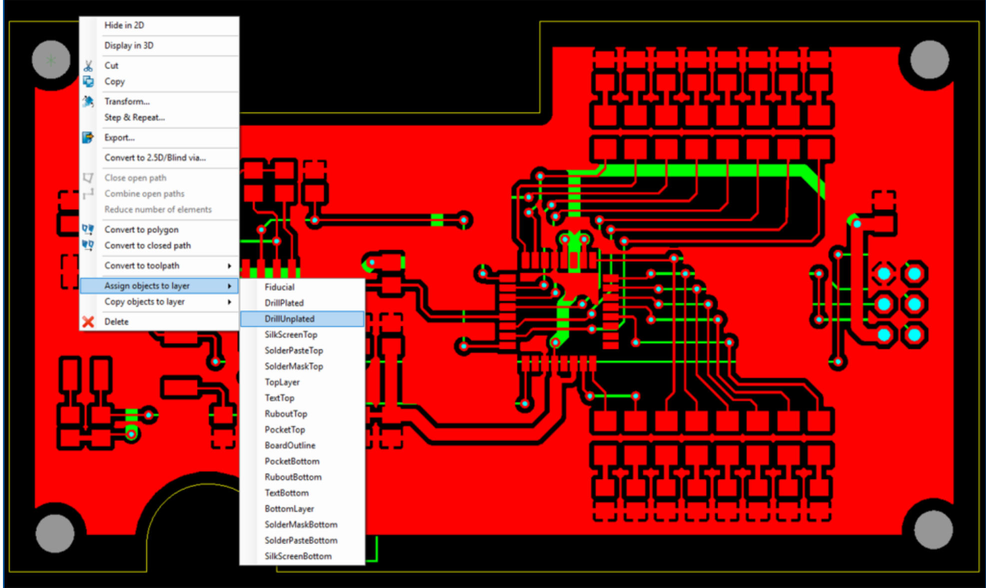


Fig. 112: Context menu *Assign objects* to *DrillUnplated* layer

- 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:

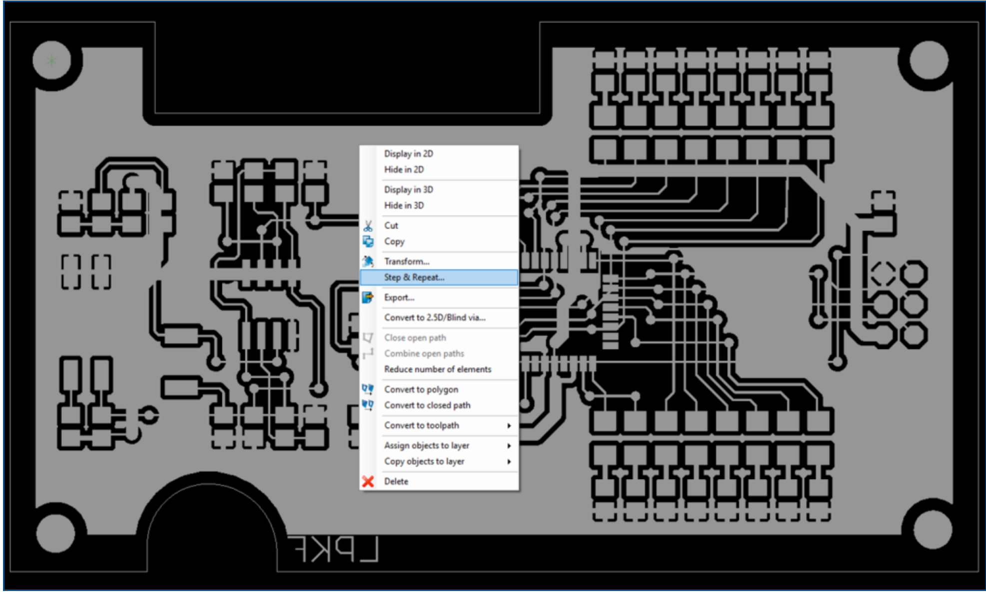


Fig. 113: Context menu *Step & Repeat*

3. Click on *Step & Repeat*...

- ☐ The following dialog is displayed:

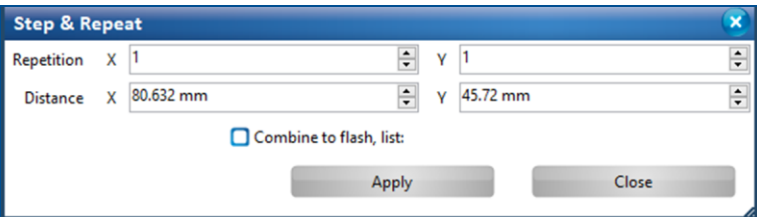


Fig. 114: Dialog *Step & Repeat*

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:

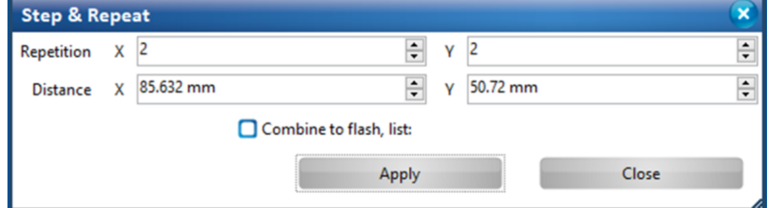


Fig. 115: Dialog *Step & Repeat* after entering values

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:

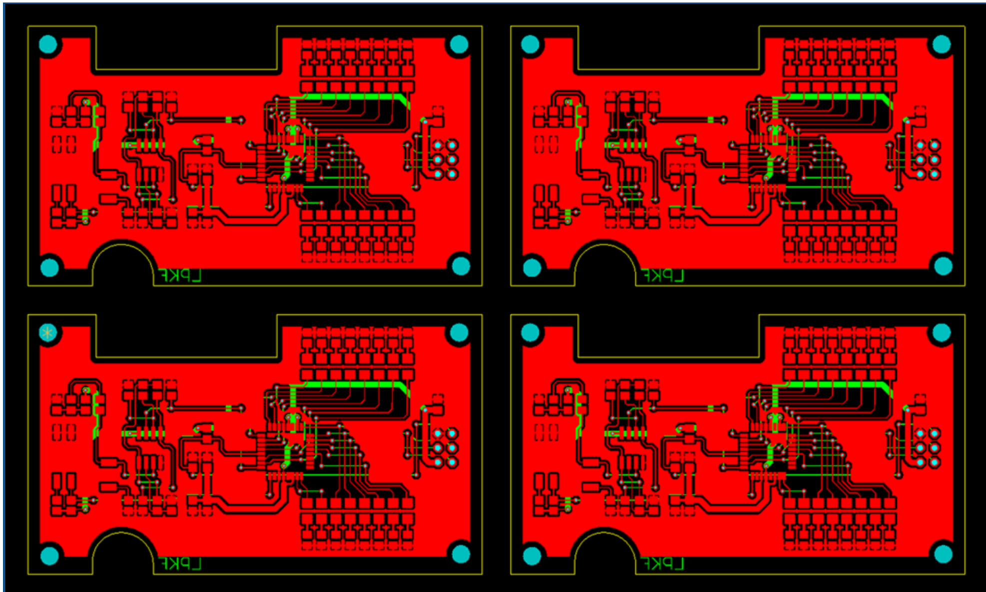


Fig. 116: *CAM view* of the multiplied layout

☑ 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:

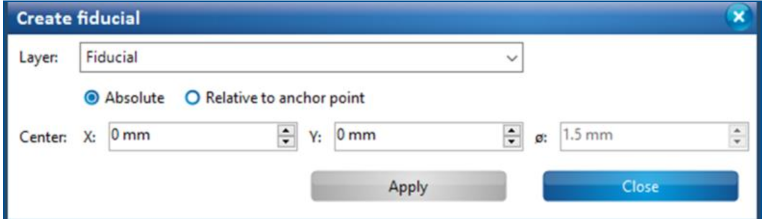


Fig. 117: Dialog *Create fiducial*

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:

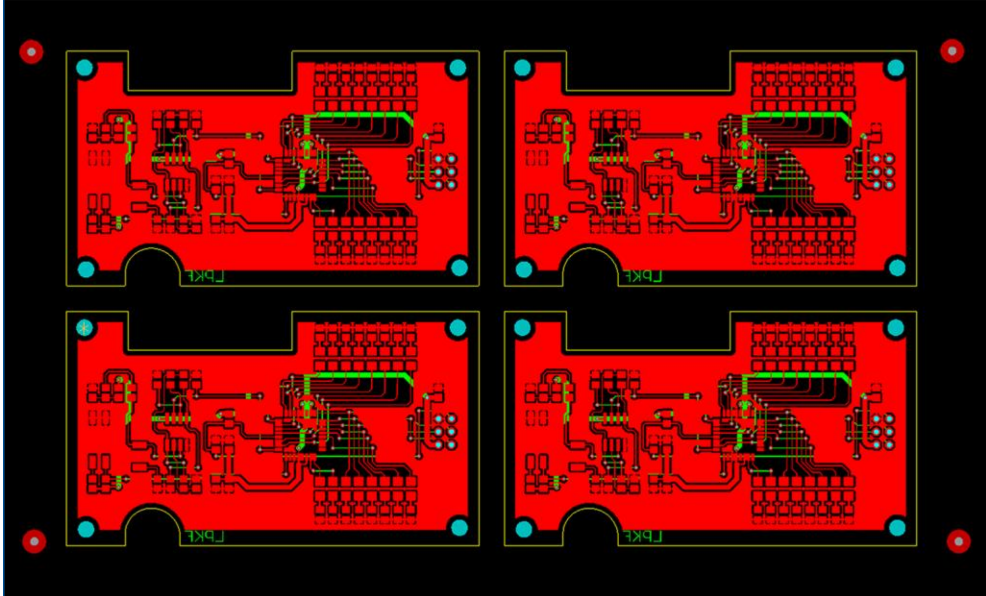


Fig. 118: *CAM view* after creating fiducials

☑ 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 [icon].

☐ The dialog *Technology Dialog* is displayed.

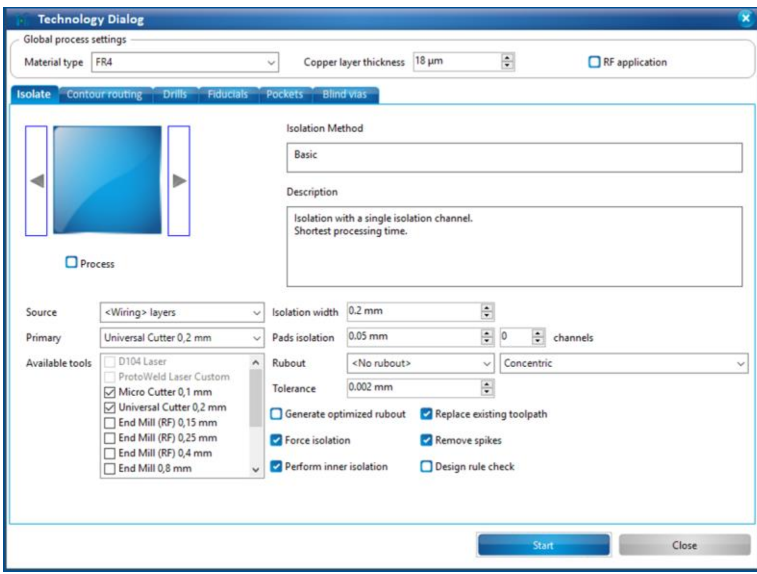


Fig. 119: Tab *Isolate*

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.

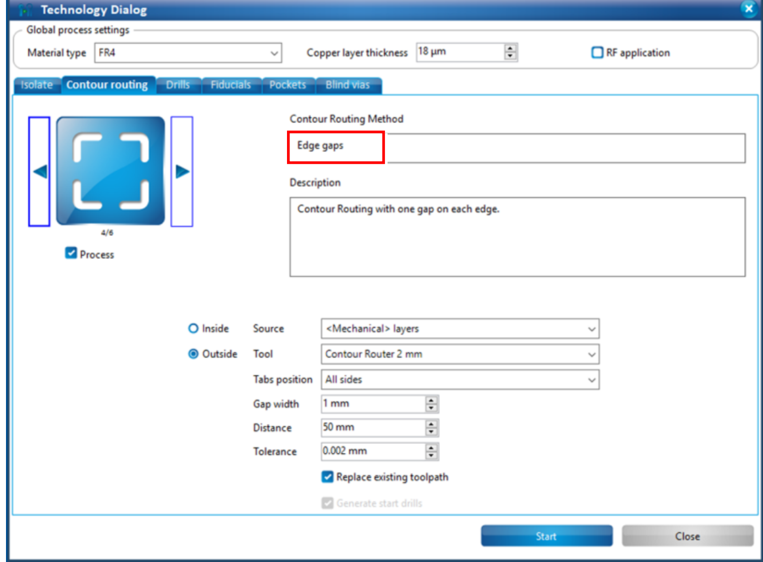


Fig. 120: Tab *Contour routing*

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

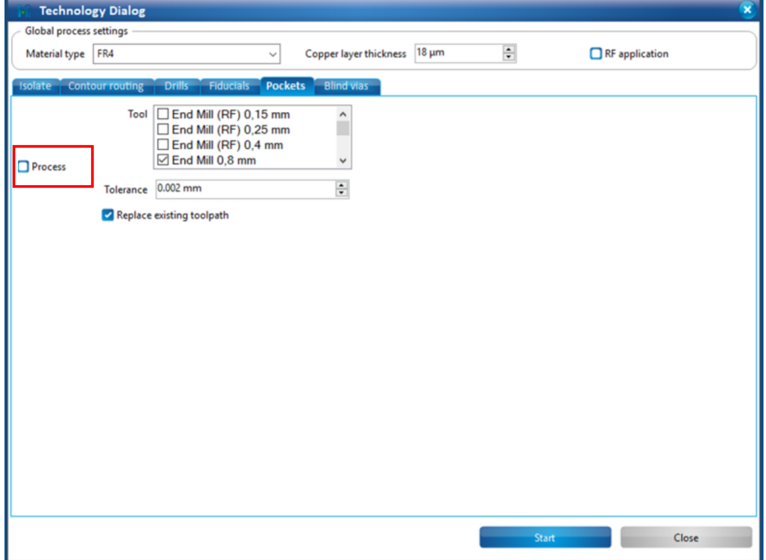


Fig. 121: Tab *Pockets*

4. Click on [Start].

☐ The software generates all toolpaths and identifies all required tools. A report of the required tools is displayed:

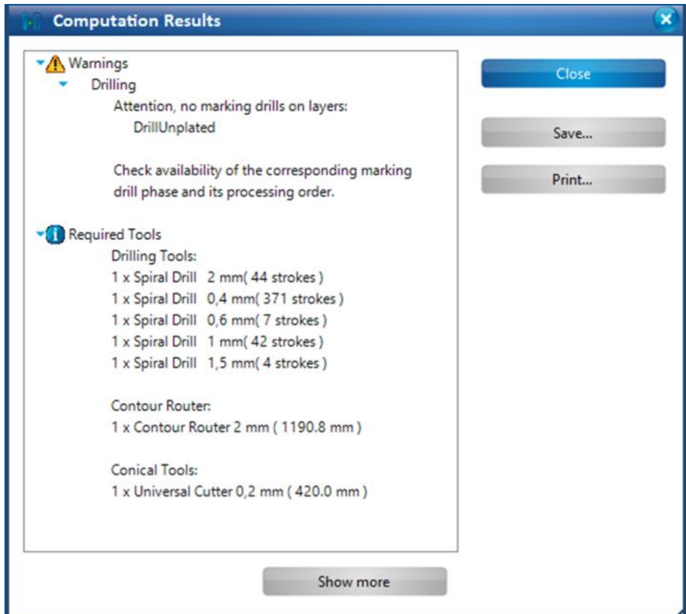


Fig. 122: Message *Computation Results*

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 µm**.

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**.

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...].

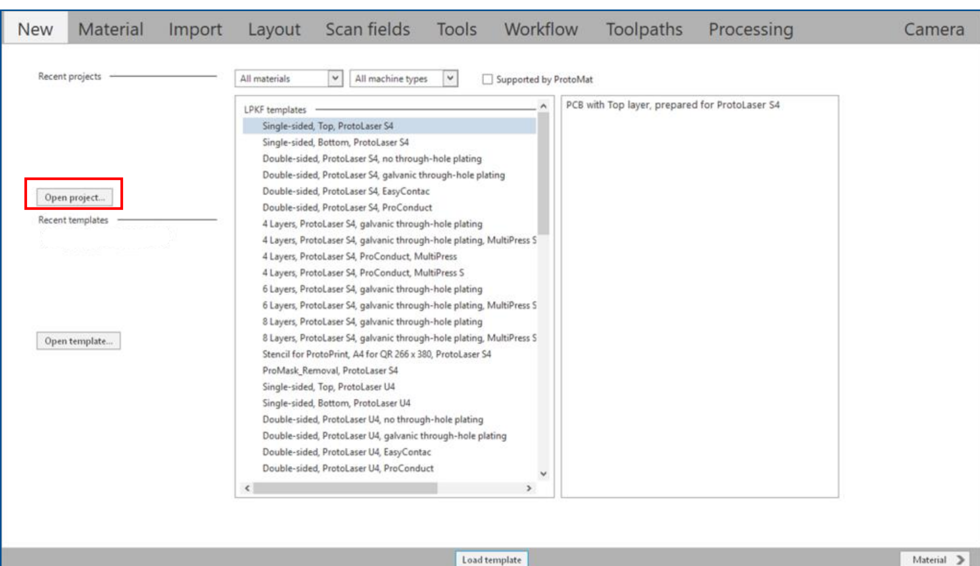


Fig. 123: Open project

☐ 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].

☐ The following dialog is displayed:

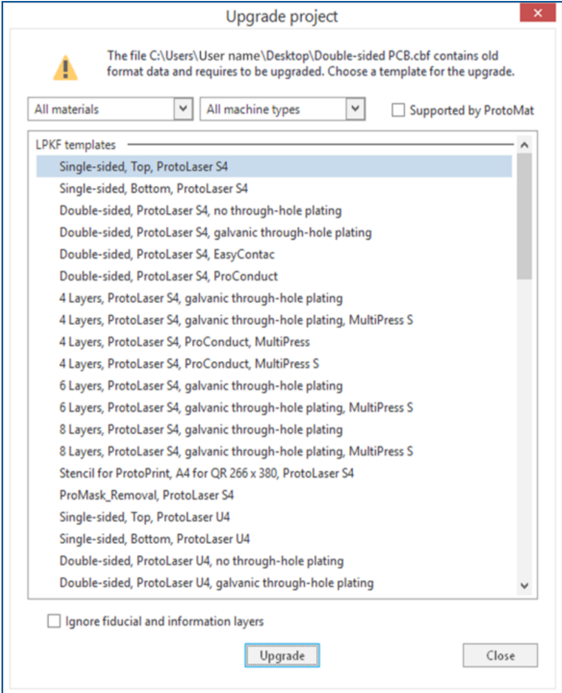


Fig. 124: Dialog *Upgrade project*

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:

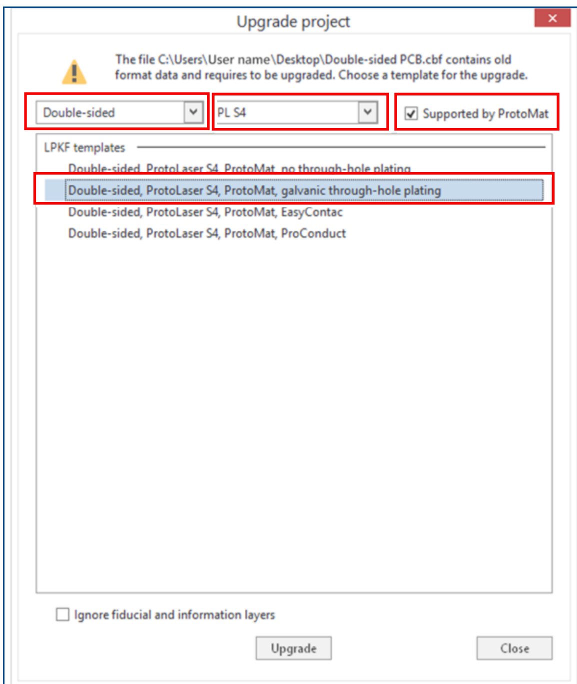


Fig. 125: Upgrade selected

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*.

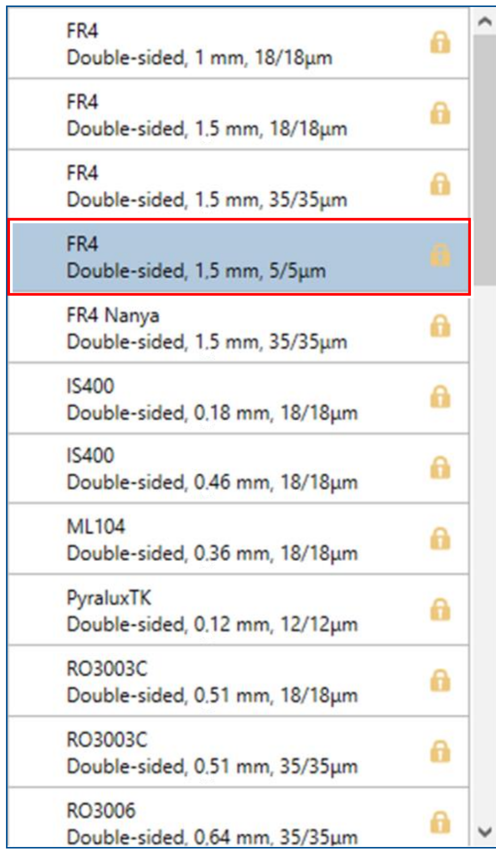


Fig. 126: Material selection

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*.

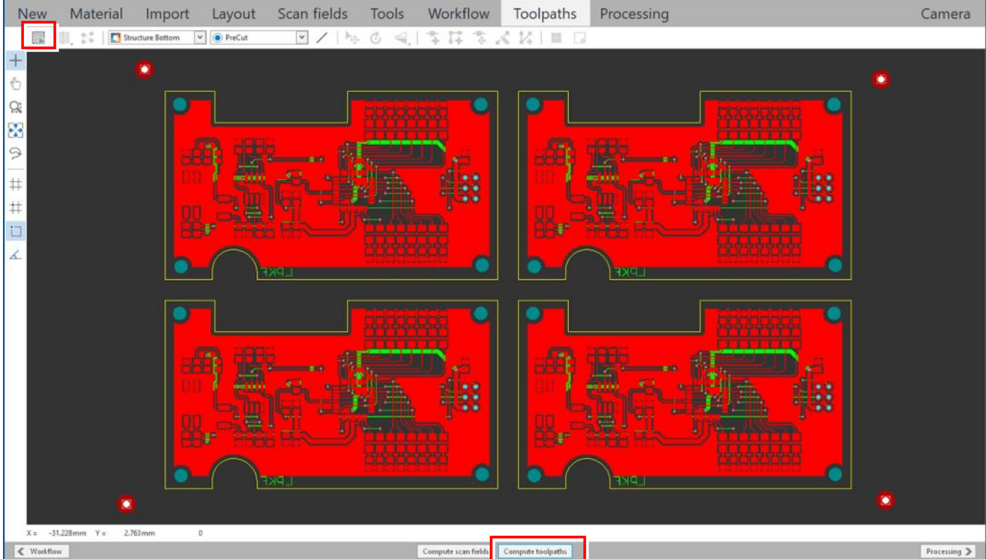


Fig. 127: Opened project in the user guidance step *Toolpaths*

11. Click on [Compute toolpaths] or on .

☐ The following dialog is displayed:

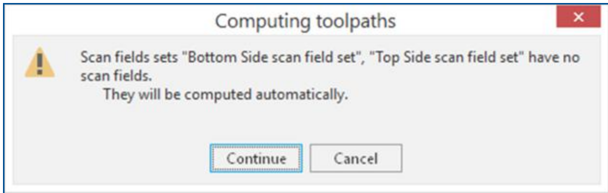


Fig. 128: Dialog *Computing toolpaths*

12. Click on [Continue].

☐ 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. Click on *File* > *Save* or on .
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*.
2. Click on ▶ or click on [Start production].

☐ The following message is displayed:

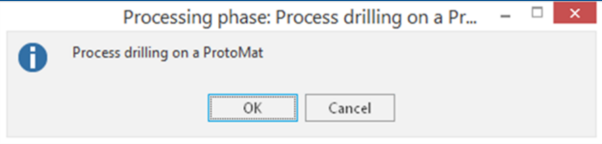


Fig. 129: Message *Processing phase: Process drilling on a ProtoMat*

3. Click on [OK].

☐ The following message is displayed:

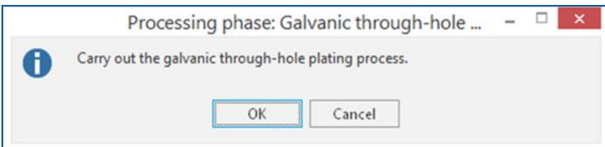


Fig. 130: Message *Processing phase: Galvanic through-hole plating*

4. Click on [OK].

☐ The following message is displayed:

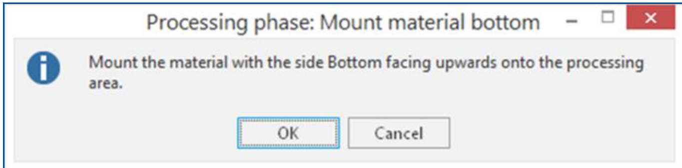


Fig. 131: Message *Processing phase: Mount material bottom*

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:

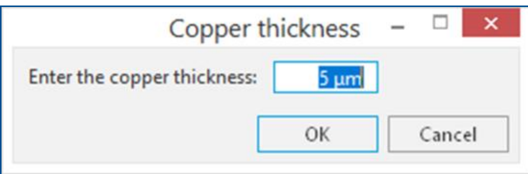


Fig. 132: Dialog *Copper thickness*

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:

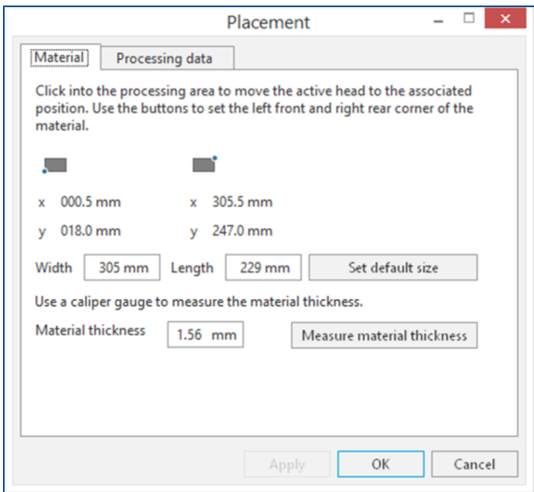


Fig. 133: Dialog *Placement*

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:

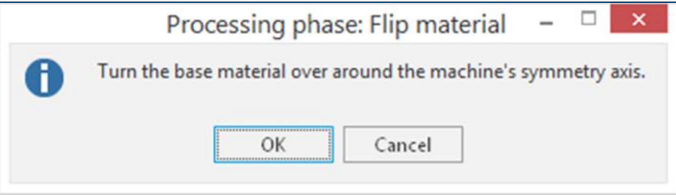


Fig. 134: Message *Processing phase: Flip material*

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:

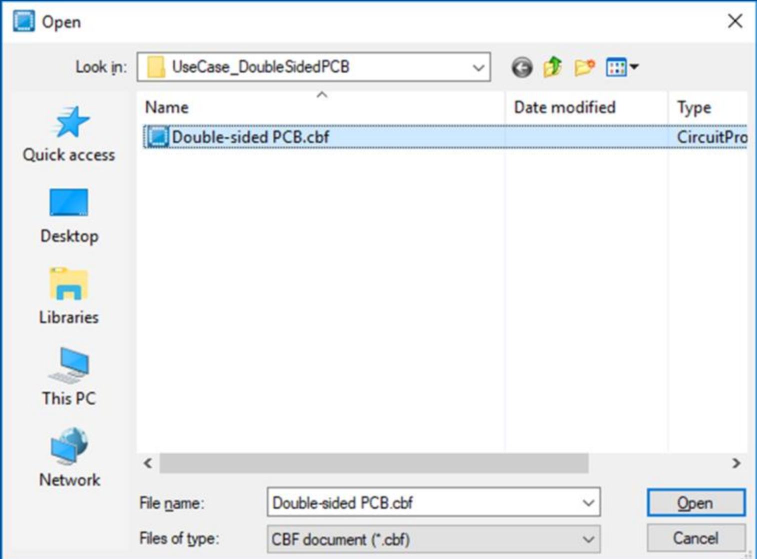


Fig. 135: Dialog *Open*

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*:

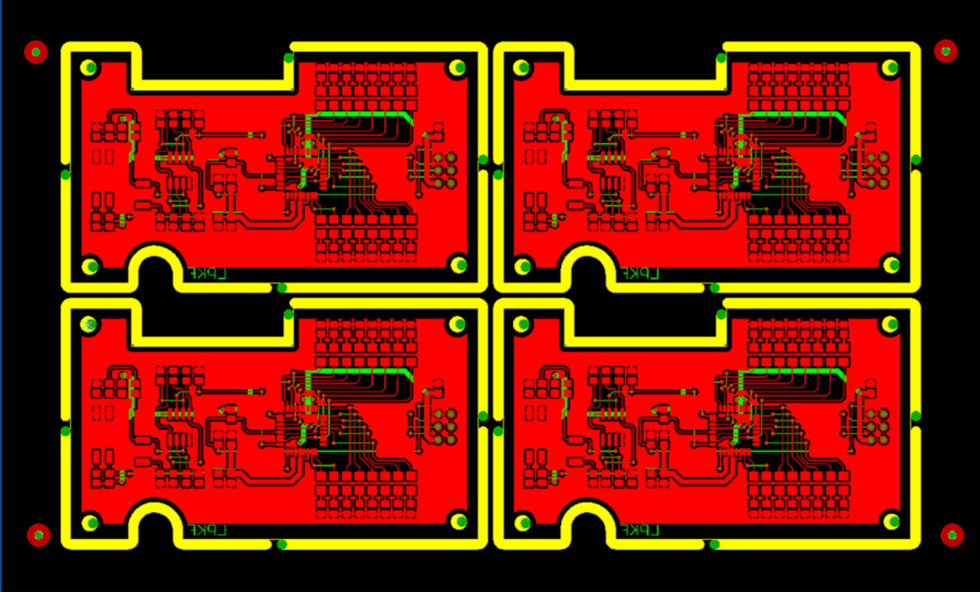


Fig. 136: *CAM view* of the opened project

4. Switch to the *Machining view*.

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

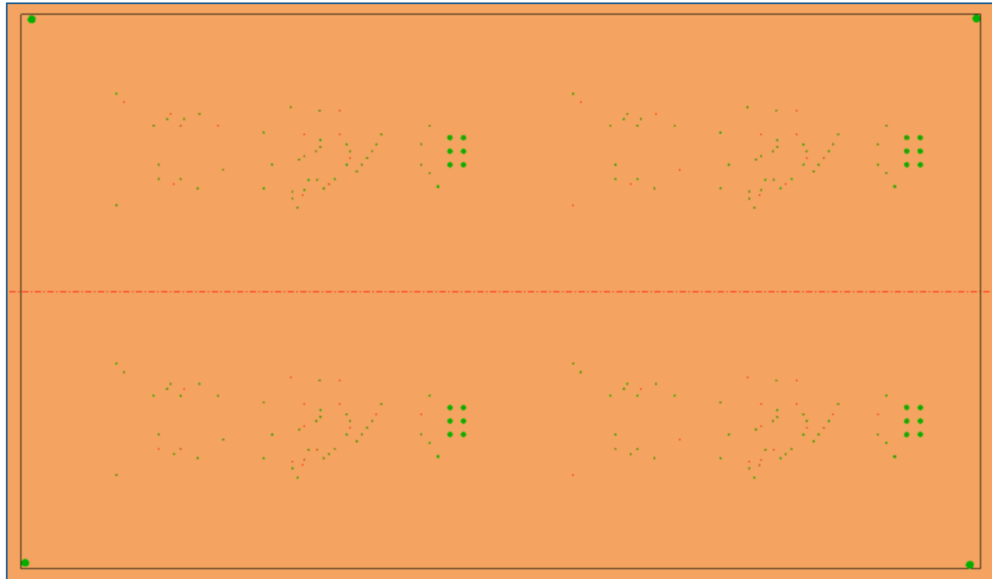


Fig. 137: *Machining view* of the opened project

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

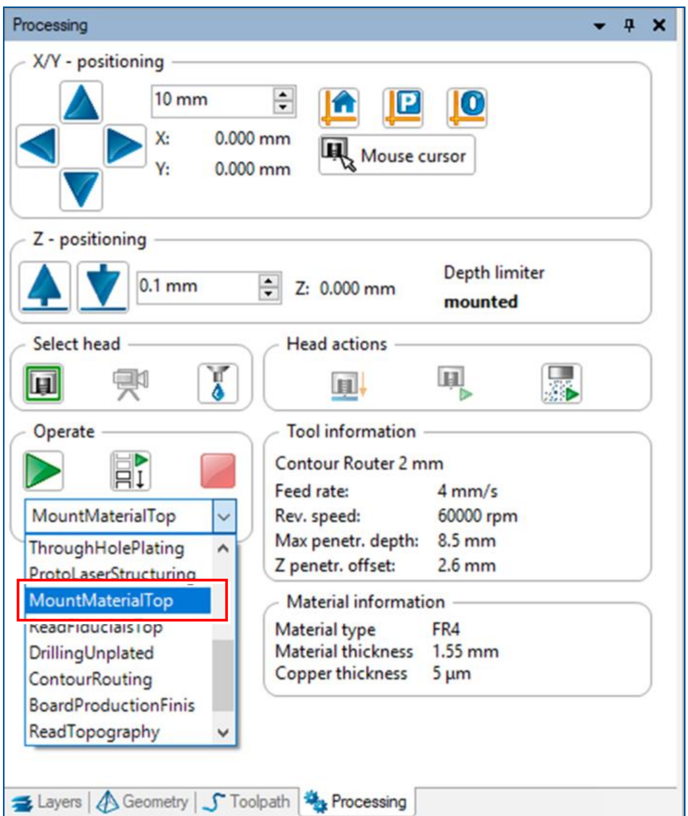


Fig. 138: Drop-down list of the processing phases

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

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

6. Click on .

☐ 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:

Consumables

- Base material FR4, 229 mm × 305 mm × 1.5 mm, 18/18 µm copper-plated (order code: 115967)

Auxiliaries

- 1 set of tools for ProtoMat
- Convection oven (order code: 115877)
- LPKF Cleaner (order code: 115891)
- Brush
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4/R4
- ProtoMat S or E
- ProConduct

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 .
    - ☐ 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*.

- The view changes as follows:

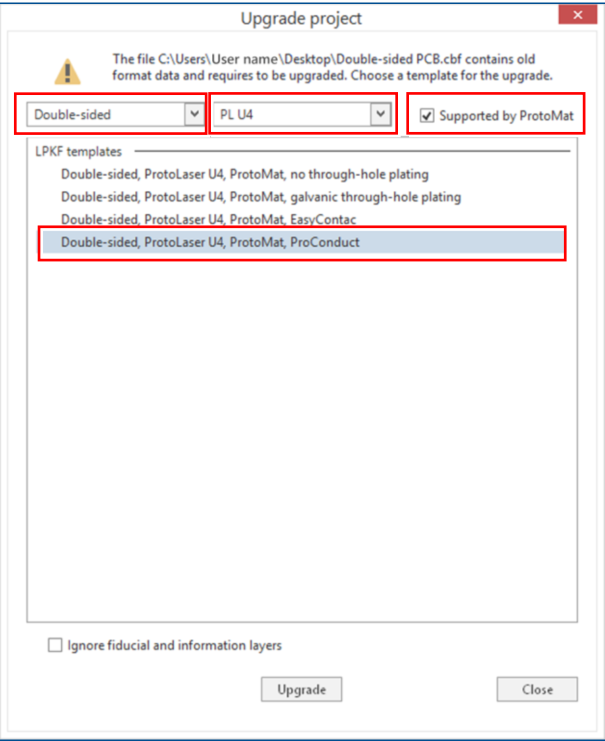


Fig. 140: Upgrade selected

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 or click on [Start production].
3. 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 .
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 °C / 320 °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.

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	System
• 1 × Pyralux® CG185018E	• LPKF Cleaner (order code: 115891)	• ProtoLaser R4
	• Hand-held microscope	• Contac S4
	• 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:

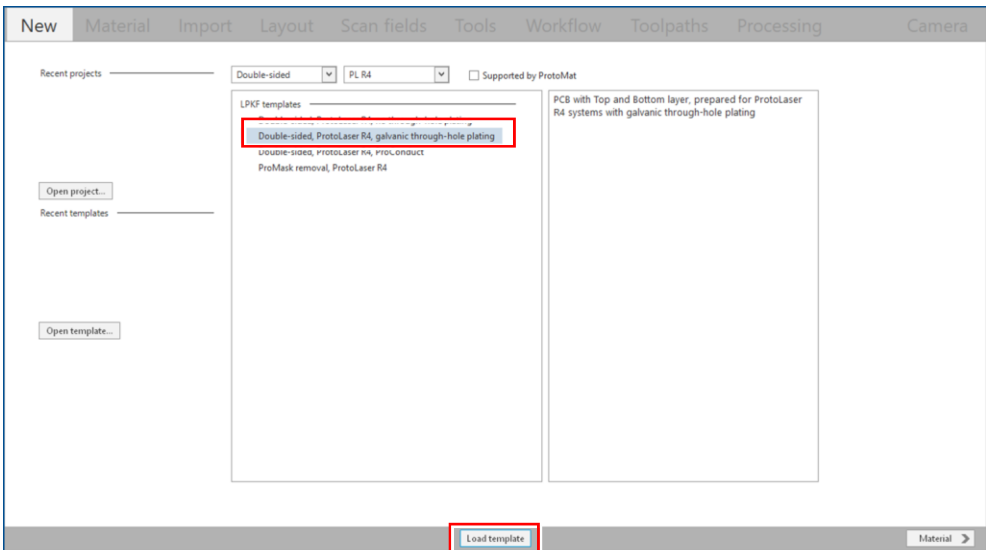


Fig. 141: List of templates

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*.

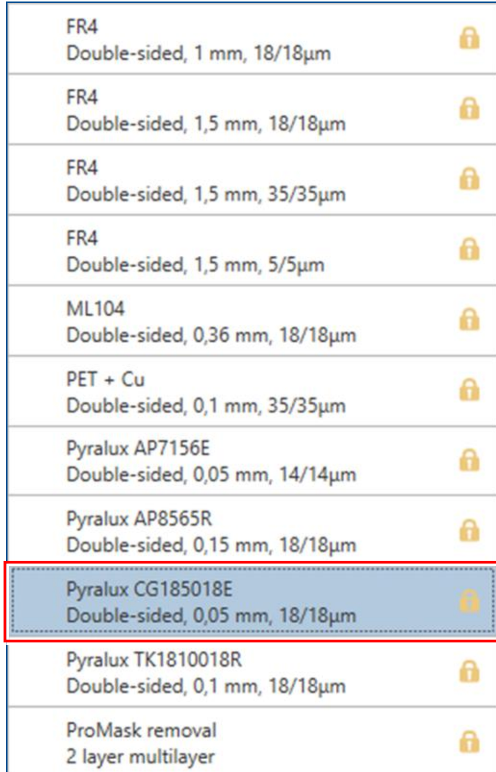


Fig. 142: Material selection

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*:

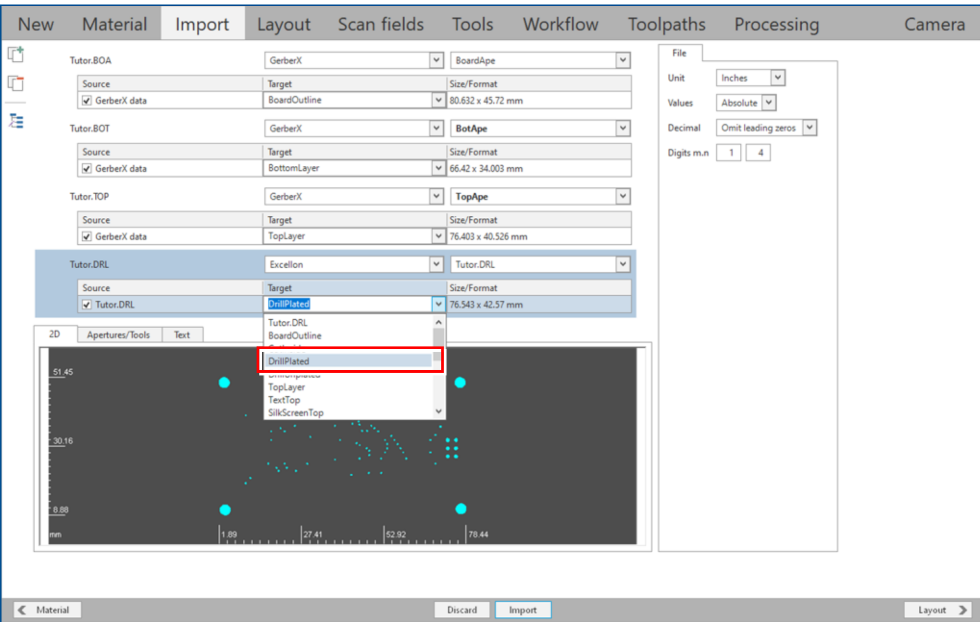


Fig. 143: **User guidance step *Import***

11. Assign the holes to the layer *DrillPlated* (in the column *Target*).
12. Click on [Import].

- ☐ The user guidance step *Layout* is displayed:

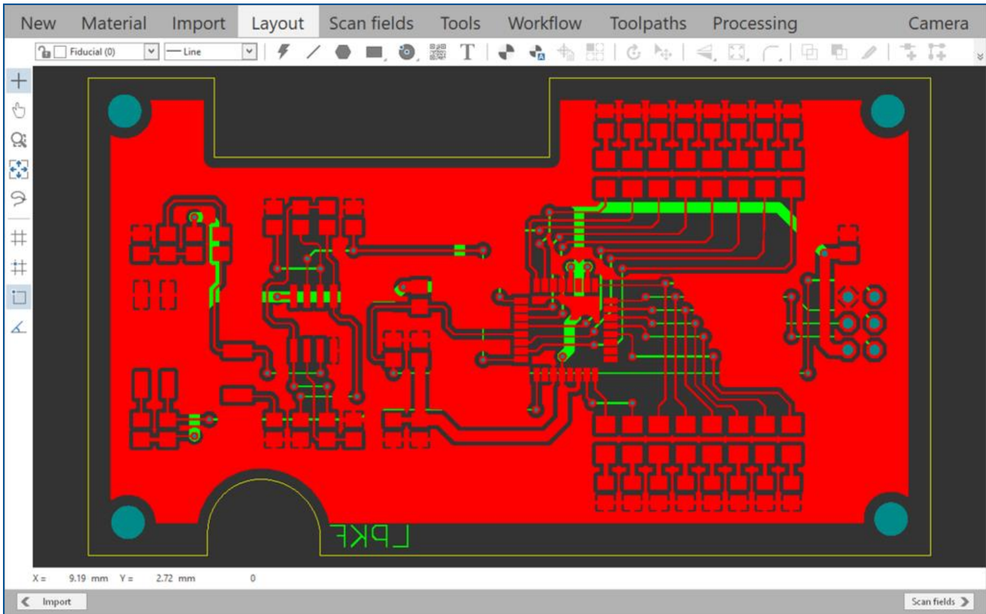


Fig. 144: **User guidance step *Layout***

13. Click on *Insert* > *Automatic fiducial creation* or click on .

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 .

☐ The dialog *Compute scan fields* is displayed.

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*.

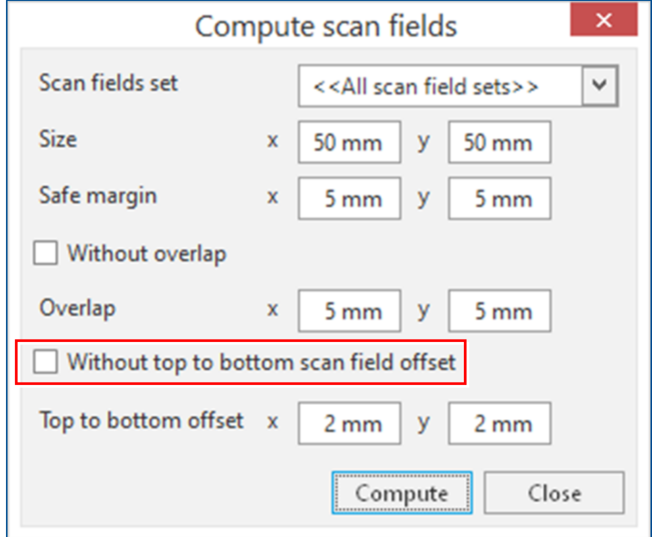


Fig. 145: Dialog *Compute scan fields*

4. Click on [Compute].

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

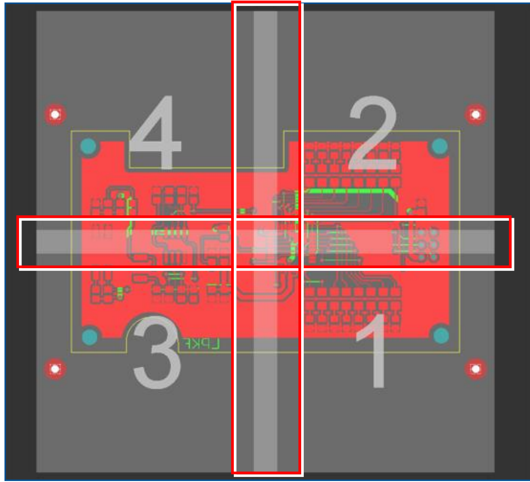


Fig. 146: Scan fields computed | *Overlap* displayed

5. Click on [Compute toolpaths] or on .

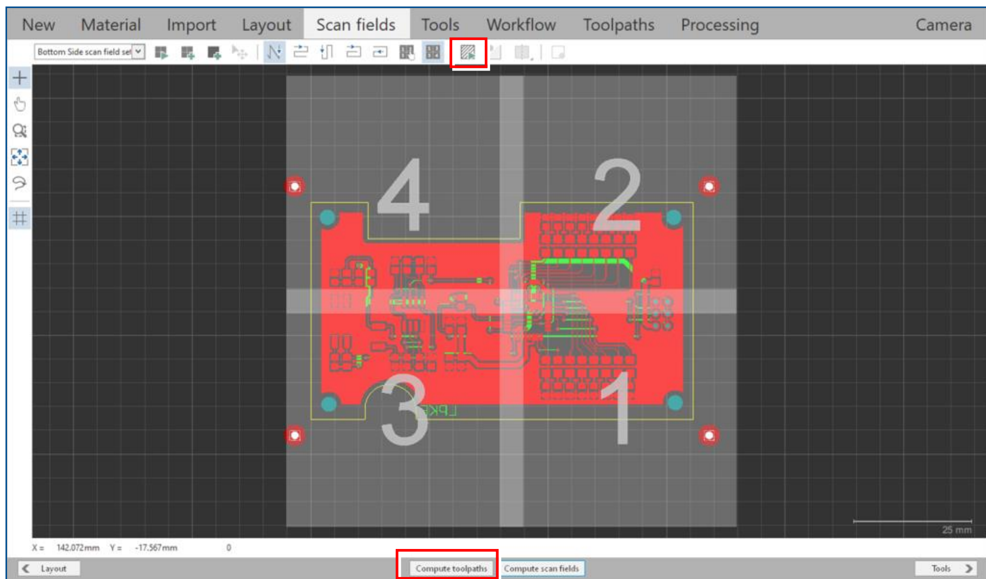


Fig. 147: *Computing toolpaths*

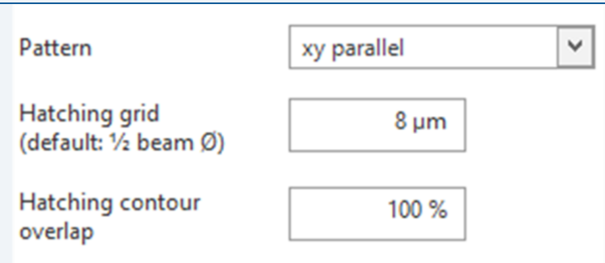
☐ The toolpaths are computed automatically with default settings. The message *Computation results* is displayed.

6. Check the computation results for any possible warnings or errors and make corrections, if needed.
7. Click on [Close].
8. Save your project.

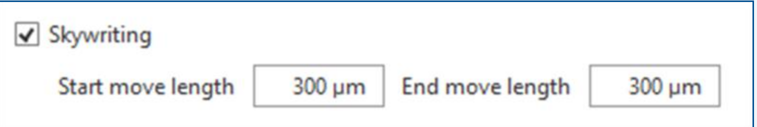
☑ The toolpaths and scan fields have been computed.

Computing toolpaths for processing flexible PCBs

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.



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



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.

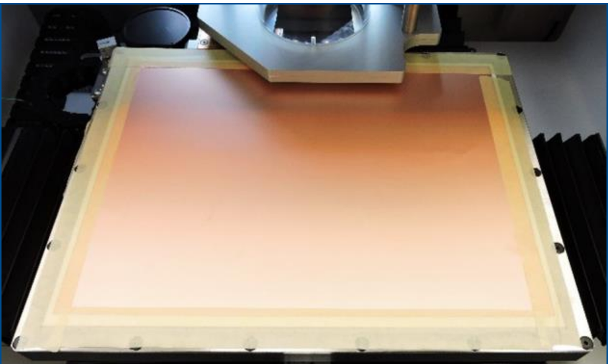


Fig. 148: Flexible material fastened to the processing table

4. Click on .

☐ The flexible material is fastened onto the processing table by vacuum.

5. Click on [OK].

☐ The following dialog is displayed:

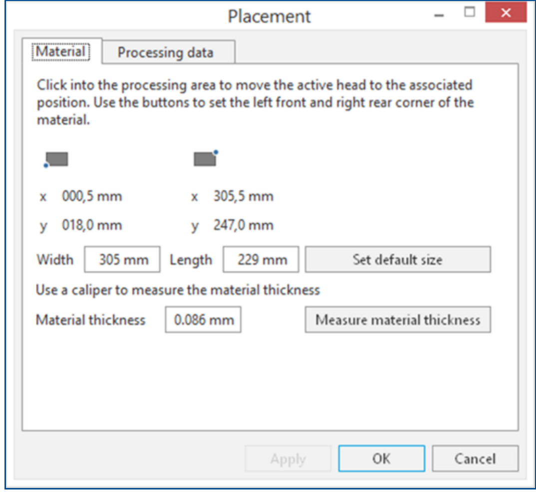


Fig. 149: Dialog *Placement*

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.

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:

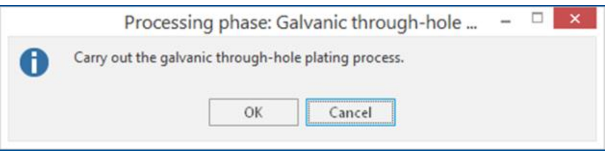


Fig. 150: Message *Processing phase: Galvanic through-hole plating*

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 µm 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*.

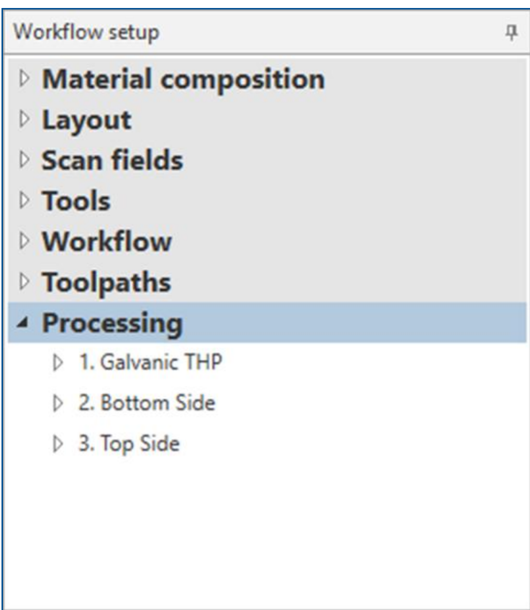


Fig. 153: Pane *Workflow setup*

3. Right-click on the entry *Bottom Side*.

☐ The following context menu is displayed:

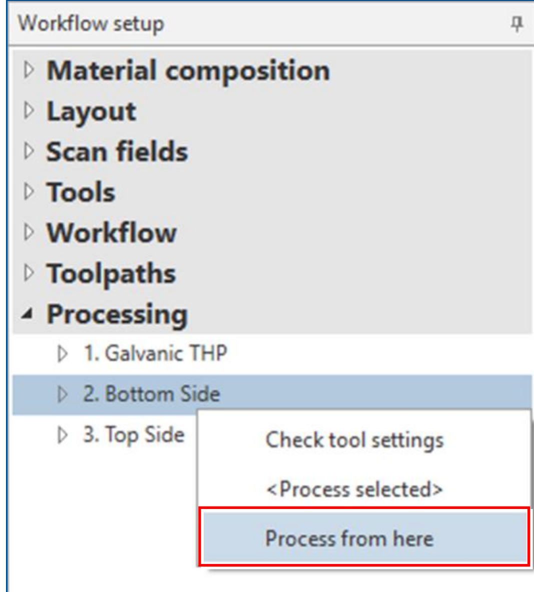


Fig. 154: Context menu *Process from here*

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:

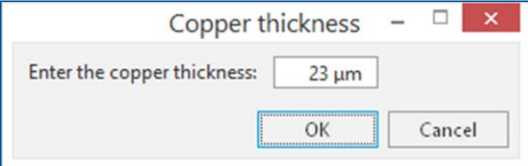


Fig. 155: Dialog *Copper thickness*

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

You can determine the total copper thickness by calculating the thickness from the copper deposition rate in Contac S4 (approx. 0.15 µm/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:

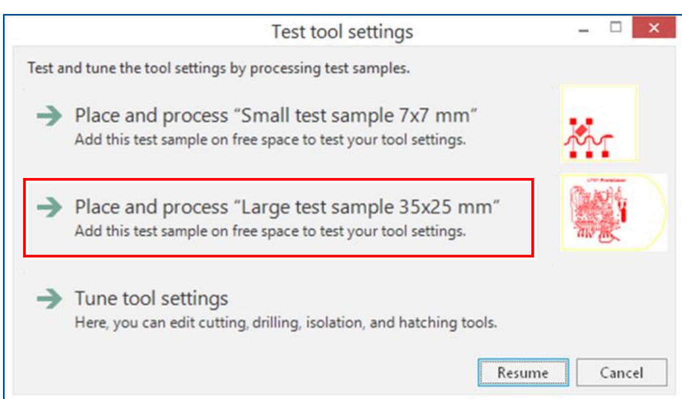


Fig. 156: Dialog *Test tool settings*

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

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

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.

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)
- 4 × Prepreg Type 2125, 200 mm × 275 mm × 0.1 mm (order code: 119572)

Auxiliaries

- Convection oven (order code: 115877)
- LPKF Cleaner (order code: 115891)
- Brush
- Oil-free compressed air
- Tap water

System

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

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:

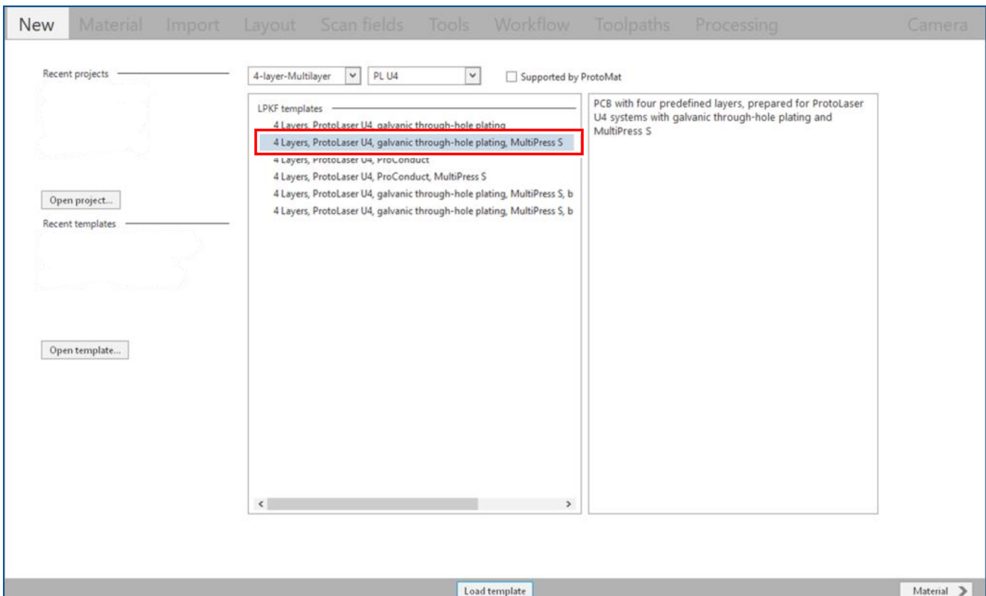


Fig. 157: List of multi-layer templates

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:

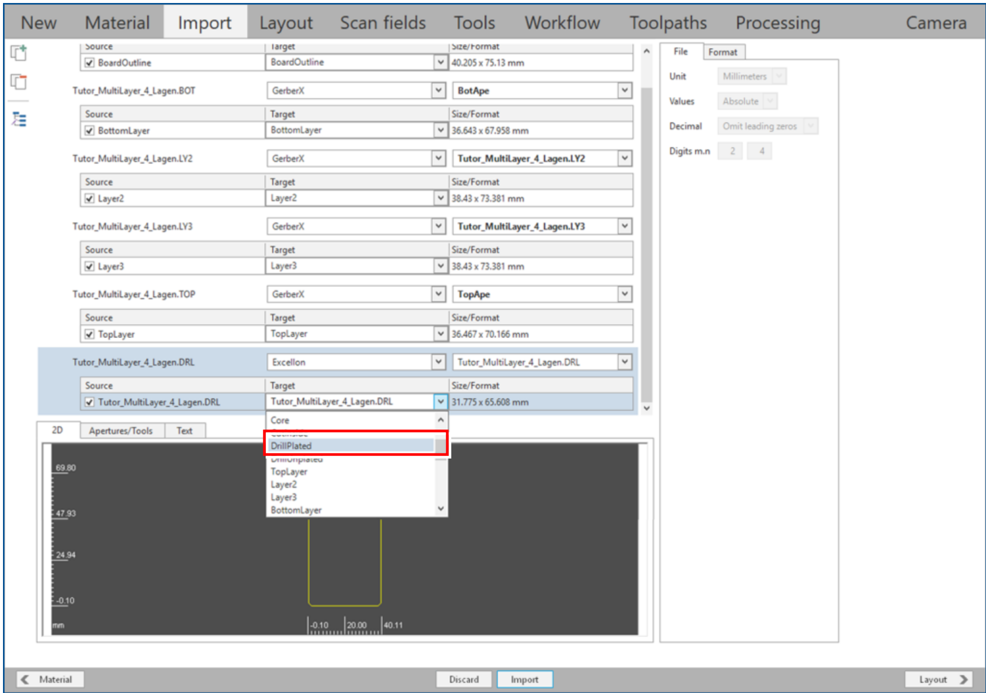


Fig. 159: User guidance step *Import*

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

12. Click on [Import].

☐ The layout is displayed in the user guidance step *Layout*.

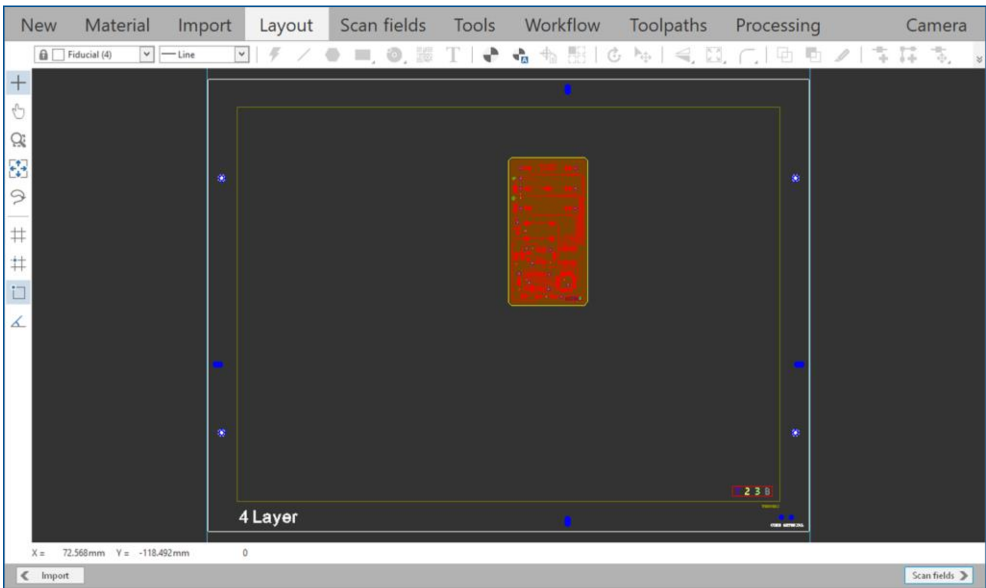


Fig. 160: Layout of a multi-layer PCB

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:

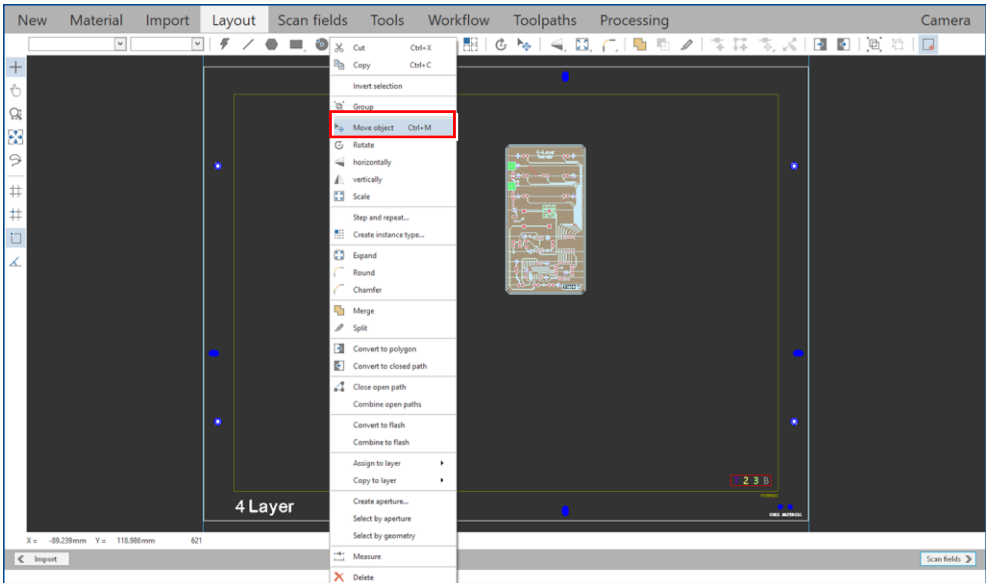


Fig. 161: Context menu *Move object*

16. Select *Move object* or click on .

- 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:

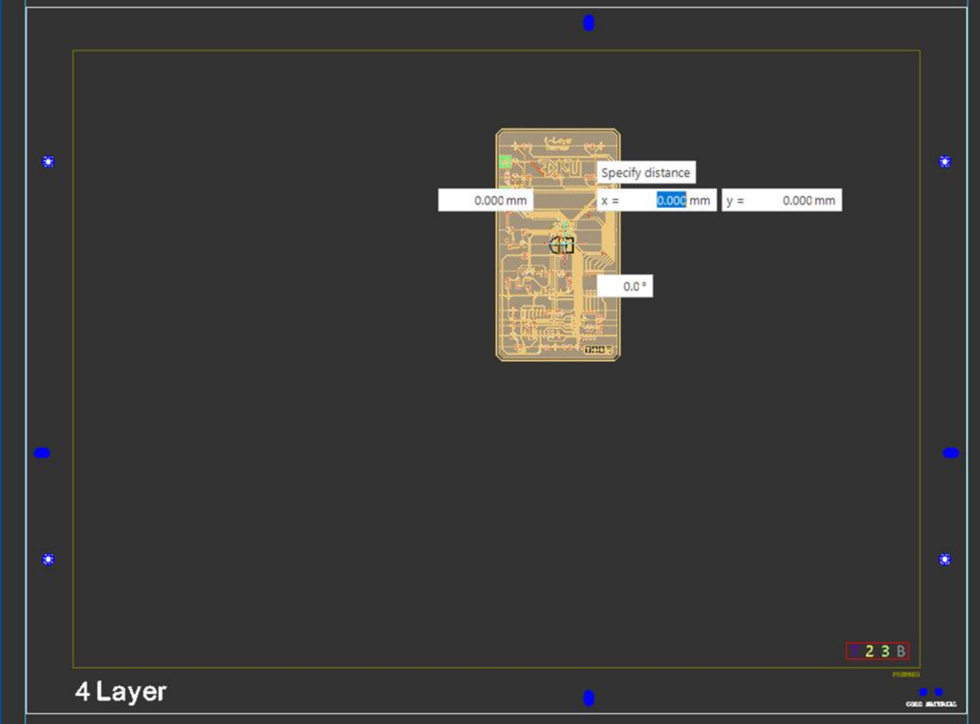


Fig. 163: ***Specify distance***

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

Fig. 164: **Desired position of the copied layout**

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:

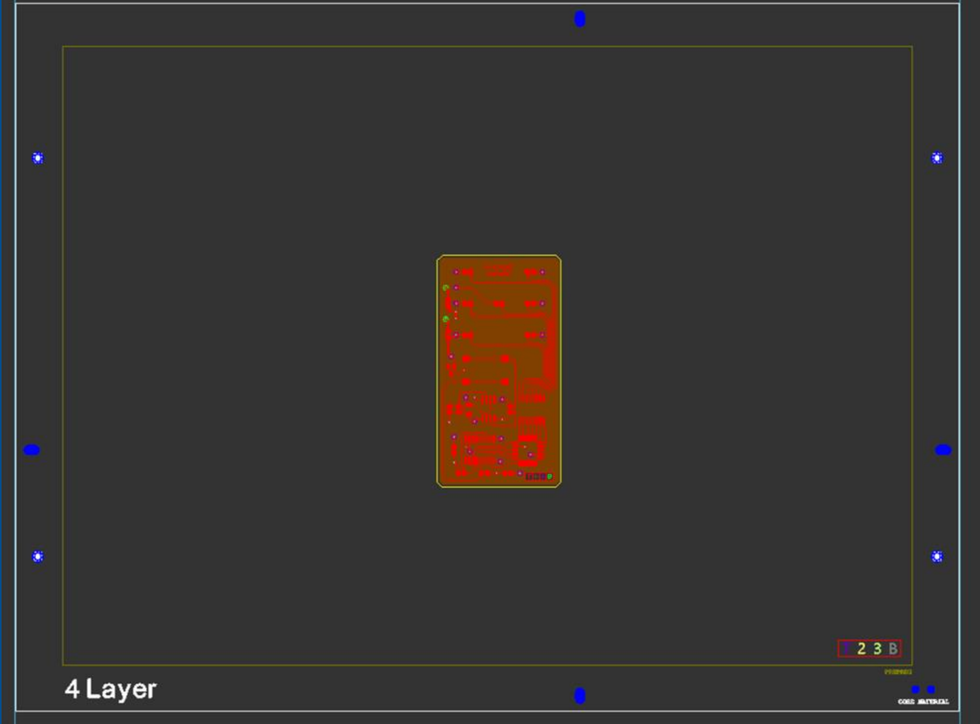


Fig. 165: **Layout moved to the desired position**

22. Switch to the user guidance step *Toolpaths*.

Fig. 166: Opened project

23. Click on [Compute toolpaths] or on .

☐ The following dialog is displayed:

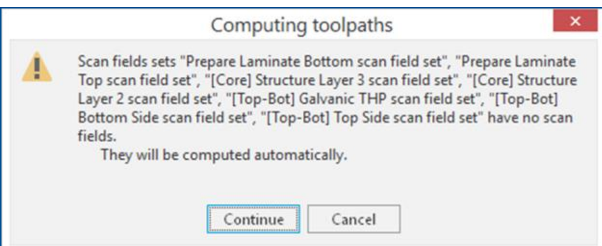


Fig. 167: Dialog *Computing toolpaths*

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*.

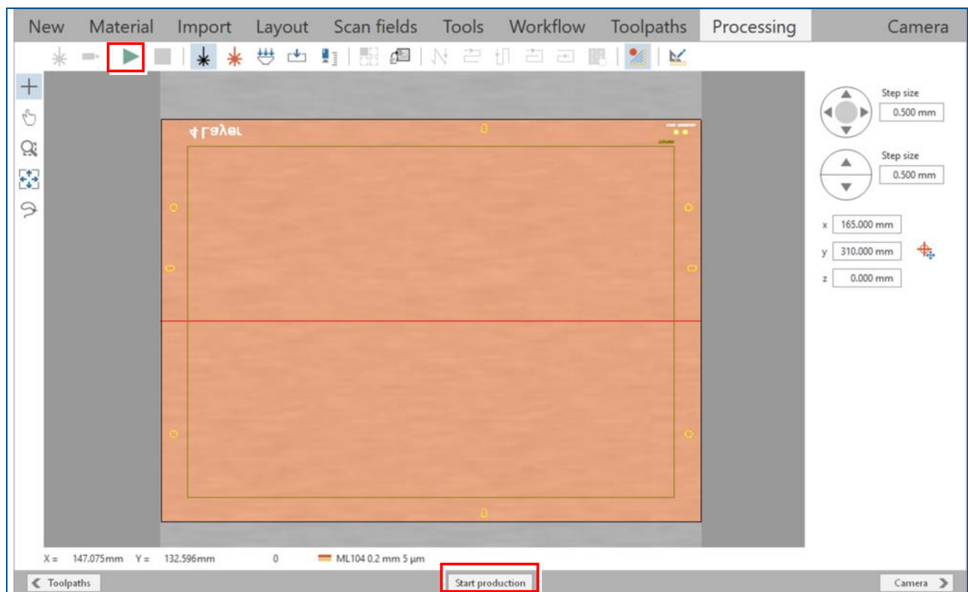


Fig. 168: User guidance step *Processing*

2. Click on [Start production] or on ▶.

☐ The following message is displayed:

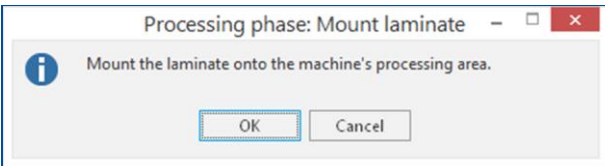


Fig. 169: Message *Processing phase: Mount laminate*

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 and fasten the laminate material along its edges with adhesive tape.

4. Click on [OK].

☐ The following dialog is displayed.

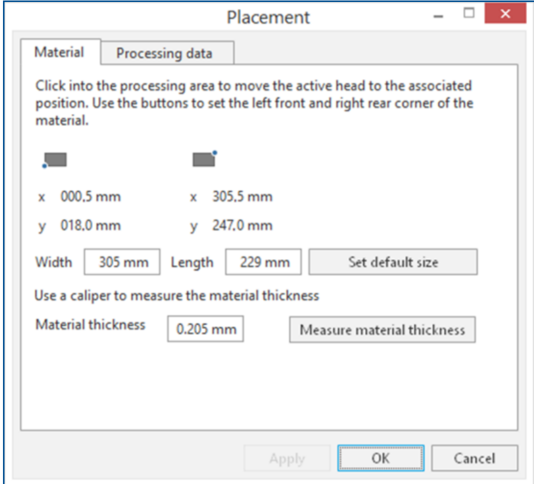


Fig. 170: Dialog *Placement*

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 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.

Structuring the core material (with ProtoLaser)

1. In the message *Processing phase: Dismount material* click on [OK].

☐ The following message is displayed:

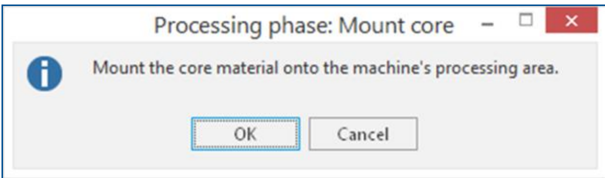


Fig. 171: Message *Processing phase: Mount core*

2. Place the core material in the center of the processing table.
3. Click on .

☐ The core material is fastened onto 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.

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.

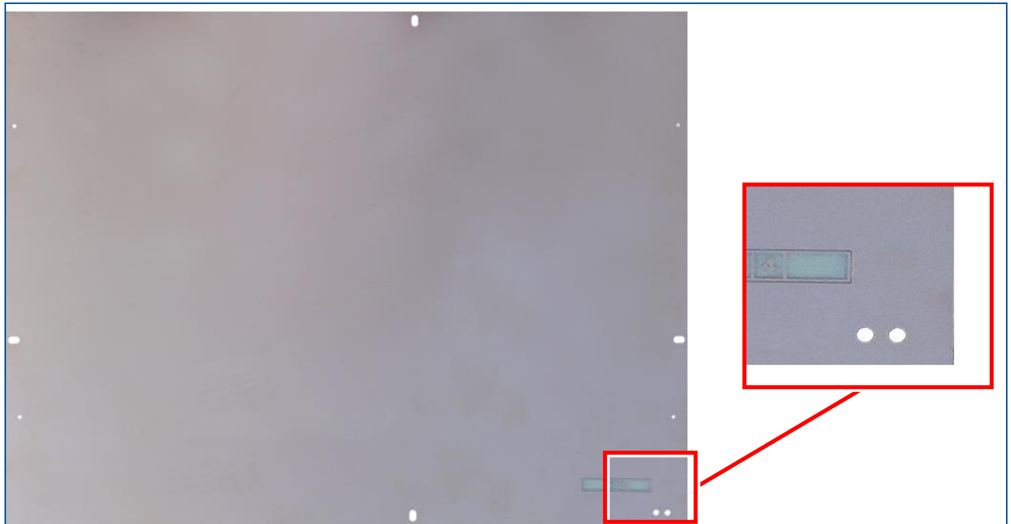
10. Click on [Resume].

☐ 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.

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.

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:

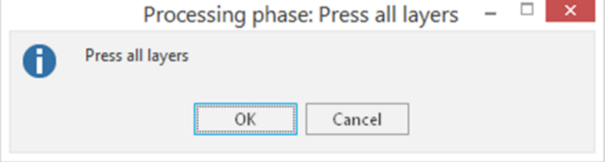


Fig. 172: Message *Processing phase: Press all layers*

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 °C (212 °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*.

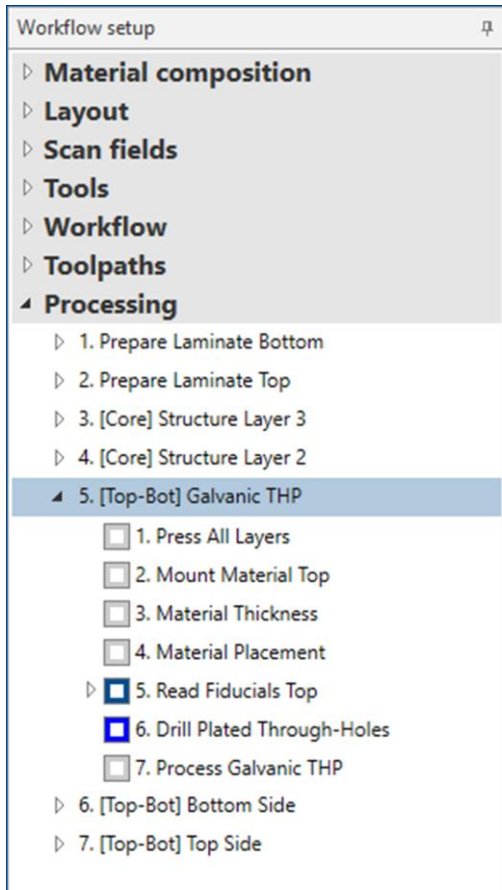


Fig. 173: Pane *Workflow setup*

5. Right-click on *Mount Material Top*.

☐ The following context menu is displayed:

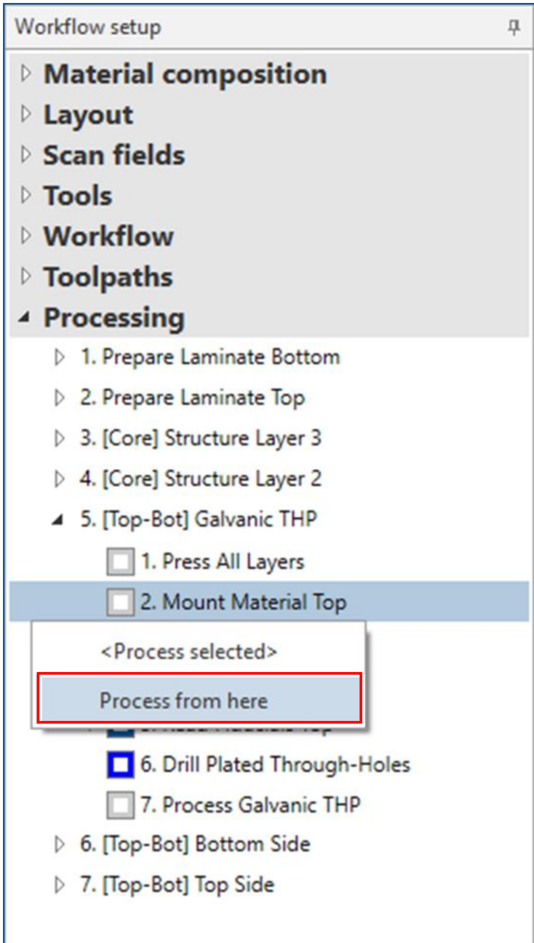


Fig. 174: Context menu *Process from here*

6. Select *Process from here*.

☐ The following message is displayed:

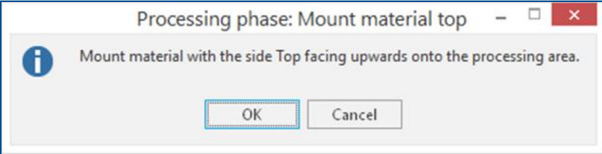


Fig. 175: Message *Processing phase: Mount material top*

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:

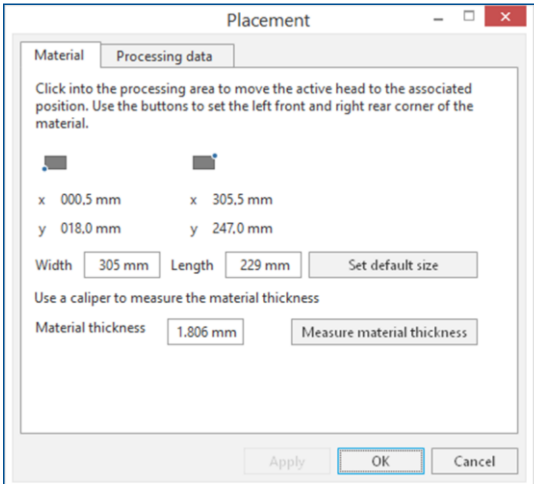


Fig. 176: Dialog *Placement*

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:

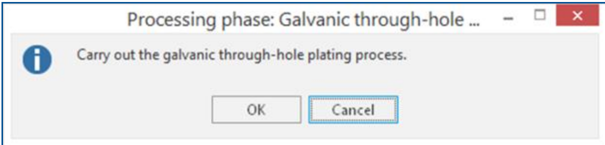


Fig. 177: Message *Processing phase: Galvanic through-hole plating*

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 µm**.

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:

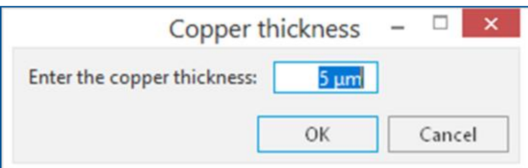


Fig. 178: Dialog *Copper thickness*

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 µm/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.

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].

☐ 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.

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)
- 4 × Prepreg Type 2125, 200 mm × 275 mm × 0.1 mm (order code: 119572)

Auxiliaries

- 1 set of tools for ProtoMat
- Convection oven (order code: 115877)
- LPKF Cleaner (order code: 115891)
- Brush
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4/R4
- MultiPress S
- ProtoMat S or E
- Contac S4

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 .
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:

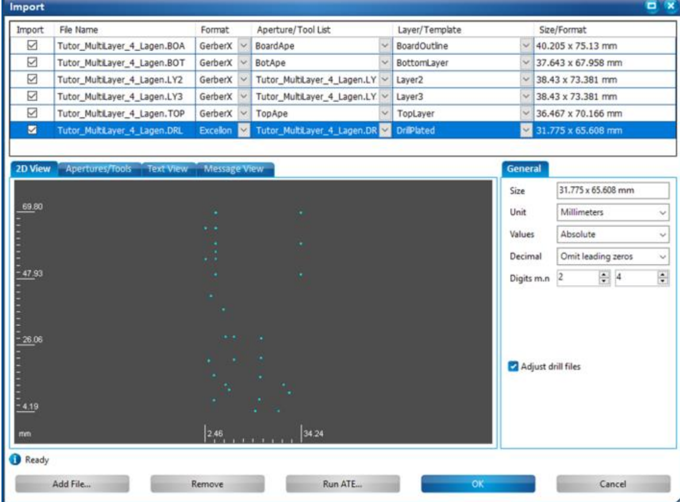


Fig. 179: Dialog *Import* | Assigned layers

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*:

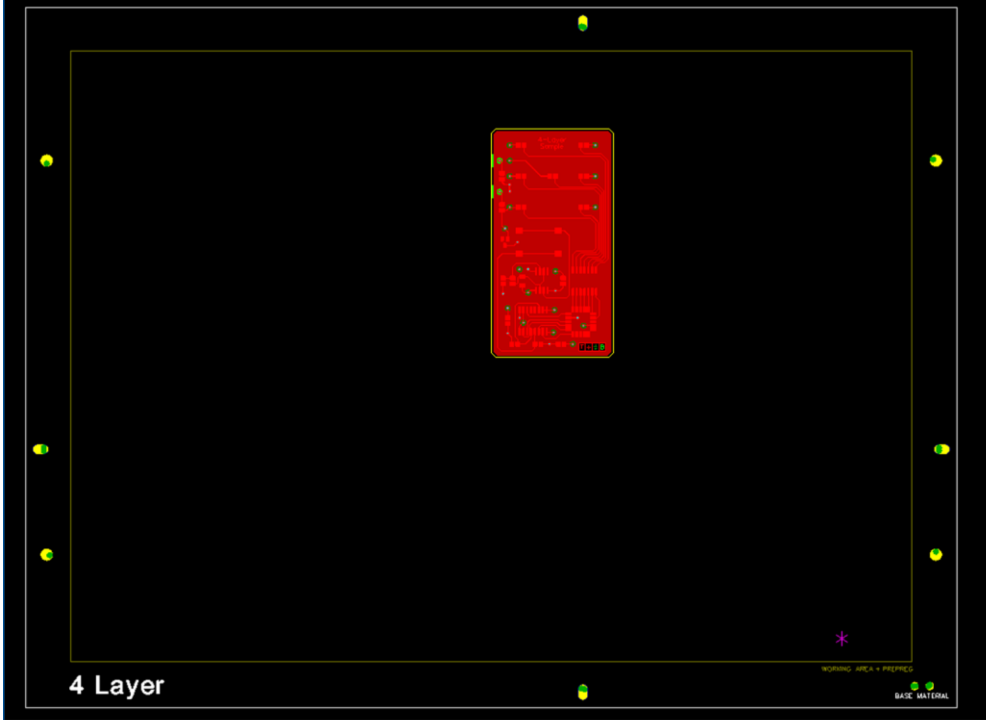


Fig. 180: *CAM view* multi-layer PCB

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 Ctrl 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 the toolbar icon.

☐ 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

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:

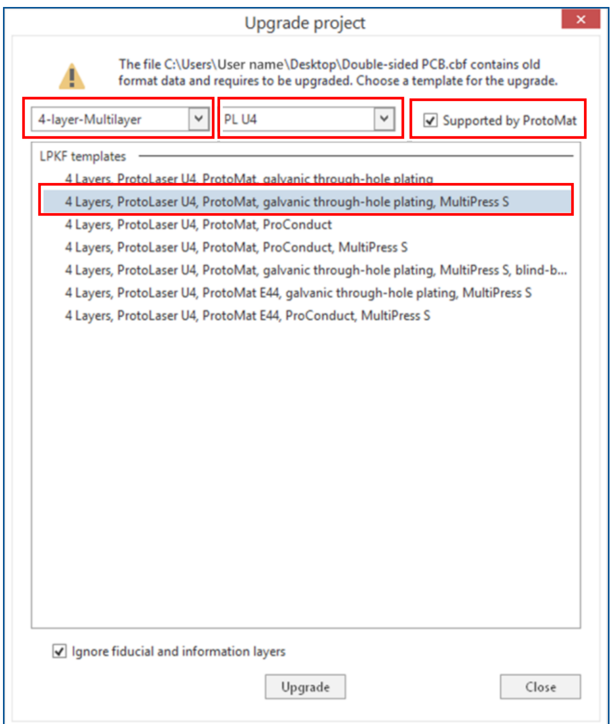


Fig. 181: Upgrade selected

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*.

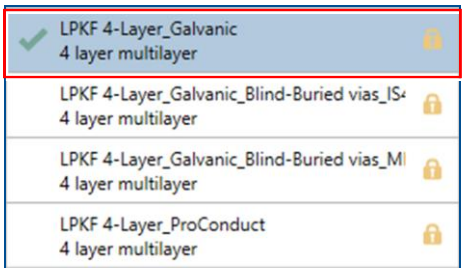


Fig. 182: Material selection

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*.

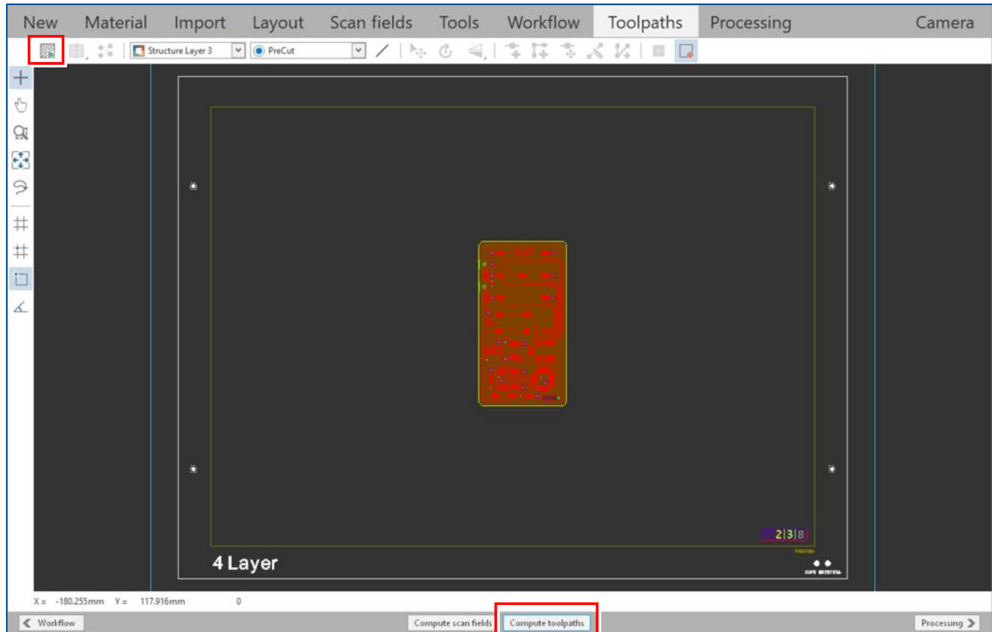


Fig. 183: Opened project

11. Click on [Compute toolpaths] or on .

☐ The following dialog is displayed:

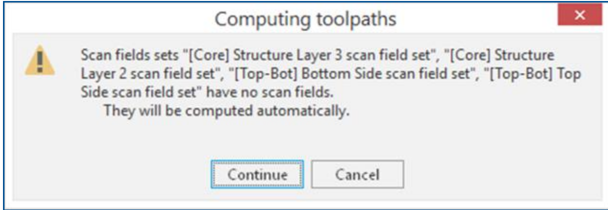


Fig. 184: Dialog *Computing toolpaths*

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)

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

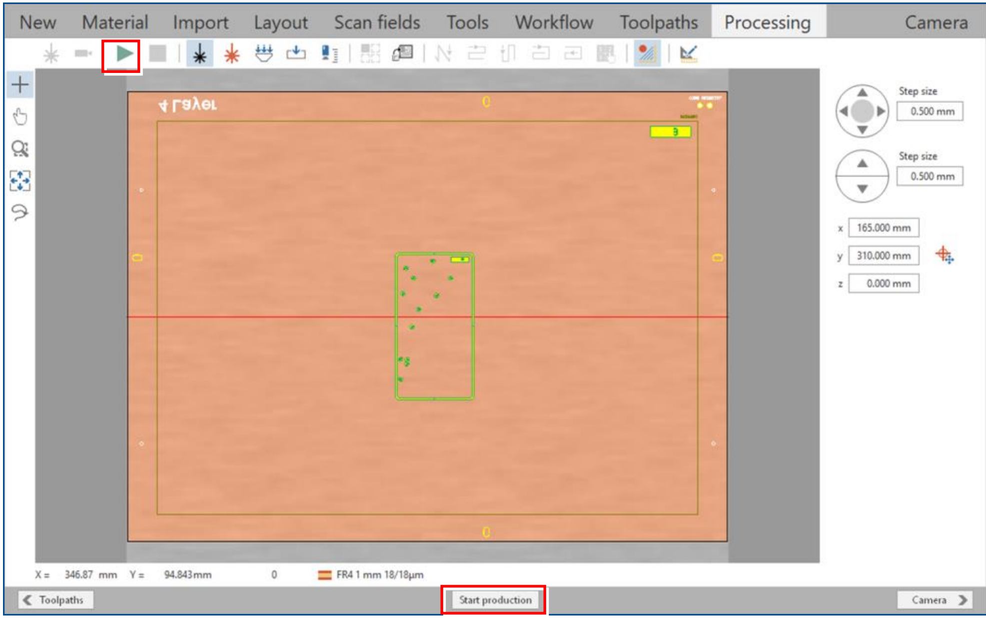


Fig. 185: User guidance step *Processing*

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

☐ The following message is displayed:

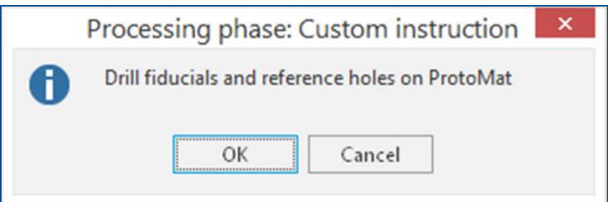


Fig. 186: Message *Processing phase: Custom instruction*

3. Click on [OK].

☐ The following message is displayed:

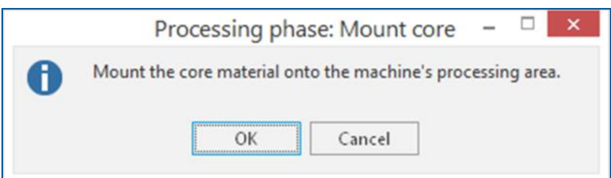


Fig. 187: Message *Processing phase: Mount core*

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:

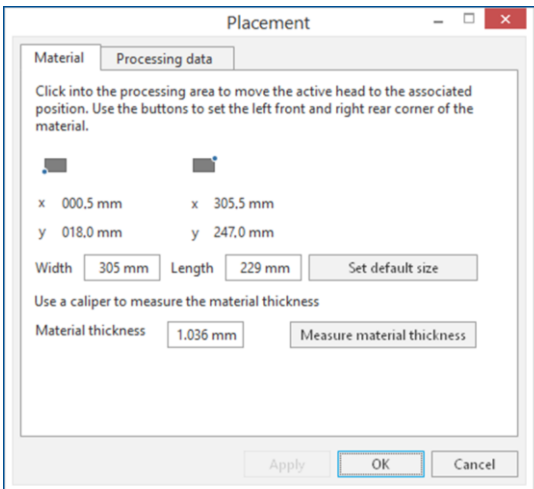


Fig. 188: Dialog *Placement*

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*).

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 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:

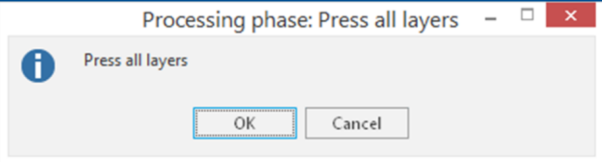


Fig. 189: Message *Processing phase: Press all layers*

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 °C (212 °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.

4. 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* 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 µm to 35 µm**.

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*.

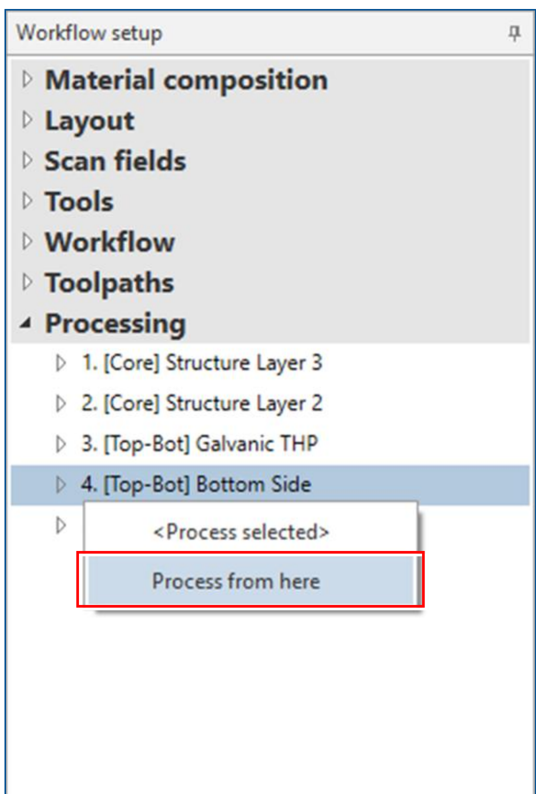


Fig. 190: Context menu *Process from here*

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:

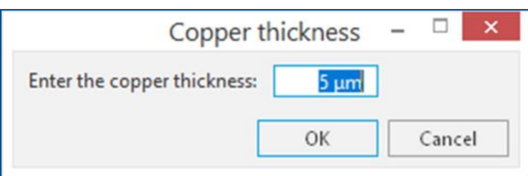


Fig. 191: Dialog *Copper thickness*

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:

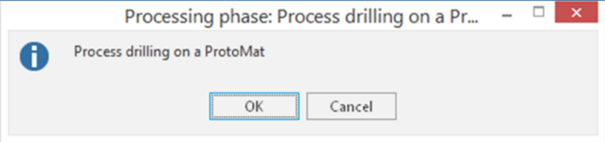


Fig. 192: Message *Processing phase: Process drilling on a ProtoMat*

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 .

☐ The message *Processing Phase: MountMaterialTop_1* is displayed.

4. 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

- 1 × Base material FR4 229 mm × 305 mm × 1 mm, 18/18 µm copper-clad (order code: 119574)
- 2 × Thin Laminate 104 ML 0/18 µm, 229 mm × 305 mm × 0.2 mm (order code: 119818)
- 4 × Prepreg Type 2125, 200 mm × 275 mm × 0.1 mm (order code: 119572)

Auxiliaries

- 1 set of tools for ProtoMat
- Convection oven (order code: 115877)
- LPKF Cleaner (order code: 115891)
- Brush
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4/R4
- MultiPress S
- ProtoMat S or E
- ProConduct

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 .
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:

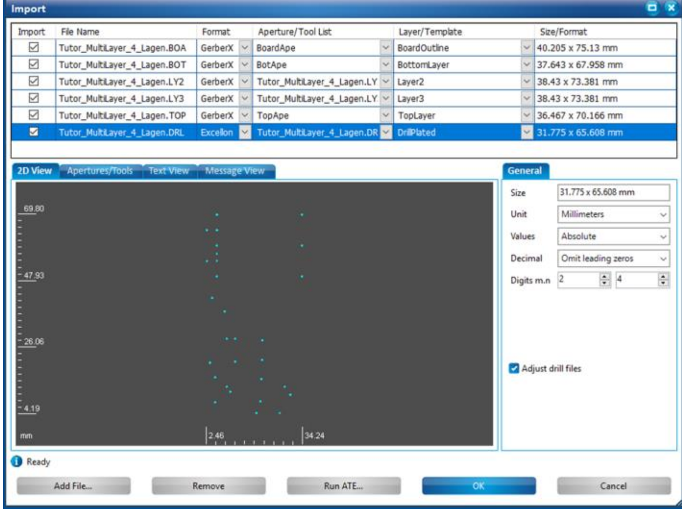


Fig. 193: Dialog *Import* | Assigned layers

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*:

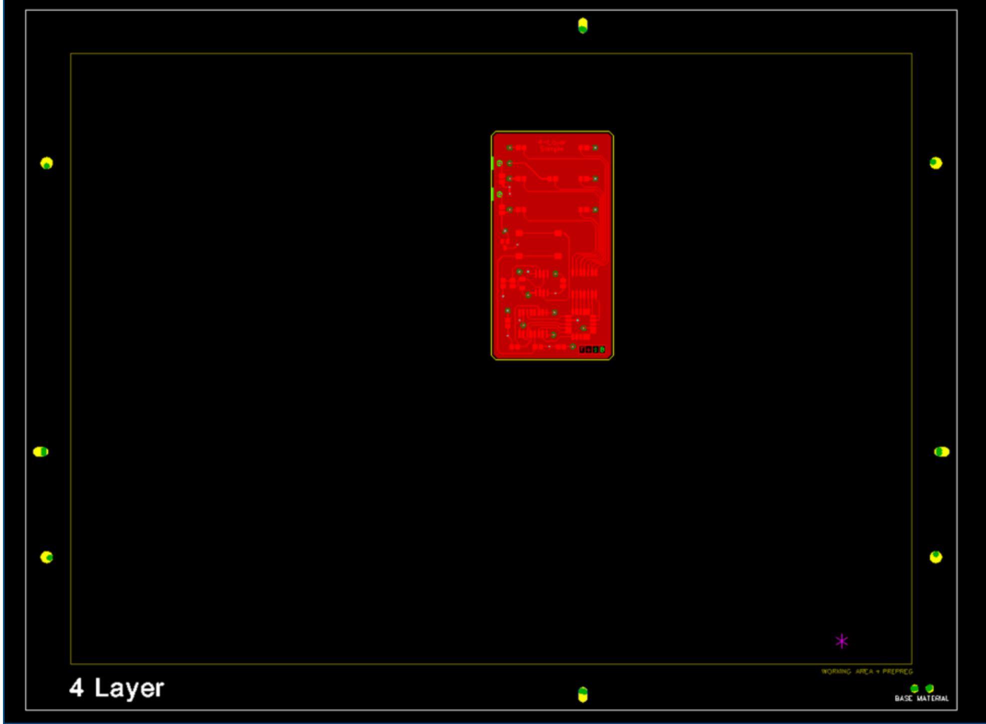


Fig. 194: *CAM view* multi-layer PCB

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 Ctrl 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 .
- ☐ 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

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:

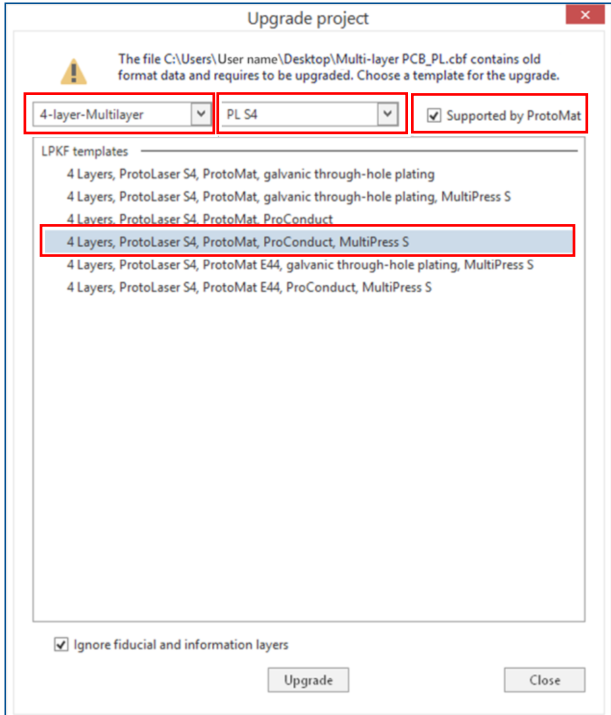


Fig. 195: Upgrade selected

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*.

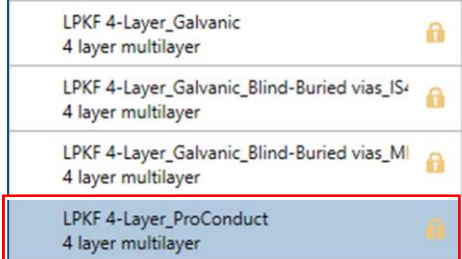


Fig. 196: Material selection

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*.

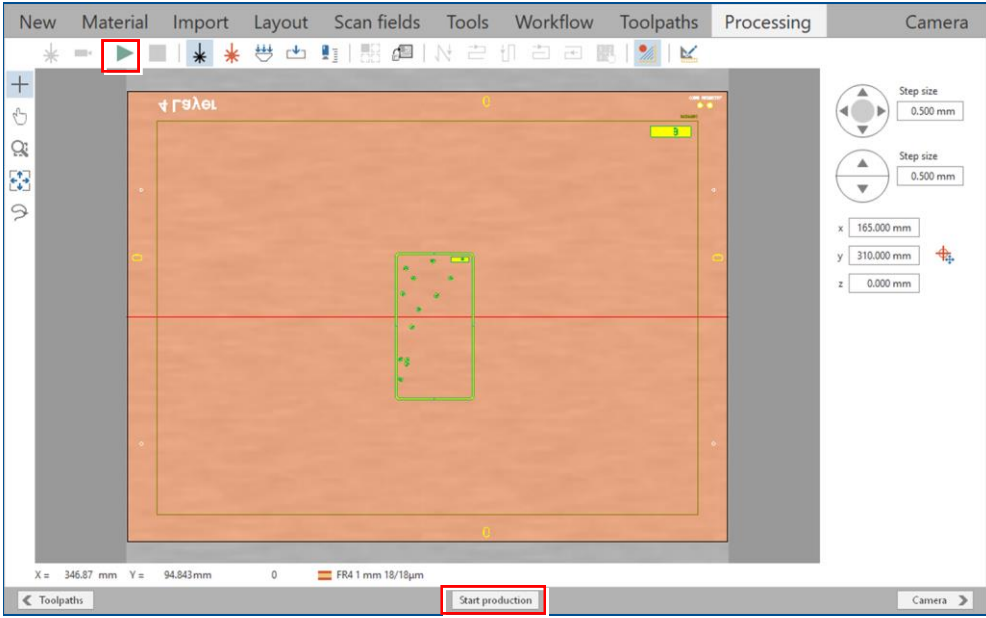


Fig. 197: User guidance step *Processing*

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

- The following message is displayed:

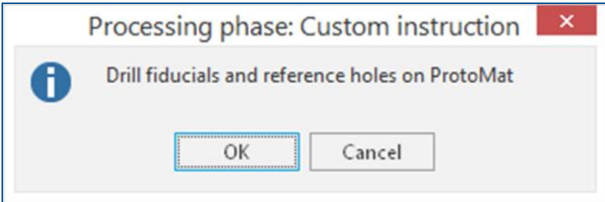


Fig. 198: Message *Processing phase: Custom instruction*

3. Click on [OK].

- The following message is displayed:

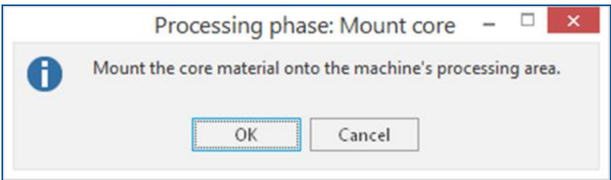


Fig. 199: Message *Processing phase: Mount core*

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:

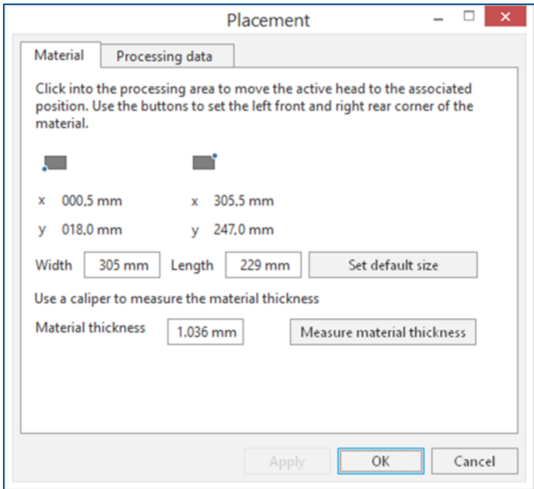


Fig. 200: Dialog *Placement*

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*).

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 1.5.

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 .

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:

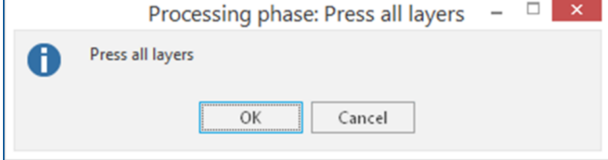


Fig. 201: Message *Processing phase: Press all layers*

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 °C (212 °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*.

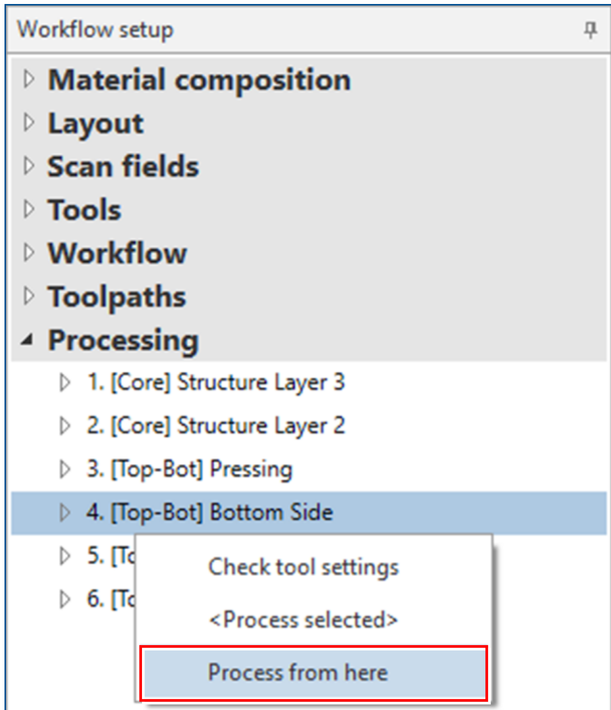


Fig. 202: Context menu *Process from here*

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.

4. 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.

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.

2. 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.
9. Slowly peel off the protective film at an angle of 90° from both sides of the multi-layer 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:

Consumables

- 1 × Base material IS400 235 mm × 305 mm × 0.46 mm, 18/18 µm copper-clad (order code 10092752)
- 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)

Auxiliaries

- 1 set of tools for ProtoMat
- Spiral Drill Blind via for ProtoMat 1/8″, 38 mm, d = 0.20 mm (order code 10099689)
- Convection oven (order code 115877)
- LPKF Cleaner (order code 115891)
- Brush
- Portable hand-held microscope
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4/R4
- MultiPress S
- ProtoMat S or E
- Contac S4

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.

1 Copper layer **2** Carrier 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)

It is essential that you read the design guidelines described in chapter 1.8 before starting this tutorial.

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_Blind (IS400).cbf*.
2. Click on *File* > *Import* or on .
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:

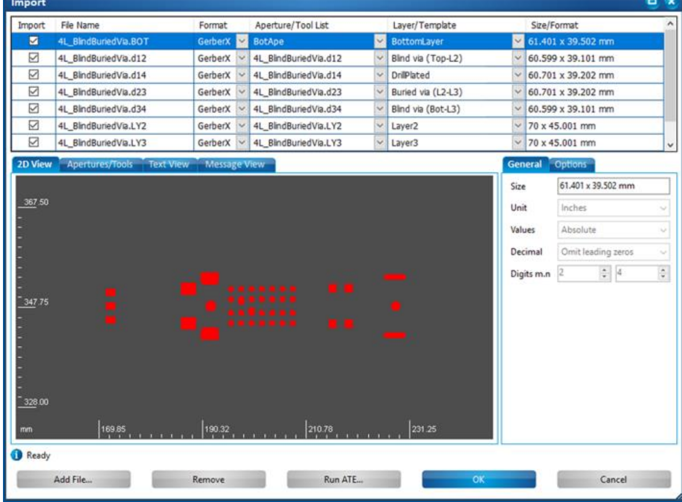


Fig. 203: Dialog *Import* | Assigned layers

5. 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 .

☐ 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.
2. 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.

4. 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)

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.

The recommended **total copper thickness** after through-hole plating is **30 µm to 35 µm**.

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:

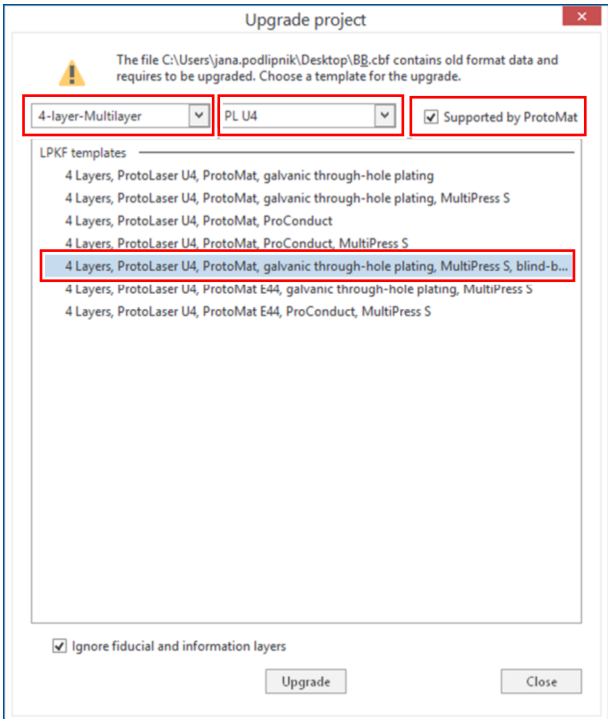


Fig. 205: Upgrade selected

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*.

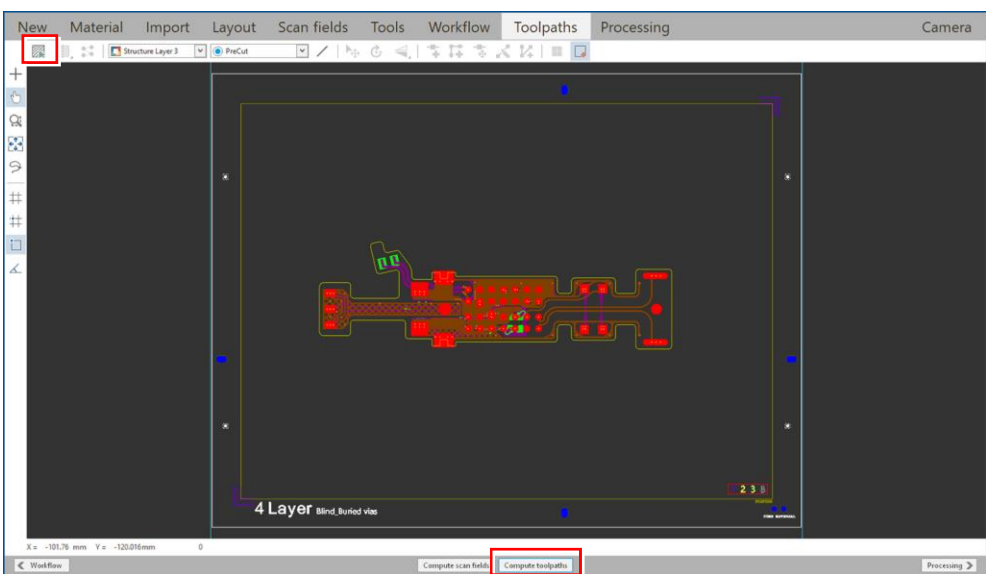


Fig. 207: Opened project

11. Click on [Compute toolpaths] or on .

☐ The following dialog is displayed:

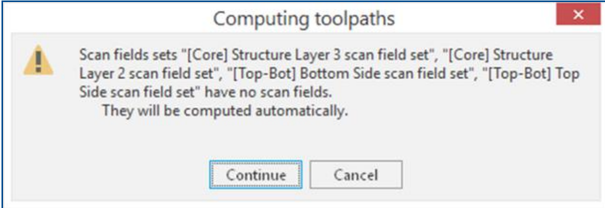


Fig. 208: Dialog *Computing toolpaths*

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.

Structuring the core material (with ProtoLaser)

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

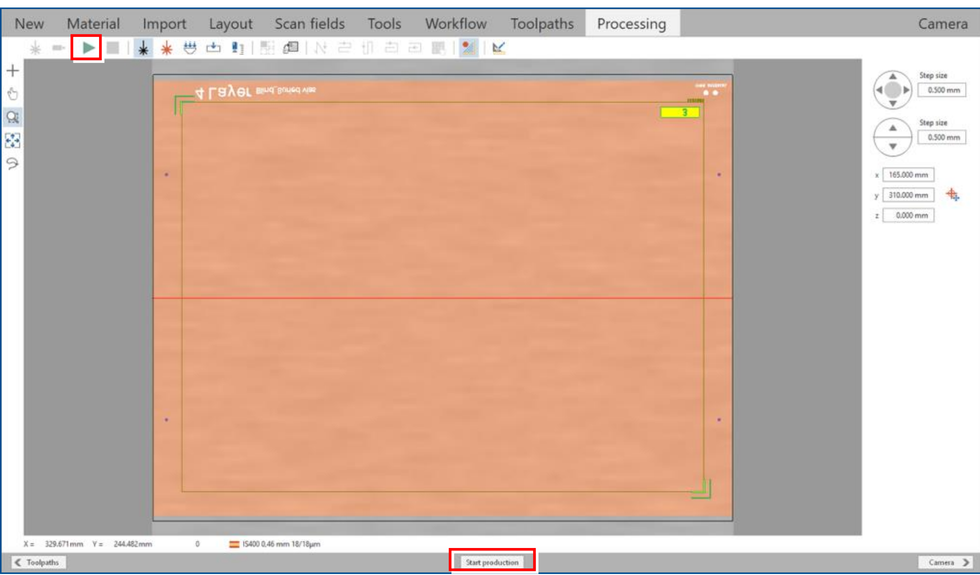


Fig. 209: User guidance step *Processing*

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

☐ The following message is displayed:

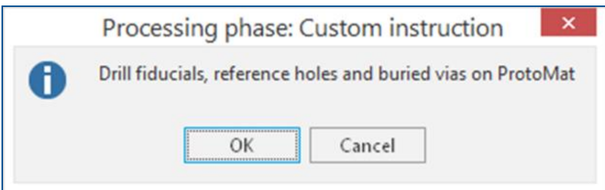


Fig. 210: Message *Processing phase: Custom instruction*

3. Click on [OK].

☐ When the message *Processing phase: Galvanic through-hole plating* is displayed, click on [OK].

☐ When the message *Processing phase: Mount material bottom* is displayed, 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.

4. Click on [vacuum icon].

☐ The core material is fastened onto the processing table by vacuum.

5. Click on [OK].

☐ The following dialog is displayed:

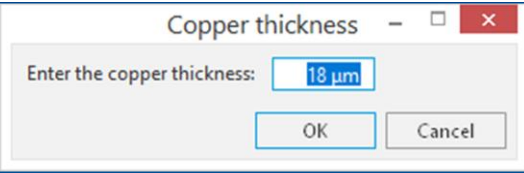


Fig. 211: Dialog *Copper thickness*

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:

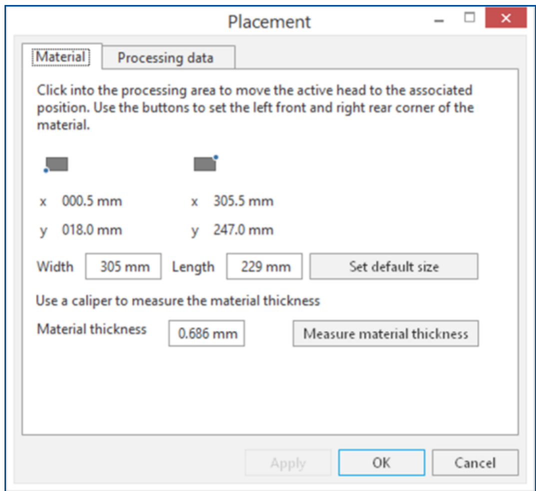


Fig. 212: Dialog *Placement*

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.

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 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:

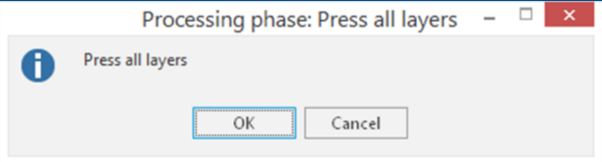


Fig. 213: Message *Processing phase: Press all layers*

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:

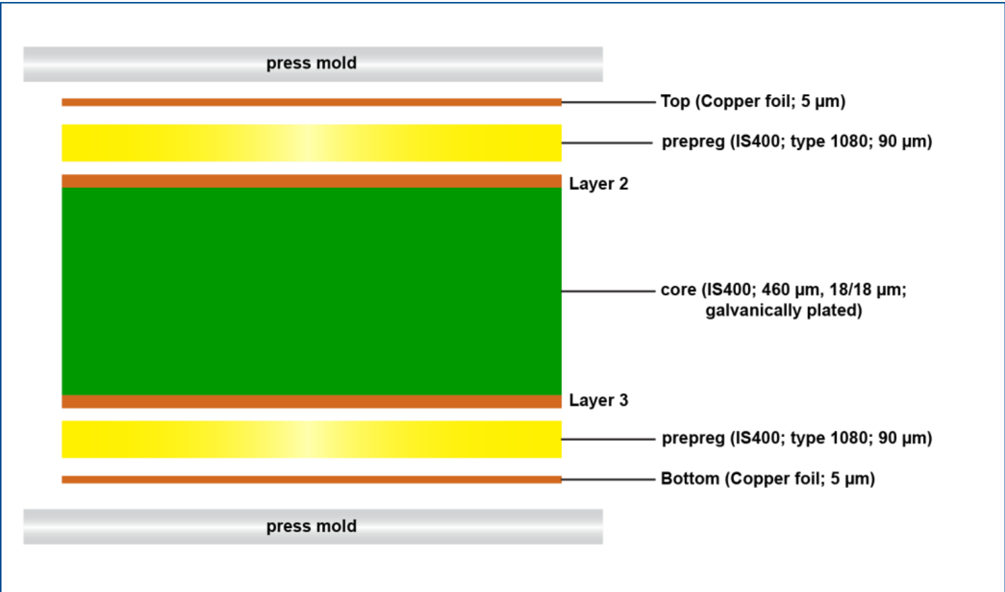


Fig. 214: Assembly of the materials in the press mold

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.

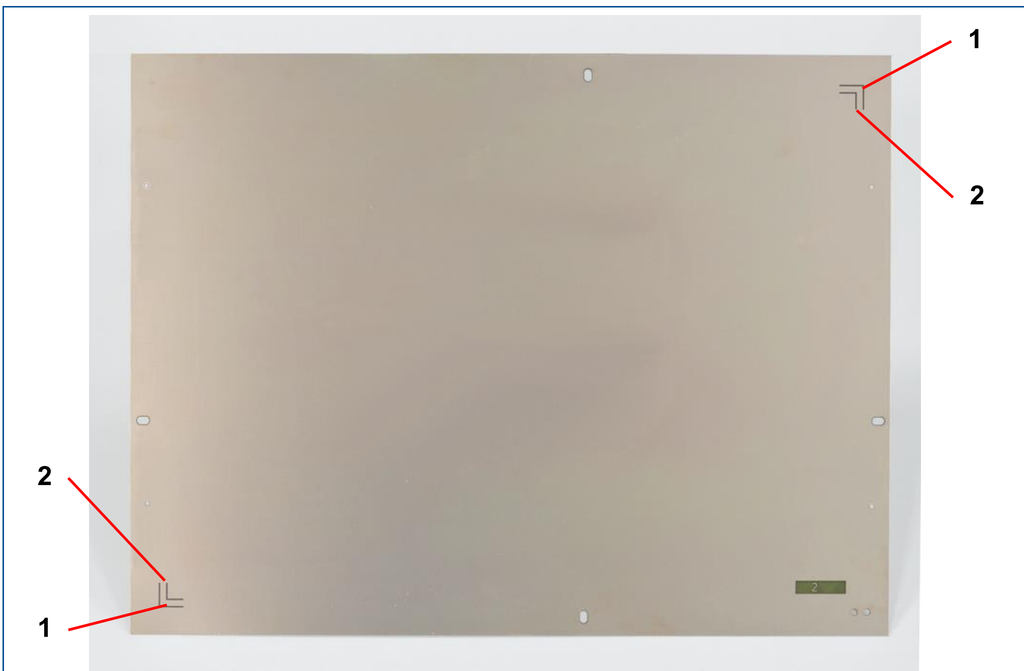


Fig. 215: Positioning markers core material

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.

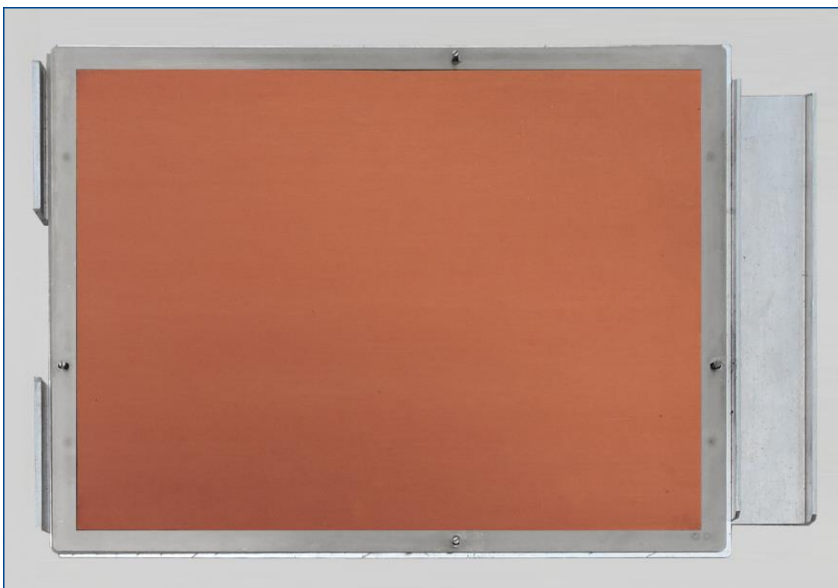


Fig. 216: Copper foil on the lower steel press sheet

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.

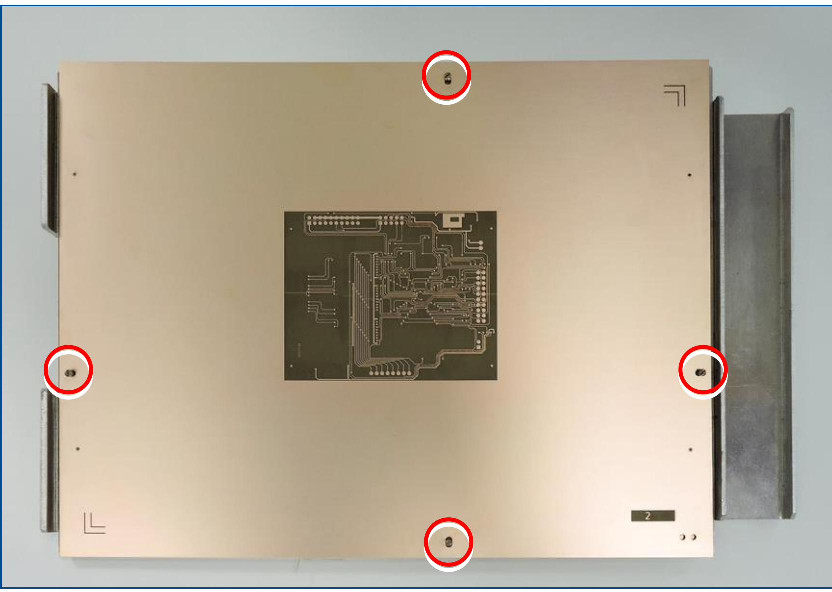


Fig. 218: Core material on the dowel pins

7. Place the prepreg material on the core material.
   Align it with the inner positioning markers (**2**).

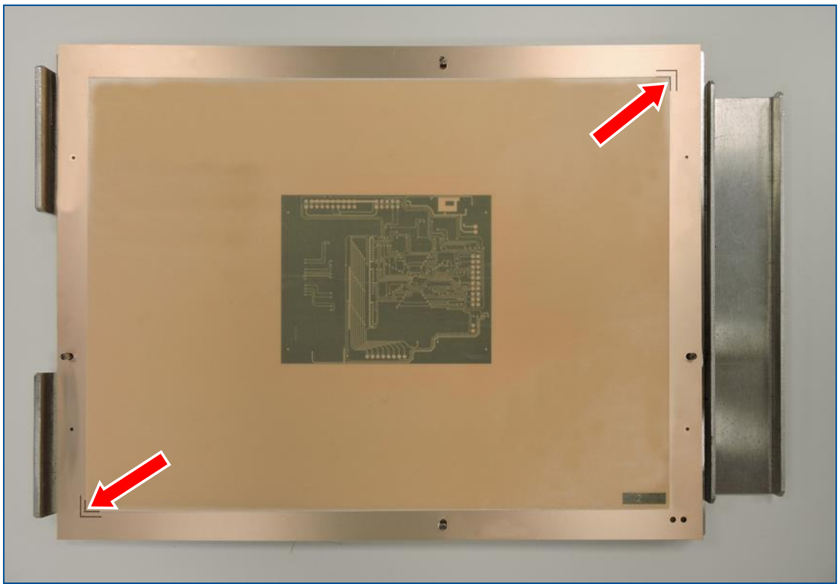


Fig. 219: Prepreg material on the core material

8. Place the copper foil on the prepreg material.
   Align it with the outer positioning markers (**1**).

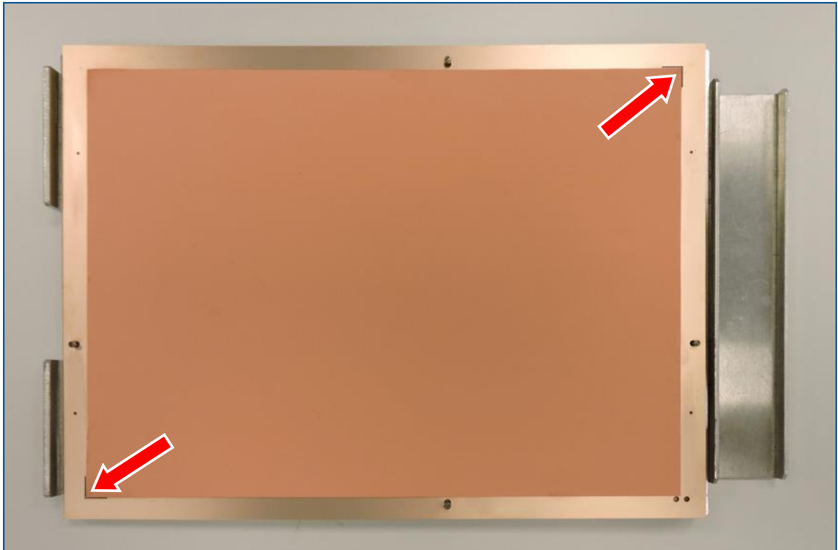


Fig. 220: Copper foil on the prepreg material

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 .

☐ The message *Processing Phase: MountMaterialBottom* is displayed.

4. 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 .
- 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*.

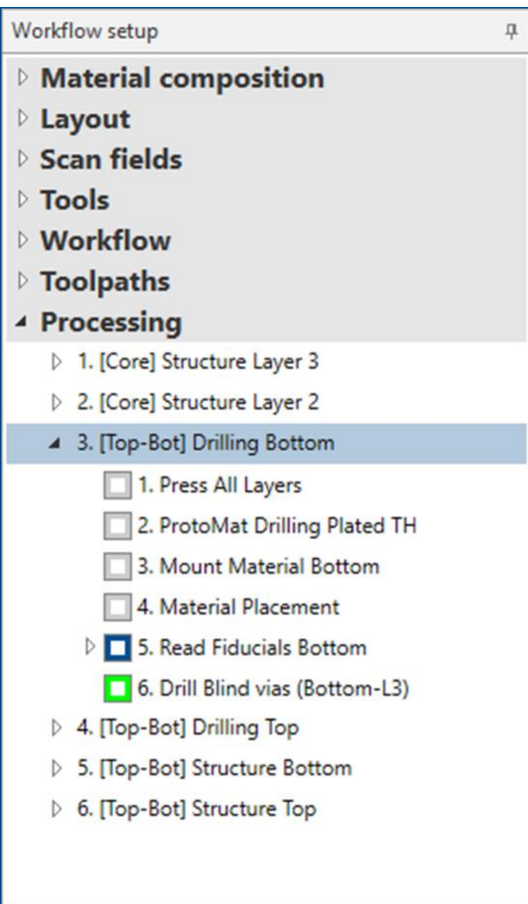


Fig. 221: Pane *Workflow setup*

5. Right-click on *Mount Material Bottom*.

- 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].

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:

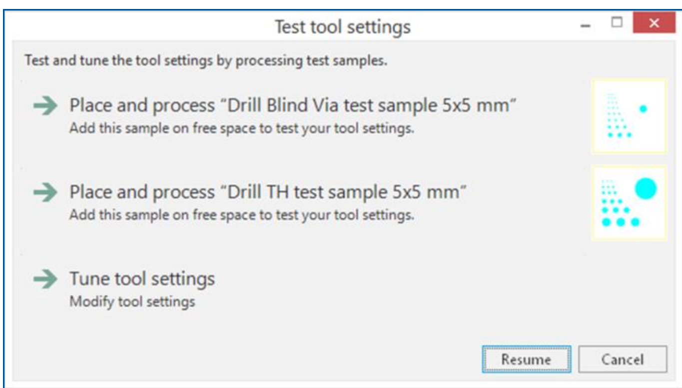


Fig. 223: Dialog *Test tool settings*

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.
15. 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.

☐ The following figure shows a blind via of good quality:

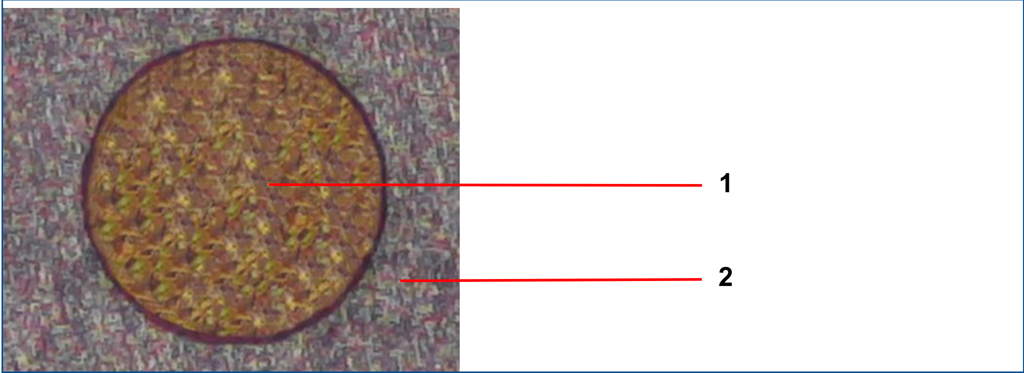


Fig. 224: Blind via of good quality under the microscope

1 Bottom of blind via (in this example *Layer3*)

2 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*).

19. 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].

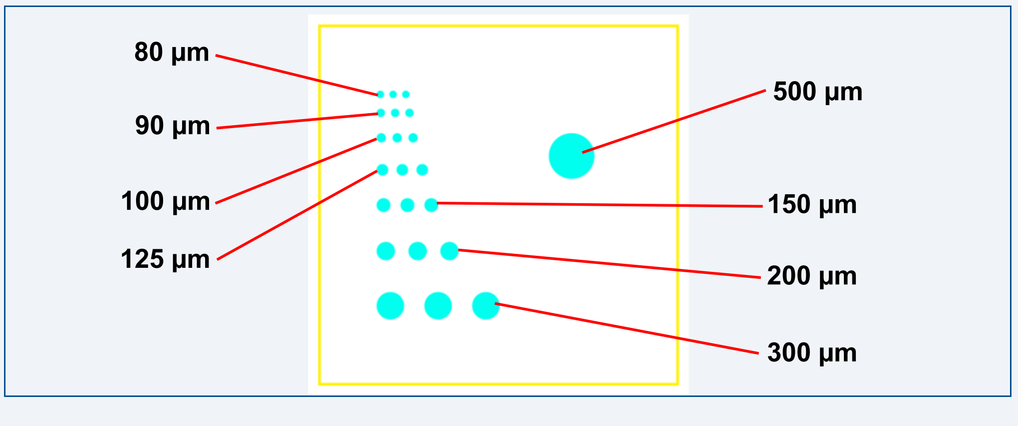
☐ The laser system drills the blind vias on the Top side (*TopLayer*).

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 µm to 35 µm**.

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. 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*.

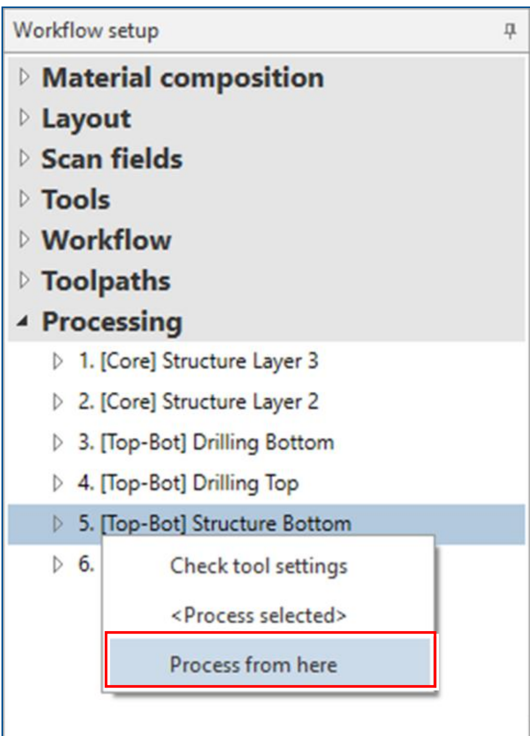


Fig. 225: Context menu *Process from here*

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:

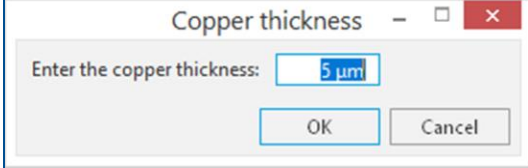


Fig. 226: Dialog *Copper thickness*

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*).

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:

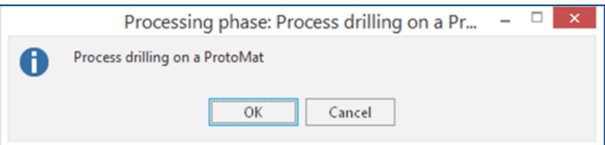


Fig. 227: Message *Processing phase: Process drilling on a ProtoMat*

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 .

- The message *Processing Phase: MountMaterialTop* is displayed.

4. 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

- Polyimide Foil A4, 125 µm, format for Stencils (order code: 108321)

System

- ProtoLaser U4/S4/R4

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:

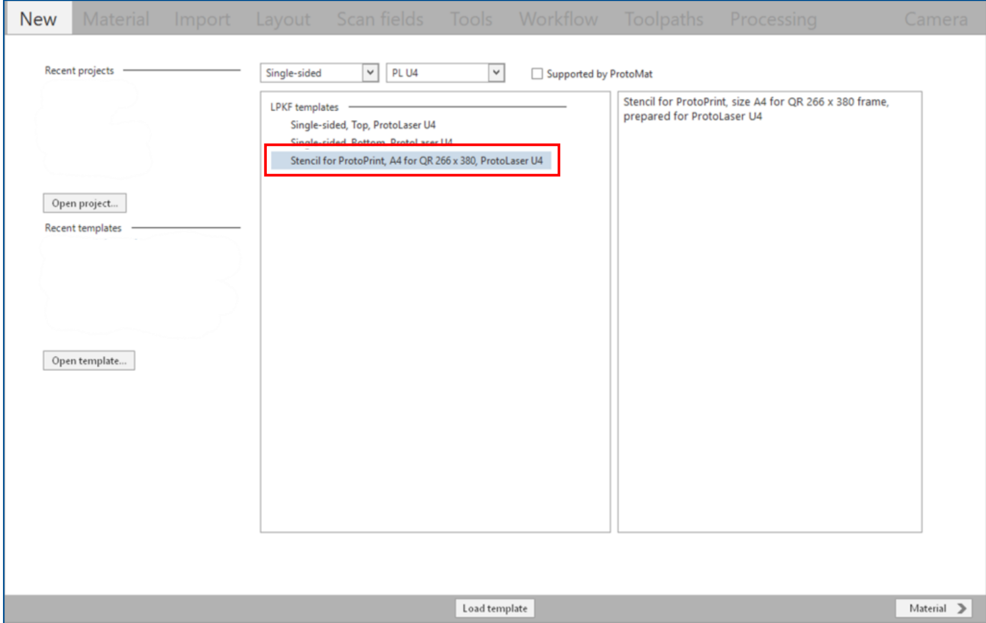


Fig. 228: List of single-sided templates

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*.

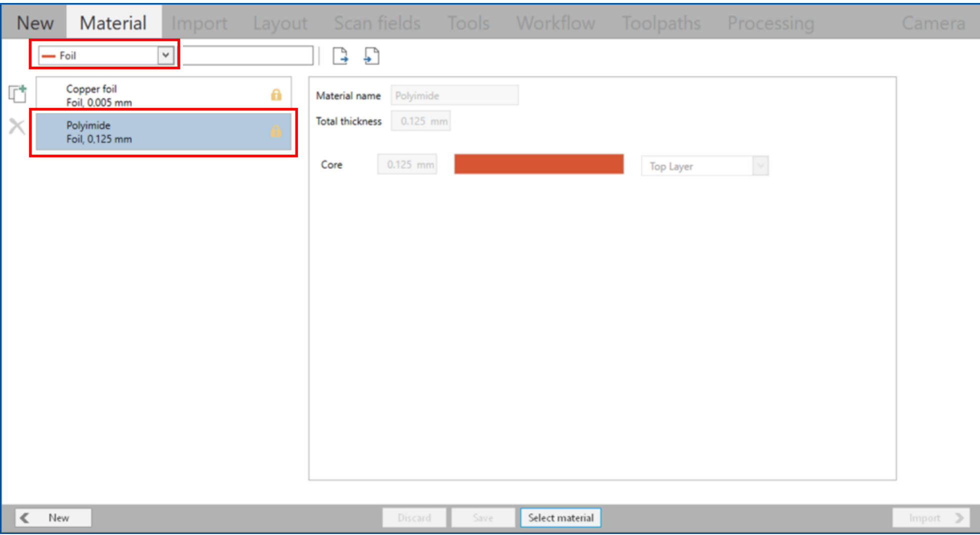


Fig. 229: Material selection

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:

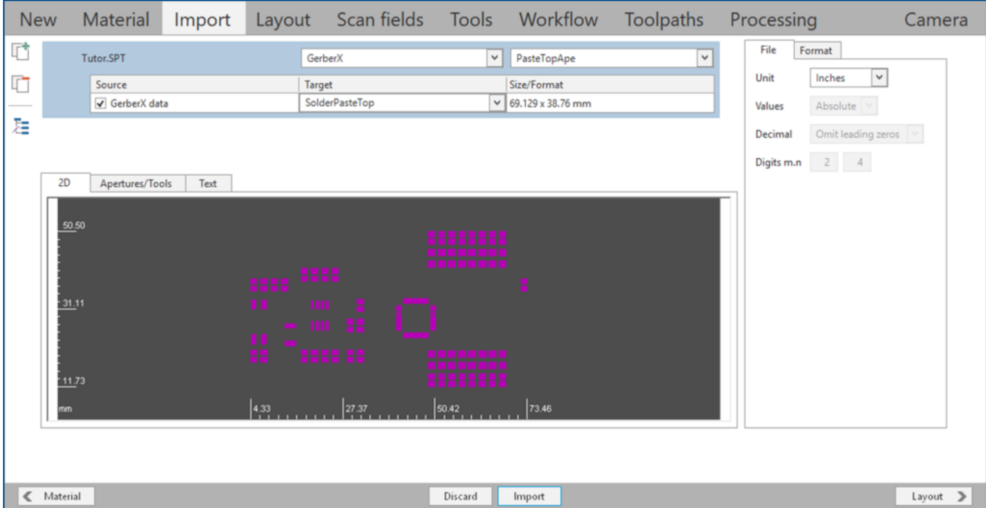


Fig. 230: User guidance step *Import*

12. Click on [Import].

☐ The layout is displayed in the user guidance step *Layout*.

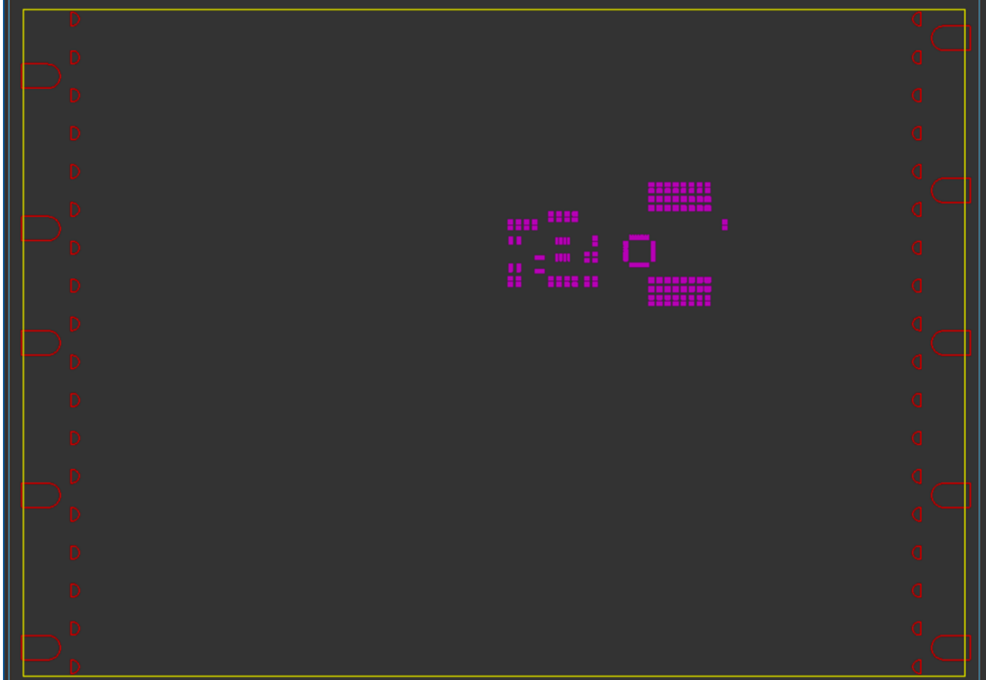


Fig. 231: Layout of a polyimide stencil

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**.

☐ 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*.
20. Click on [Compute toolpaths] or on ▨.

☐ 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*.

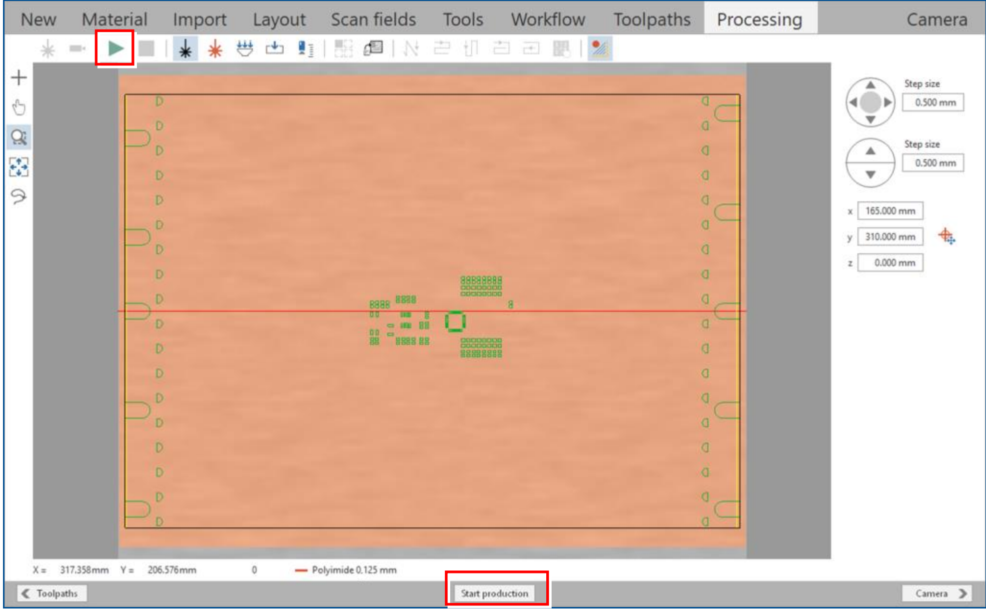


Fig. 232: User guidance step *Processing*

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.
4. Click on [vacuum icon].

☐ 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.

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.

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 adhesive
- Portable hand-held microscope
- Brush
- Oil-free compressed air
- Tap water

System

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

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*.

Fig. 233: User guidance step *Material*

3. Click on .

☐ A copy of the selected material is created:

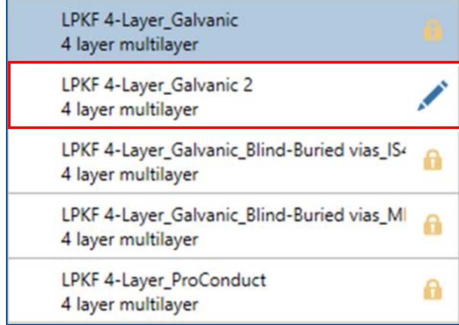


Fig. 234: Copy of selected material

4. Click on .

- 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*).

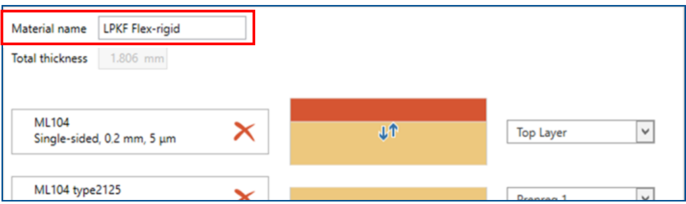


Fig. 236: Entering the material name

- Delete the top five materials in the set of materials by clicking on ✗ next to the material name.

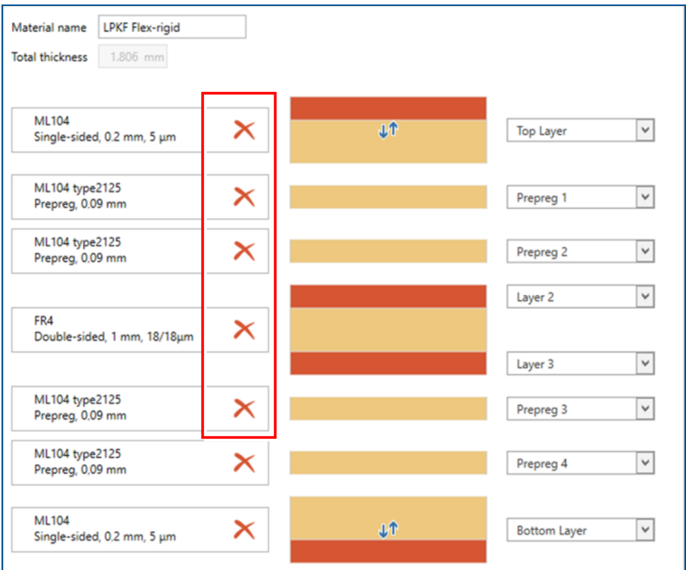


Fig. 237: Deleting the materials

☐ 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.

7. 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 ➕.

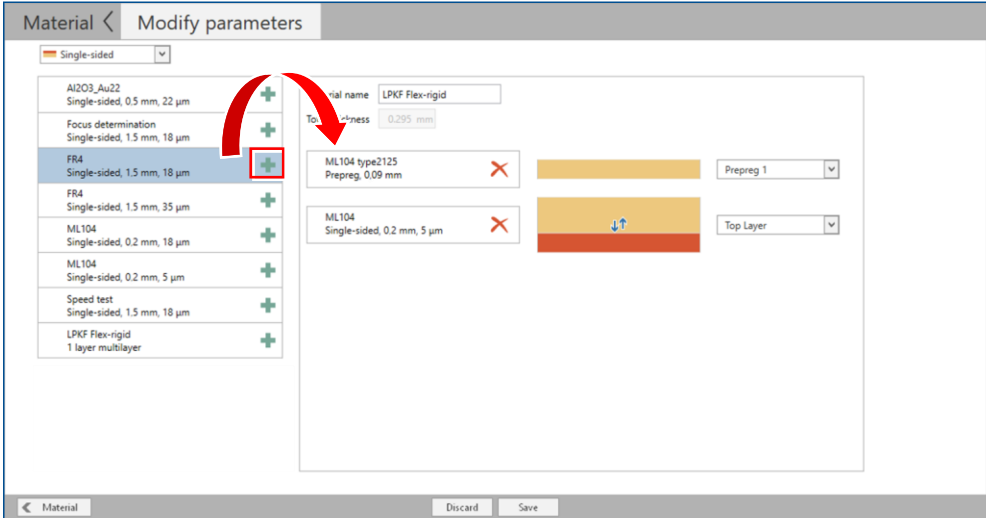


Fig. 239: Adding the material

☐ The material *FR4 Single-sided, 1.5 mm, 18 µm* is added to the top of the new set of materials:

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*.

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 .

☐ The dialog *Open* is displayed.

2. 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].

- 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*.

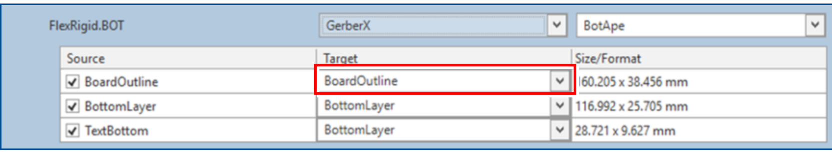


Fig. 243: Assigning board outline to layer *BoardOutline*

6. Deactivate the check box *BoardOutline* in the column *Source*.

Fig. 244: Deselecting the source *BoardOutline*

7. In the column *Target* assign the data *FlexRigid.DRL* to the layer *DrillPlated*.

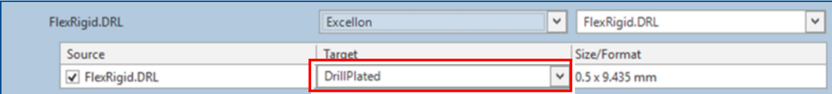


Fig. 245: Assigning the holes to layer *DrillPlated*

8. Click on [Import].

- The following message is displayed:

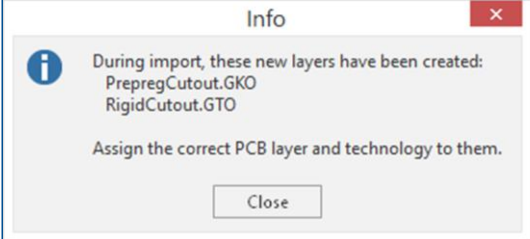


Fig. 246: Message *Info*

9. Click on [Close].

☐ The user guidance step *Layout* is displayed:

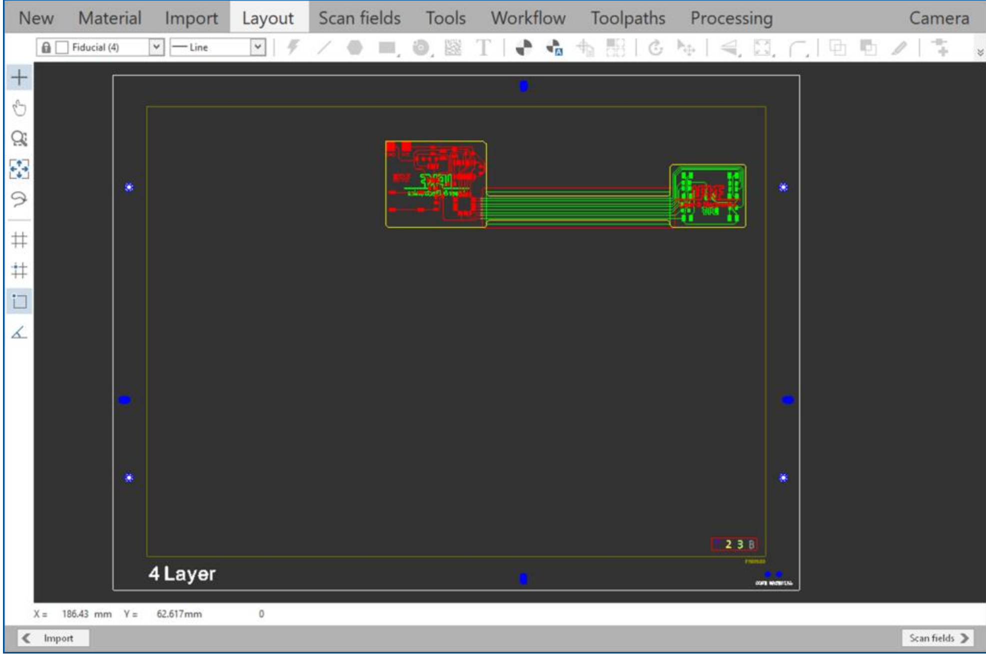


Fig. 247: User guidance step *Layout*

☑ 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.

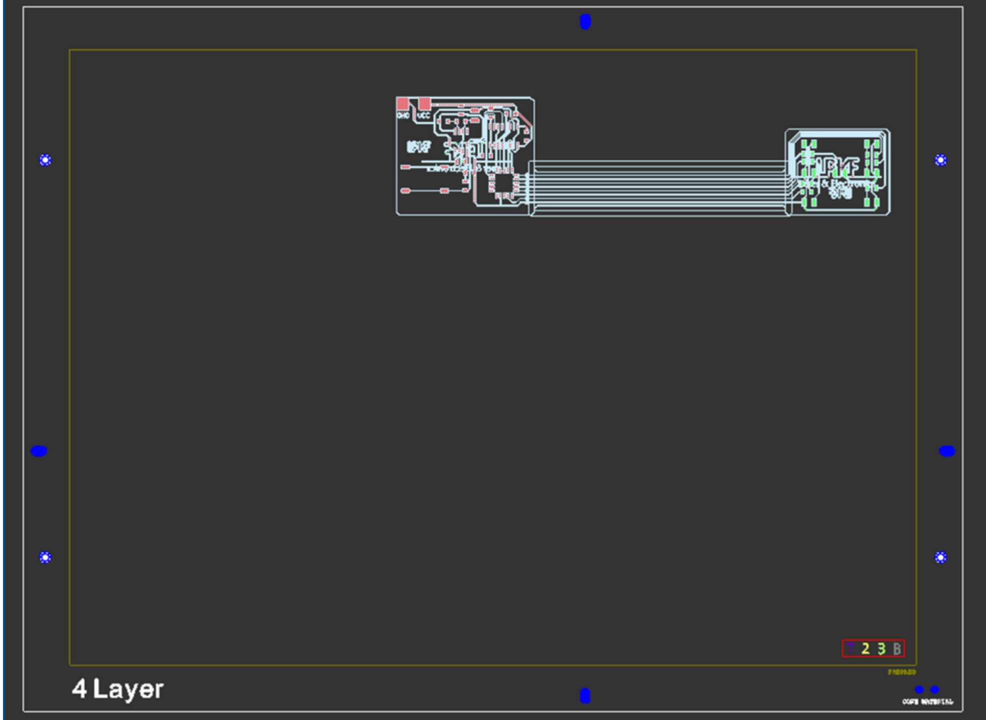


Fig. 248: Highlighted layout

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.

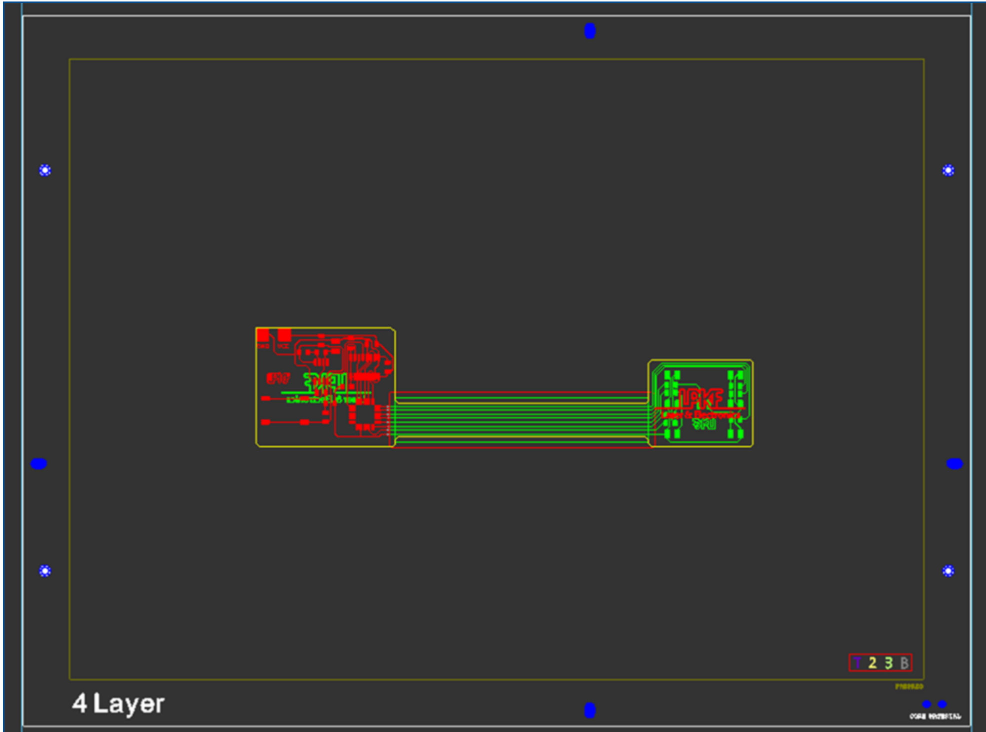


Fig. 249: Layout moved

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*.

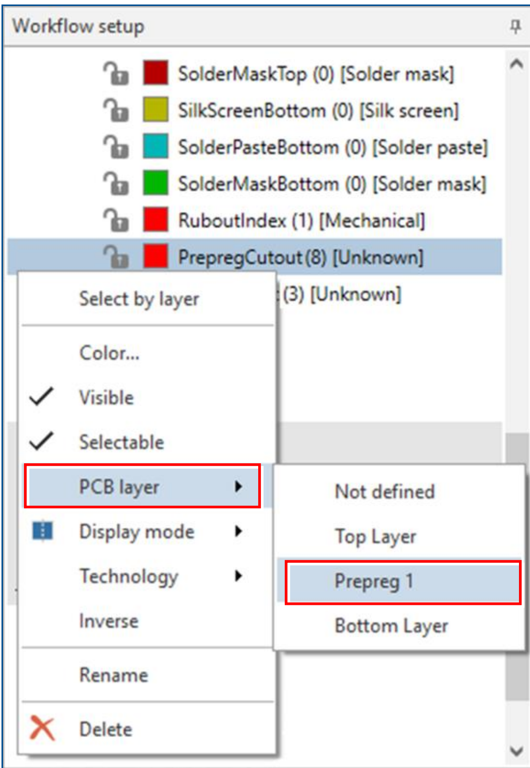


Fig. 250: Context menu *PCB layer > Prepreg 1*

9. Right-click on the layer *PrepregCutout* and click on the context menu item *Display mode* > *True Width*.

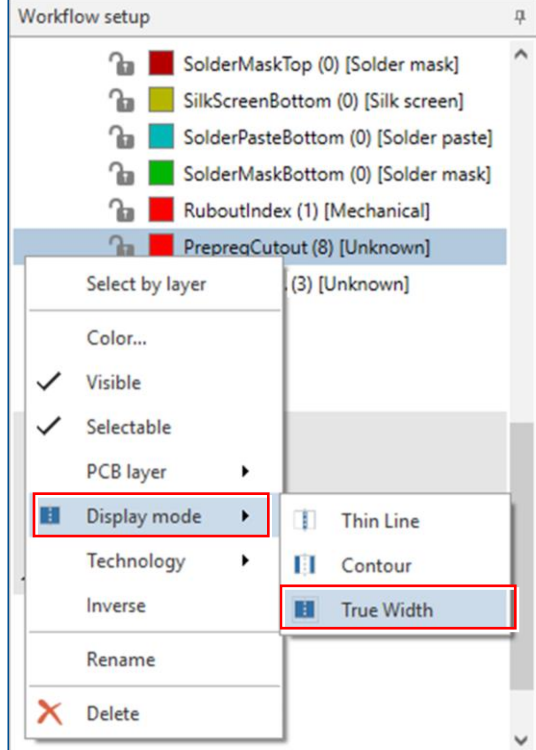


Fig. 251: Context menu *Display mode > True Width*

10. Right-click on the layer *PrepregCutout* and click on the context menu item *Technology > Cutting*.

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.

Creating the positioning holes for prepreg material and flexible material

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*.

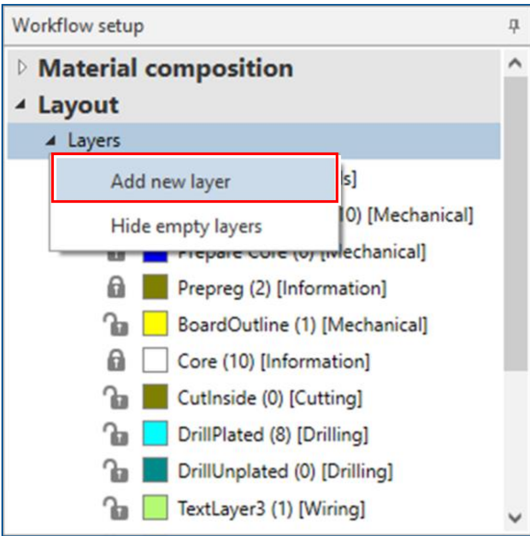


Fig. 253: Menu item *Add new layer*

- ☐ A new layer is created.

3. Right-click on the new layer and click on the context menu item *Rename*.

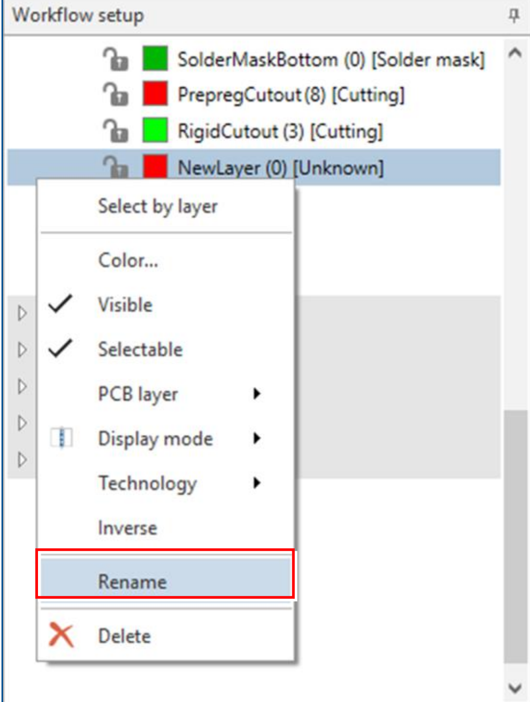


Fig. 254: Context menu item *Rename*

4. Enter *FlexPositioningHoles* in the input field.

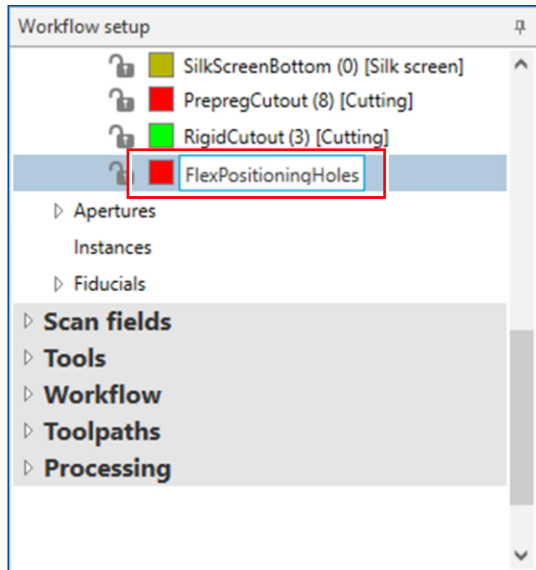


Fig. 255: Renaming layer

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 [icon].

☐ The input fields for *Specify center point* are displayed:

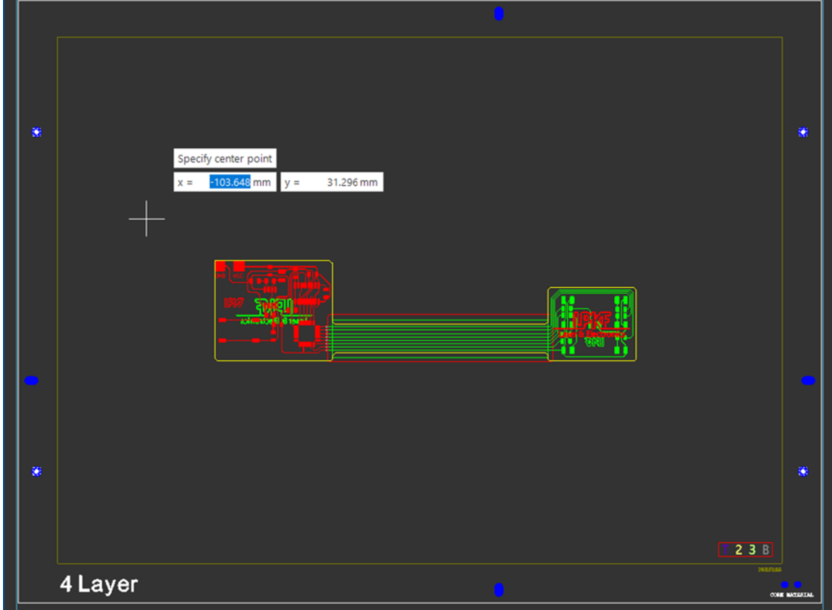


Fig. 256: Input fields *Specify center point*

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:

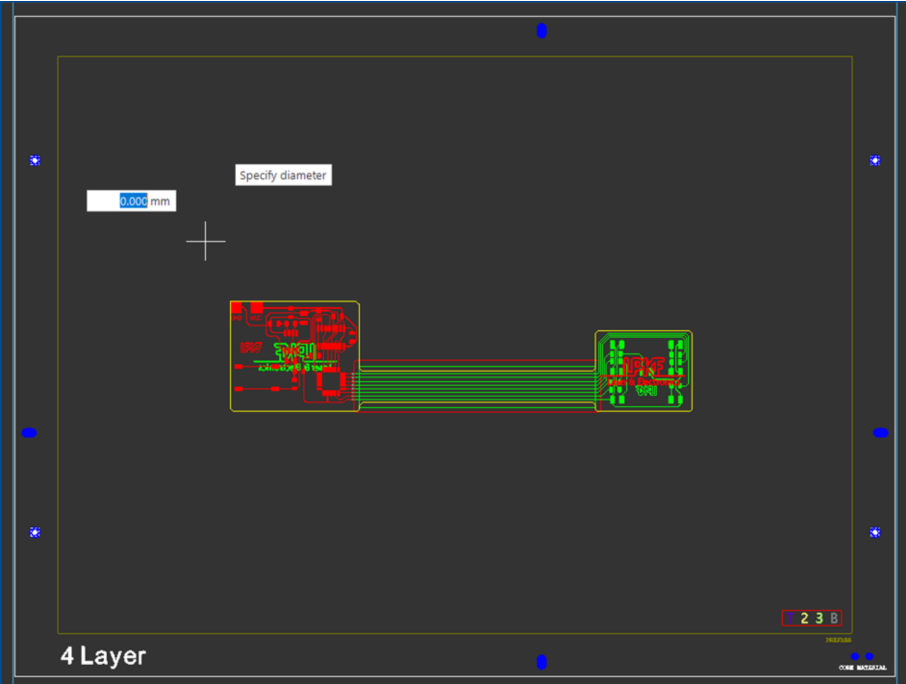


Fig. 257: Input fields *Specify diameter*

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:

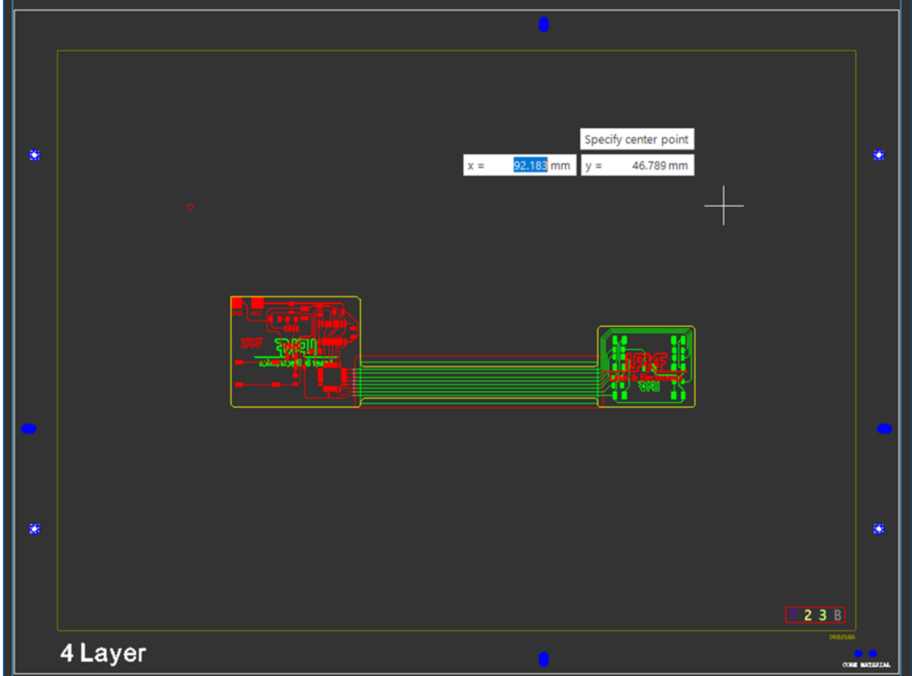


Fig. 258: Specifying the next center point

- ☐ 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*:

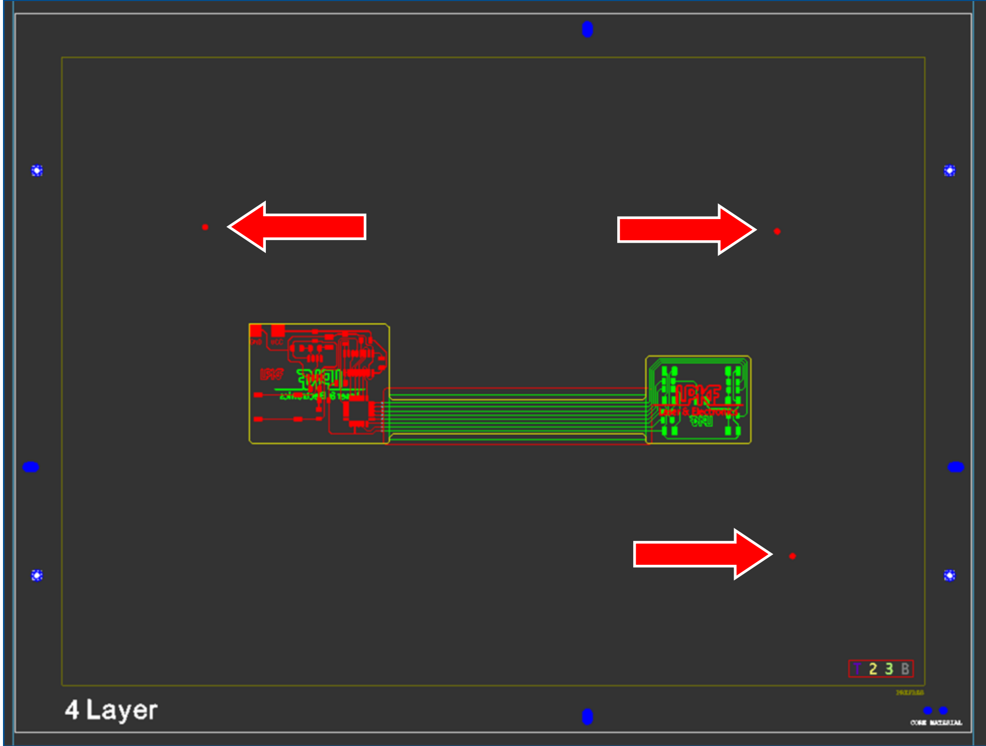


Fig. 259: Flexible material positioning holes

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*.

Cut Ctrl+X
Copy Ctrl+C
Invert selection
Group
Move object Ctrl+M
Rotate
horizontally
vertically
Scale
Step and repeat...
Create instance type...
Expand
Merge
Split
Convert to closed path
Convert to flash
Combine to flash
Create fiducial with defaults
Assign to layer
Copy to layer
Create aperture...
Select by geometry
Measure
Delete

BoardOutline
CutInside
DrillPlated
DrillUnplated
TopLayer
Layer2
Layer3
BottomLayer
TextTop
TextLayer2
TextLayer3
TextBottom
RuboutTop
RuboutLayer2
RuboutLayer3
RuboutBottom
SilkScreenTop
SolderPasteTop
SolderMaskTop
SilkScreenBottom
SolderPasteBottom
SolderMaskBottom
RuboutIndex
PrepregCutout
RigidCutout
FlexPositioningHoles
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:

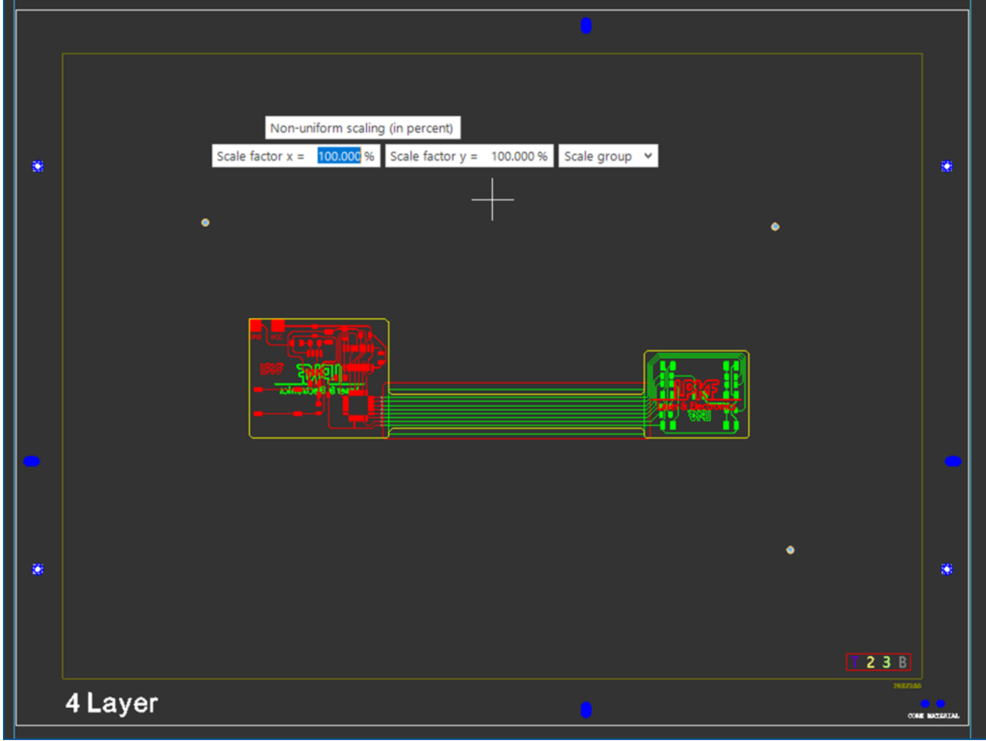


Fig. 261: Input fields *Non-uniform scaling (in percent)*

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 [↓] key twice.
- The option *Individually* is selected in the drop-down list.

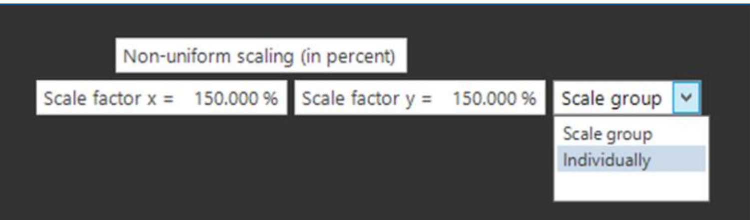


Fig. 262: Values for *Non-uniform scaling (in percent)*

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.

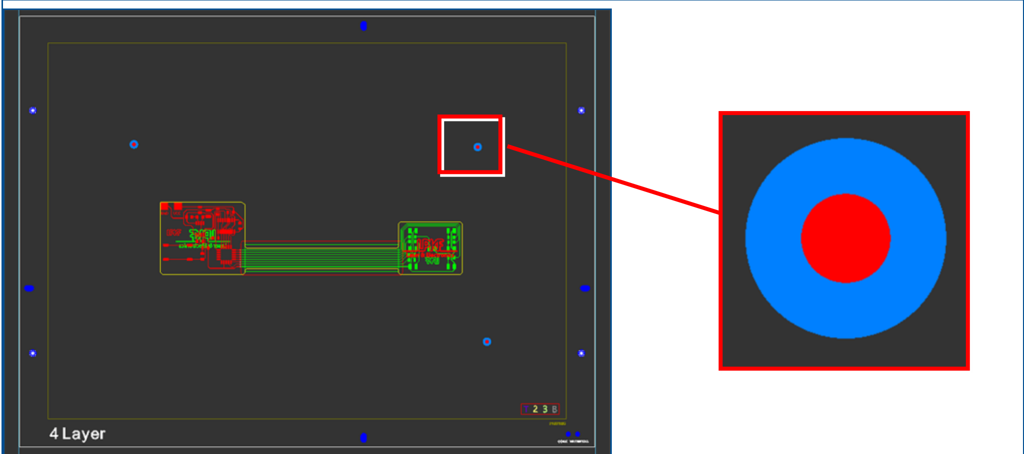


Fig. 264: Prepreg positioning holes

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*.

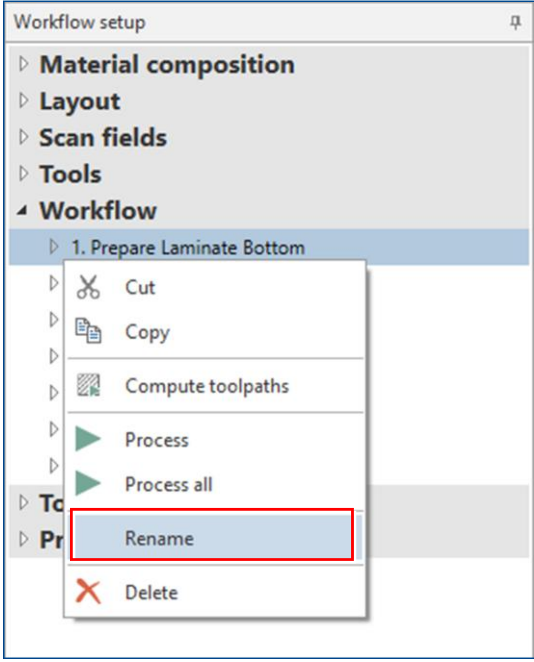


Fig. 265: Rename

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*.

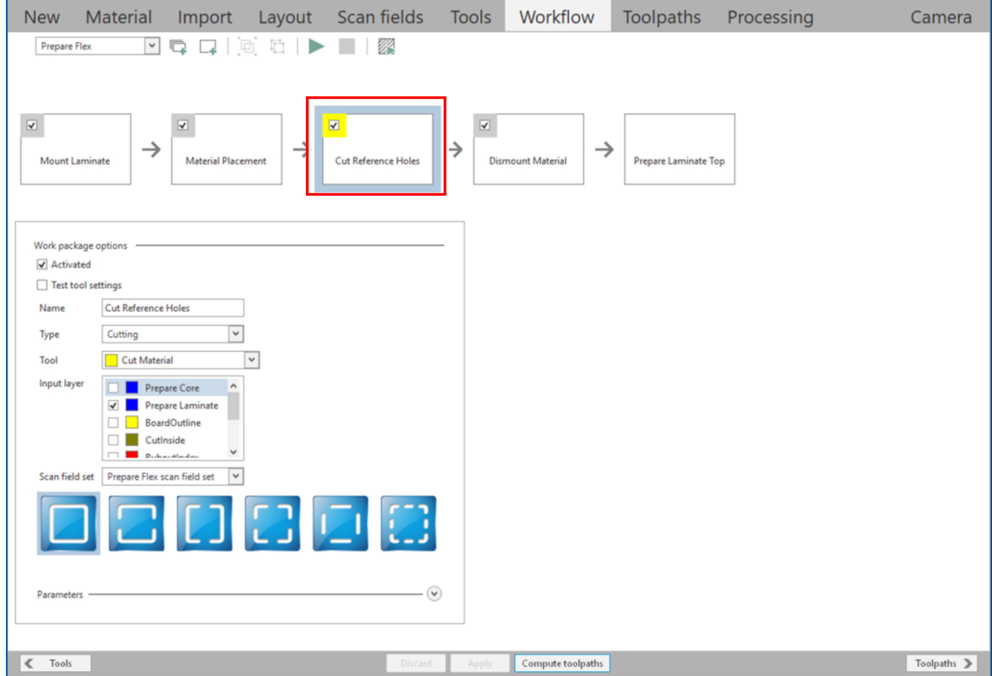


Fig. 266: Work package selected

7. Under *Input layer* activate the check box *FlexPositioningHoles*.

Work package options
Activated
Test tool settings
Name Cut Reference Holes
Type Cutting
Tool Cut Material
Input layer PrepregCutout.... RigidCutout FlexPositioning... PrepregPositioni...
Scan field set Prepare Flex scan field set
Parameters

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*.

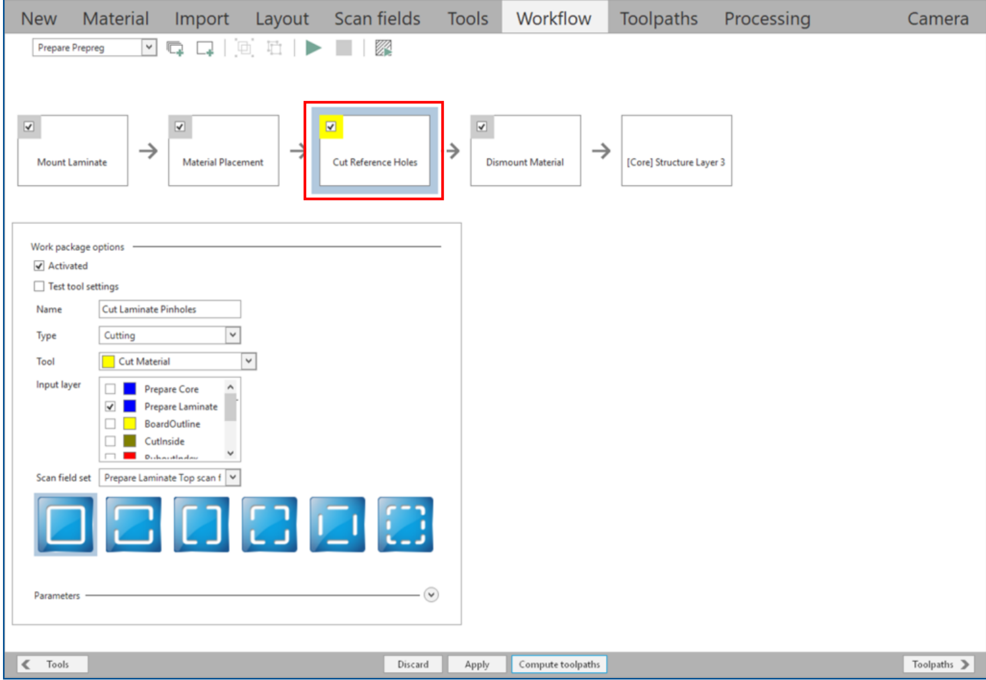


Fig. 268: Work package selected

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:

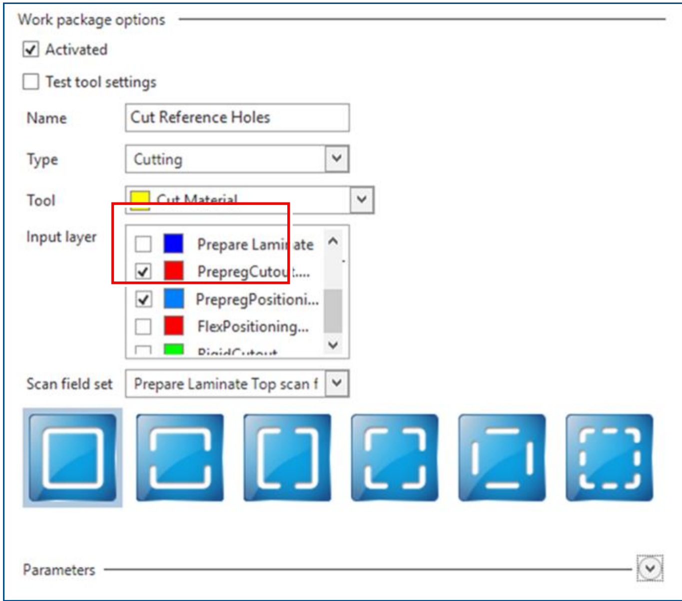


Fig. 269: Settings of the work package *Cut Reference Holes*

14. Expand the group *Parameters*.
15. Enter *20 µm* in the input field *Channel width*.

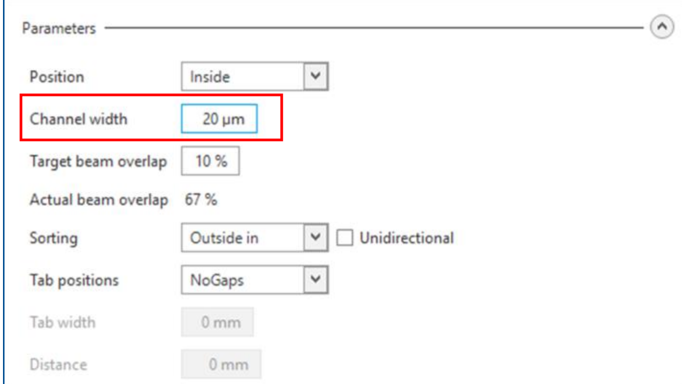


Fig. 270: *Channel width* modified

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*.

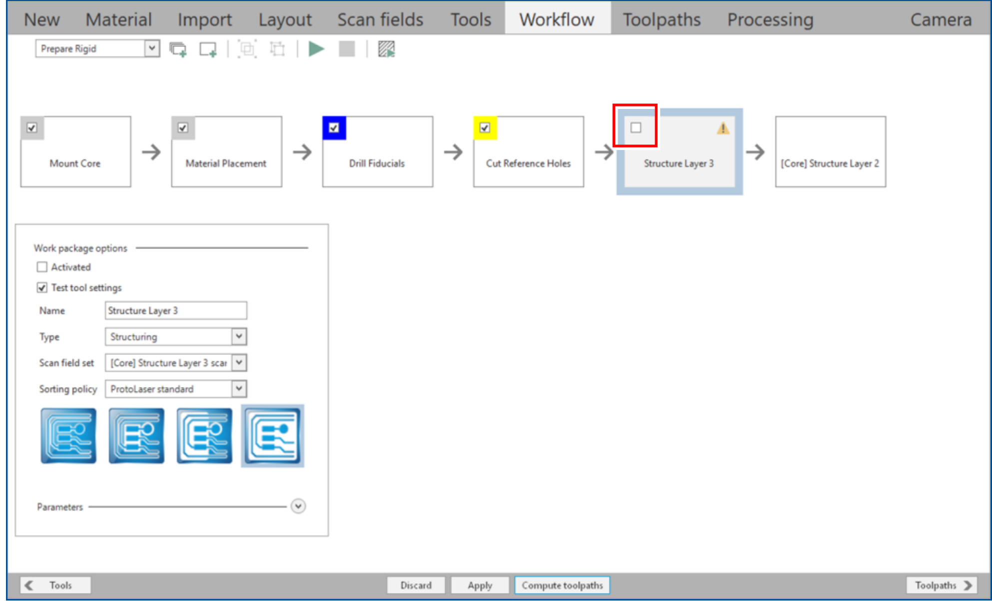


Fig. 271: Work package deselected

- The work package is deselected.

21. Select the work package *Cut Reference Holes*.

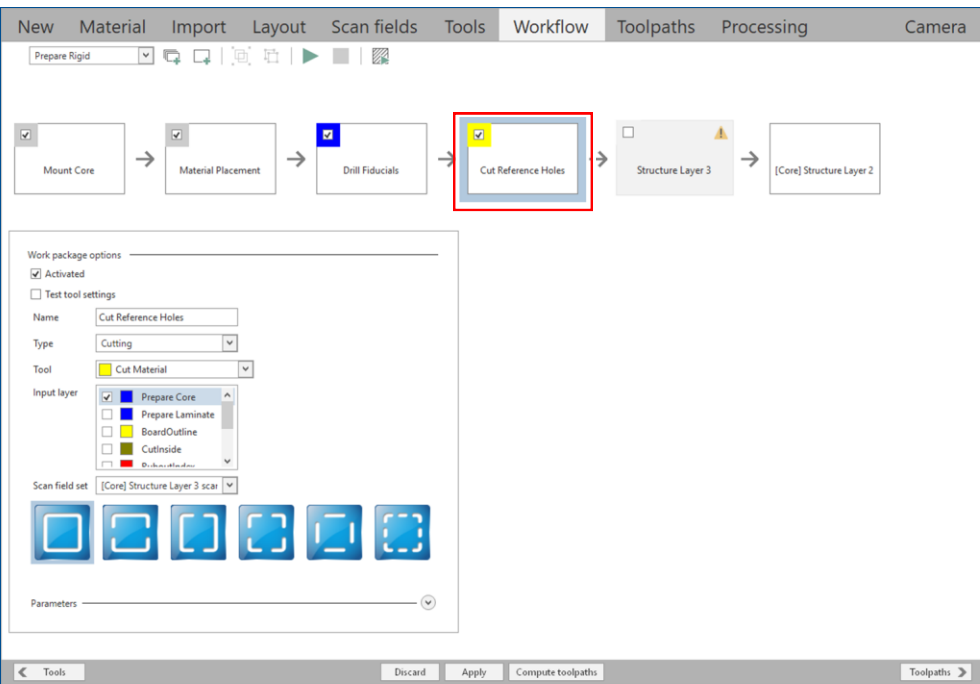


Fig. 272: Work package selected

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 ✕.

☐ 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*.

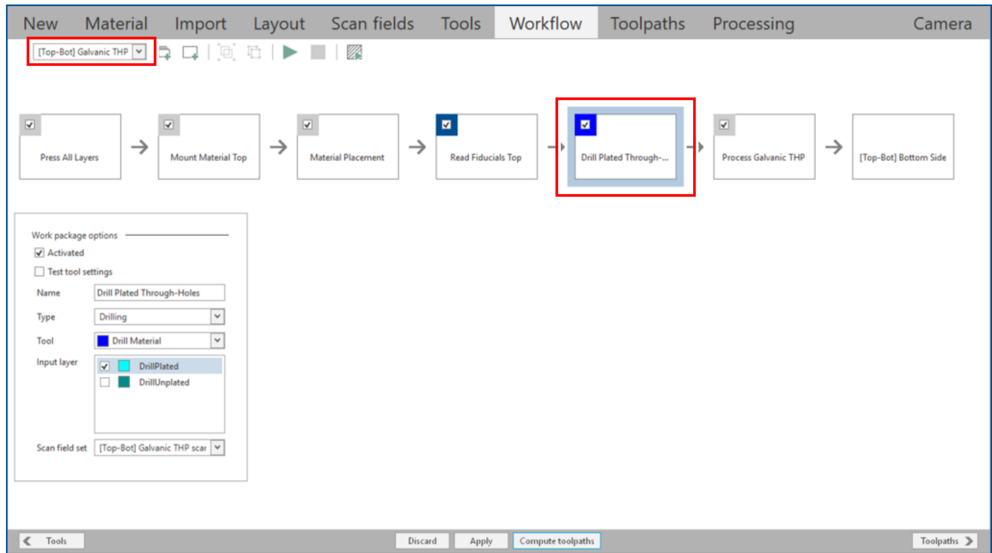


Fig. 274: **Phase and work package selected**

26. Activate the check box *Test tool settings*.

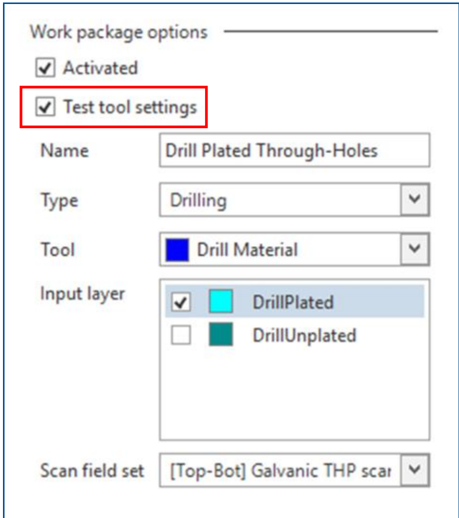


Fig. 275: Modified settings

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*.

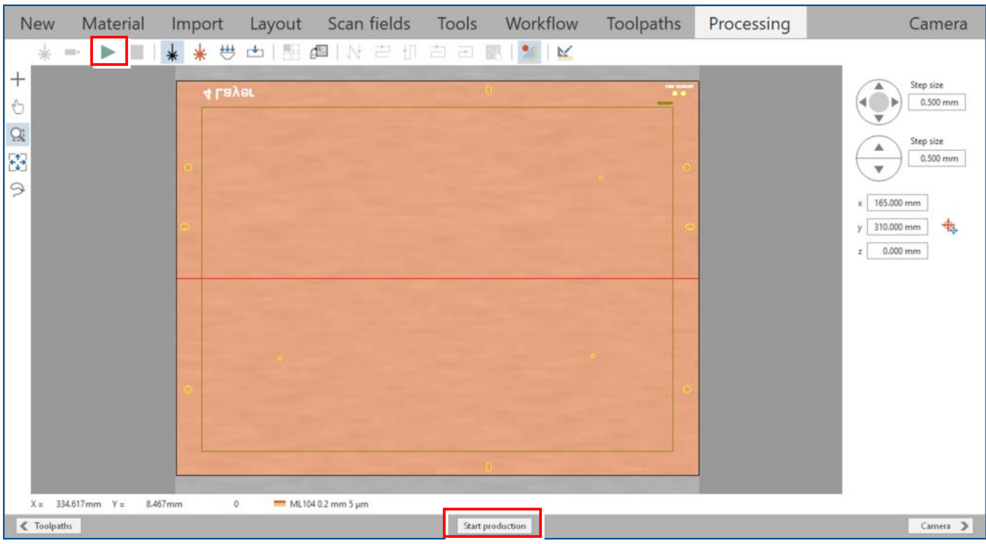


Fig. 276: User guidance step *Processing*

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 [vacuum icon].

☐ 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.

Drilling plated through holes into the flex-rigid 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 group Processing and expand the phase *[Top-Bot] Galvanic THP*.
5. Right-click on *Mount Material Top* and select *Process from here*.

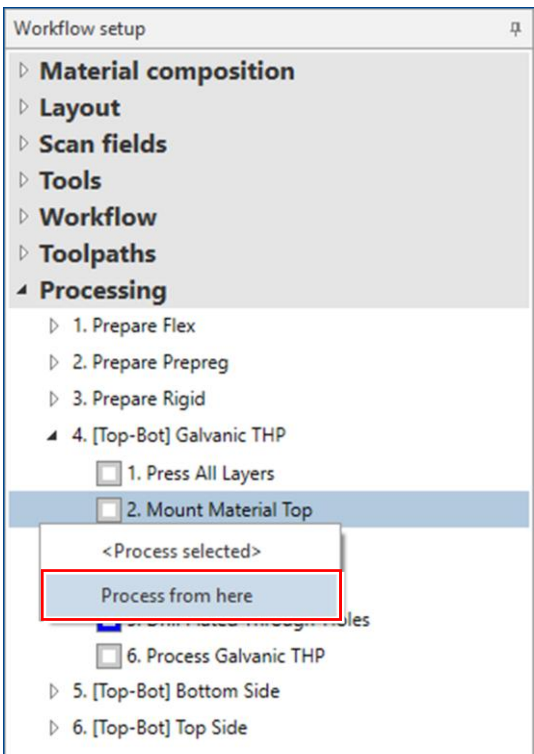


Fig. 277: Context menu *Process from here*

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:

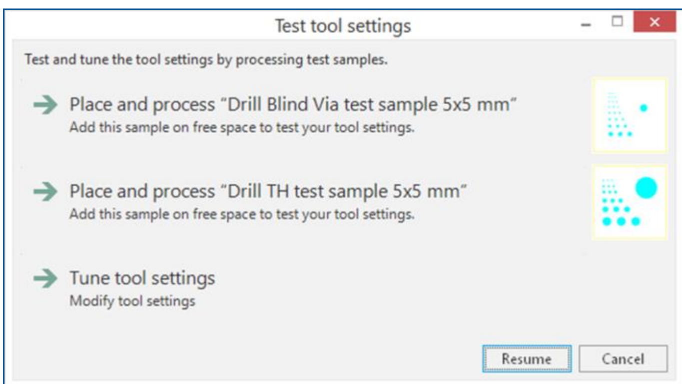


Fig. 278: Dialog *Test tool settings*

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.

For detailed information on tuning the tool settings, refer to chapter 1.5.

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 µm**.

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:

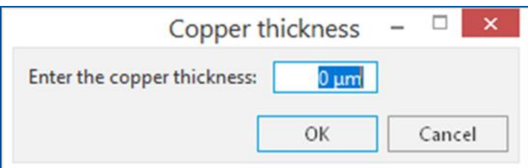


Fig. 279: Dialog *Copper thickness*

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 µm/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*.
12. 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:

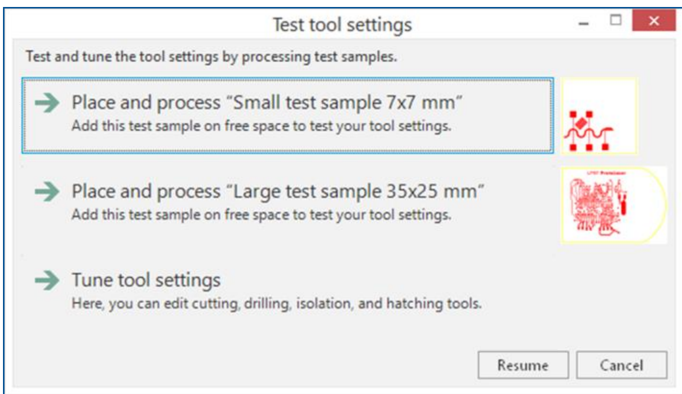


Fig. 280: Dialog *Test tool settings*

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

- Fully structured and through-plated PCB

Auxiliaries

- LPKF Cleaner (order code: 115891)
- Convection oven (order code: 115877)
- Brush
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4/R4
- ProMask

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.

Applying the solder resist on the PCB (with ProMask)

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 . Make sure you assign them to the layers *SolderMaskBottom* and *SolderMaskTop*.

2. Switch to the user guidance step *Layout*.

☐ The layout changes as follows:

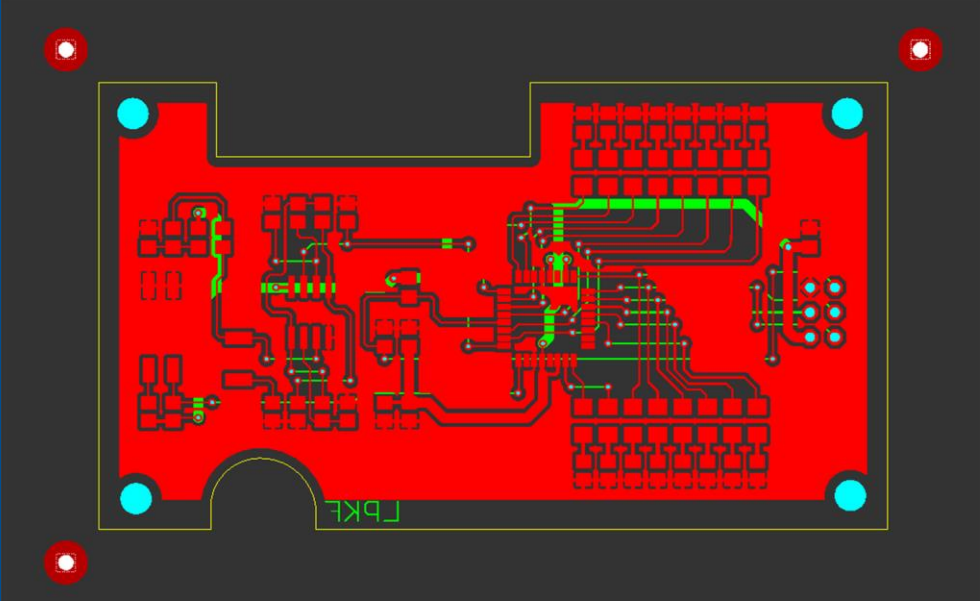


Fig. 281: User guidance step *Layout*

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*.

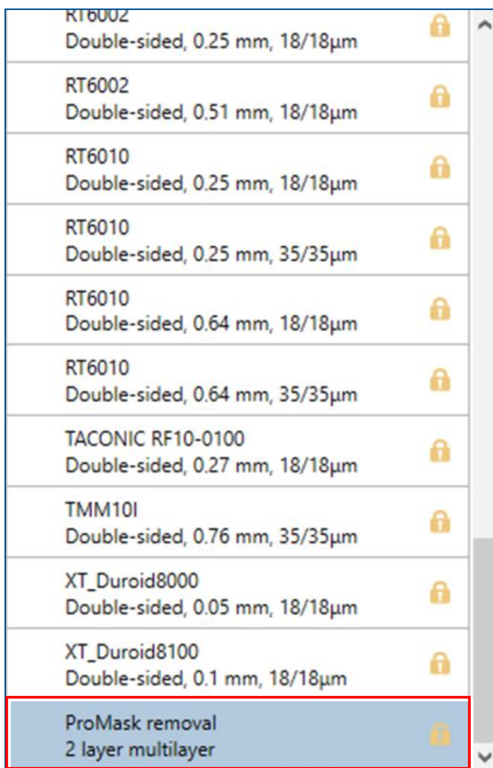


Fig. 282: Material selection

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:

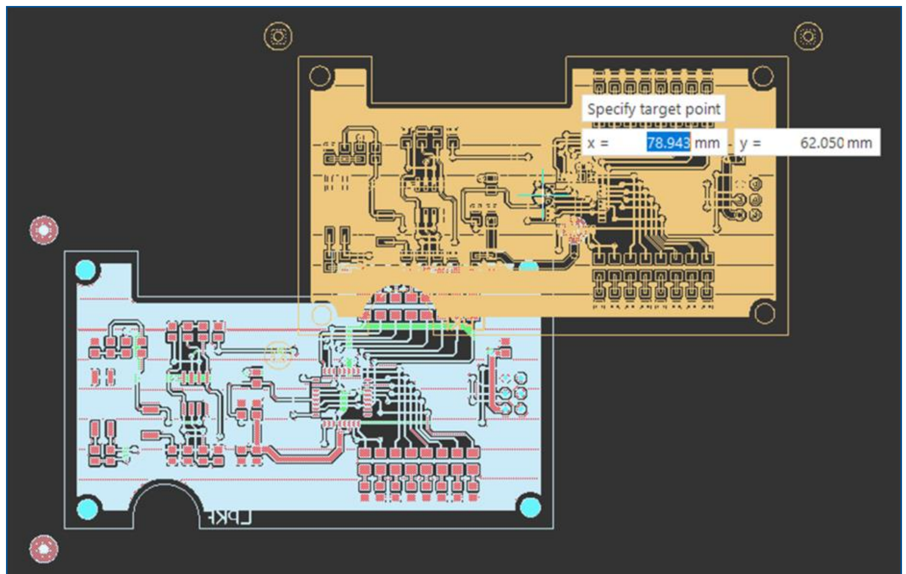


Fig. 283: *Specify target point*

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*:

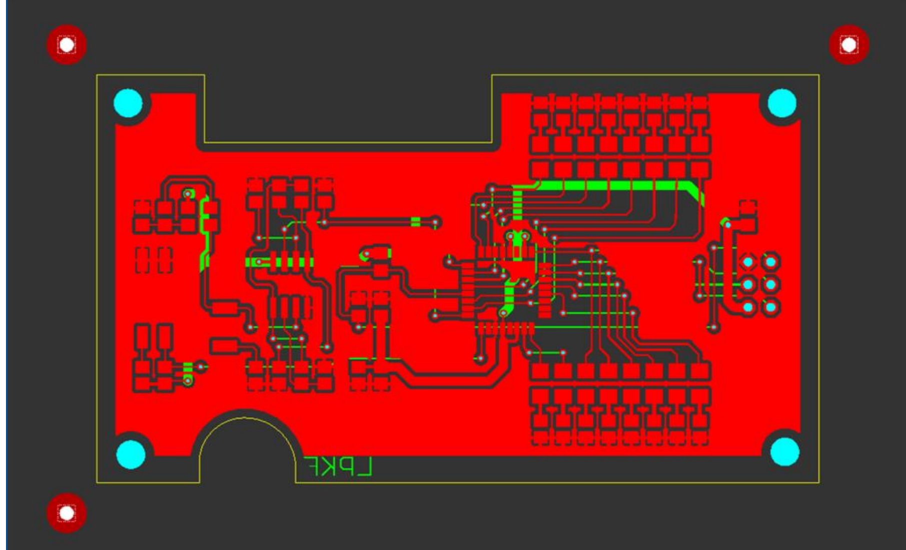


Fig. 284: Pasted layout

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.

Structuring the solder mask (with ProtoLaser)

1. Switch to the user guidance step *Processing*.
2. Click on ▶ 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

- Base material Aluminum oxide - Al_2O_3, 0.5 mm, 0/23 µm gold-plated

Auxiliaries

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

System

- ProtoLaser U4/S4/R4

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*.

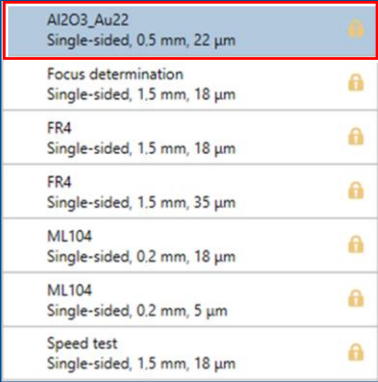


Fig. 285: Material selection

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:

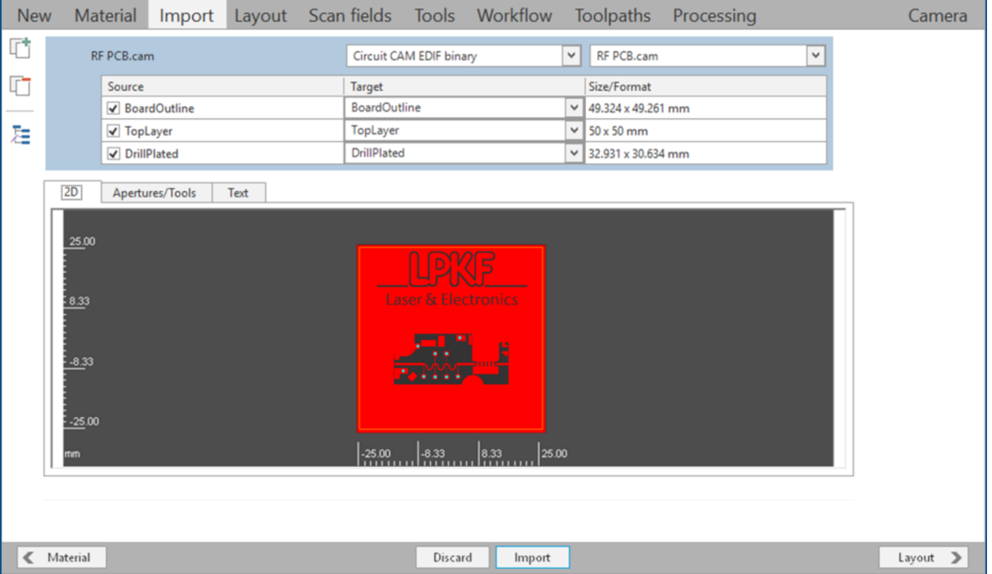


Fig. 286: User guidance step *Import*

11. Click on [Import].

- The layout is displayed in the user guidance step *Layout*.

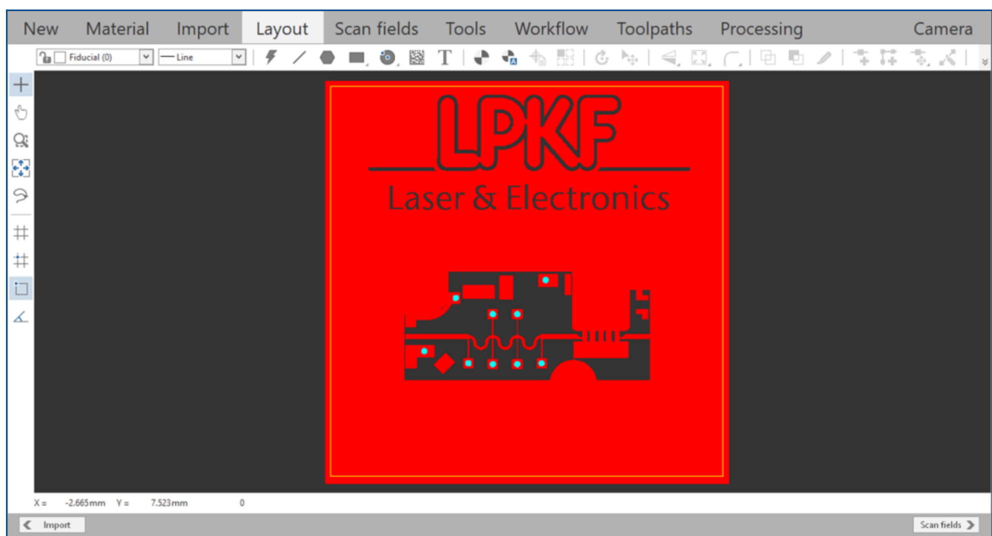


Fig. 287: Layout of an RF PCB

Before processing an RF PCB, the workflow settings need to be modified.

12. Switch to the user guidance step *Workflow*.

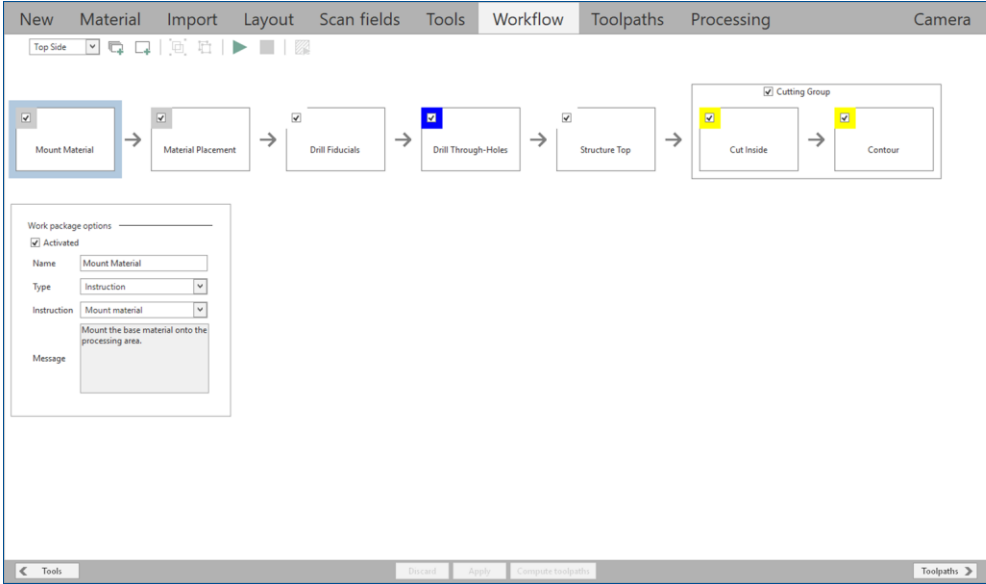


Fig. 288: User guidance step *Workflow*

13. Select the work package *Structure Top*.

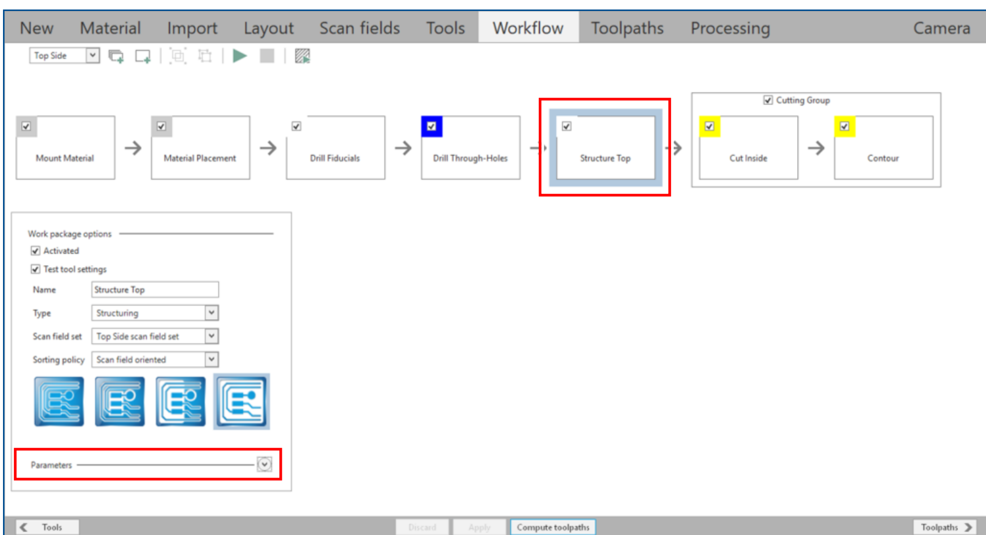


Fig. 289: Work package *Structure Top* selected

14. Expand the group *Parameters*.

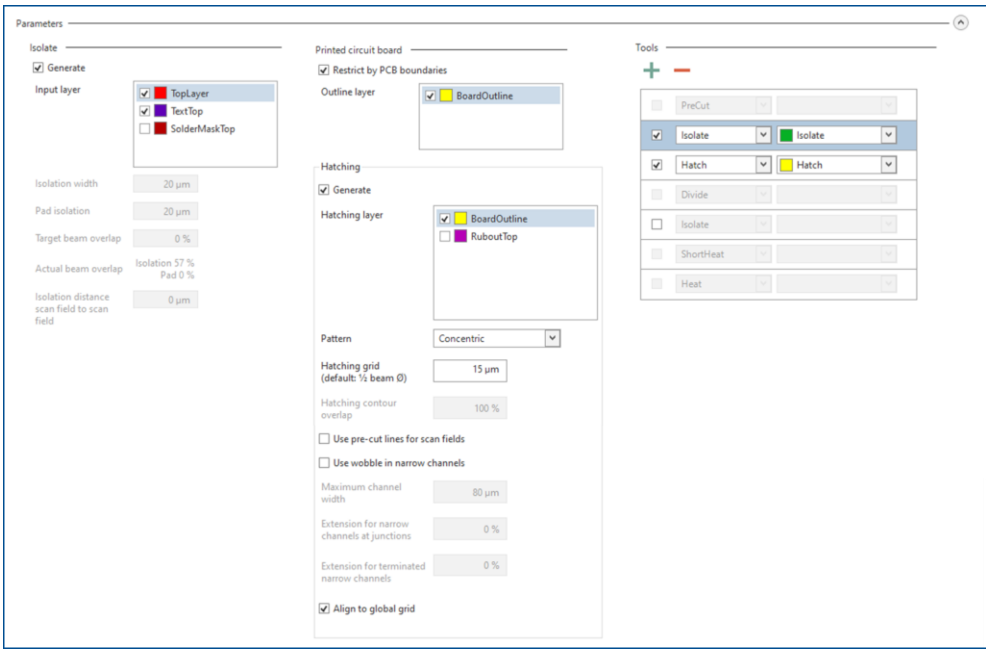


Fig. 290: Group *Parameters*

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:

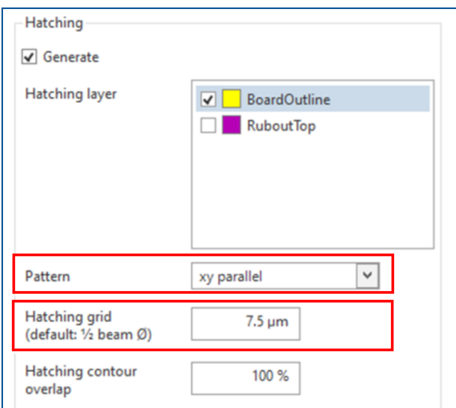


Fig. 291: Sub-group *Hatching*

The order of the tools *Isolate* and *Hatch* needs to be reversed for this how-to example.

17. Select the tool *Isolate*.

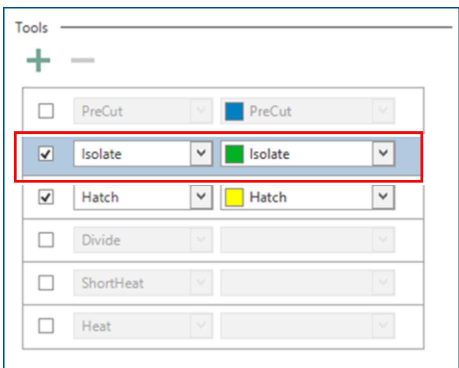


Fig. 292: Tool selected

18. Move it to the second position using drag & drop.

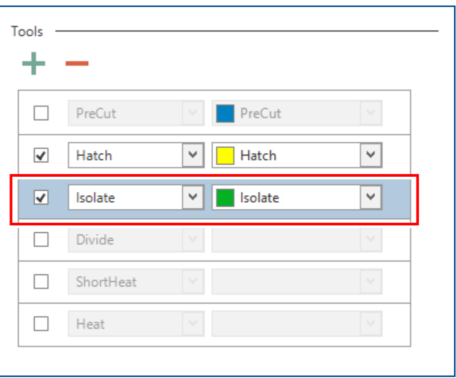


Fig. 293: Tool moved

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 .

- 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*.

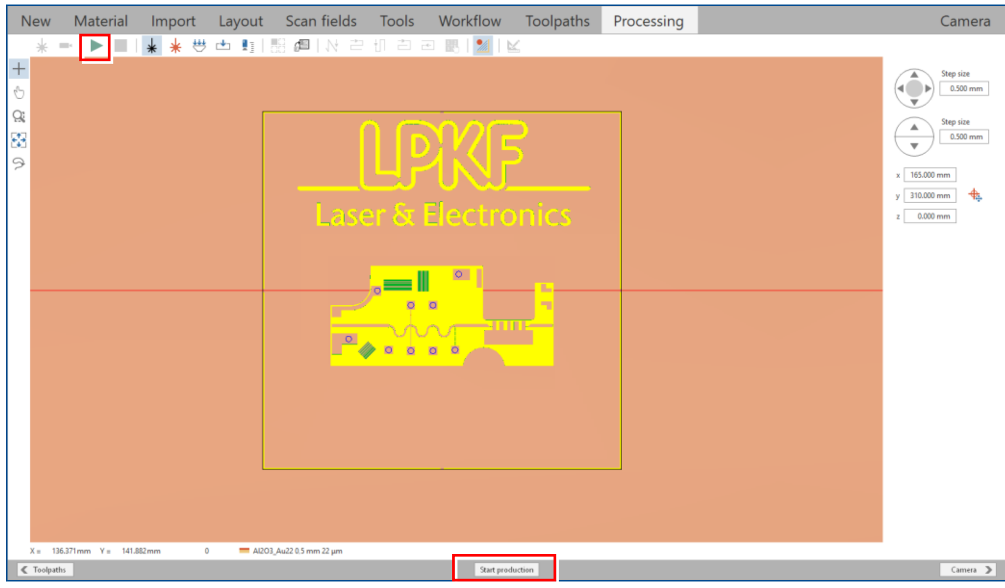


Fig. 294: User guidance step *Processing*

2. 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.*

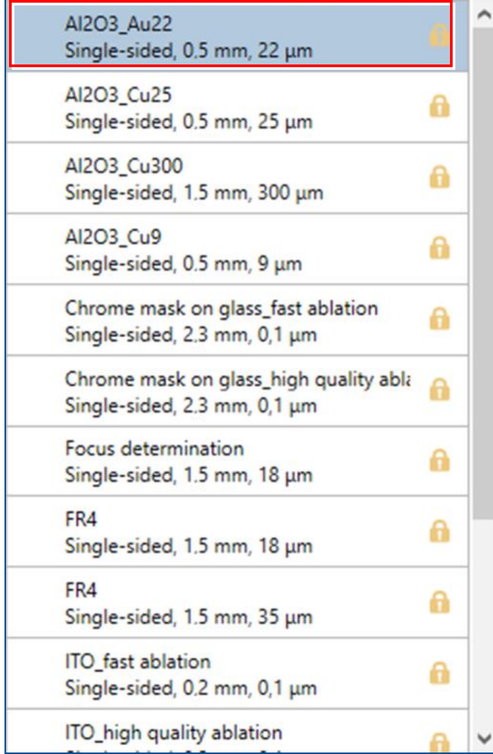


Fig. 295: Select material

4. Click on [Select material] or double-click on the material.

- ☐ The user guidance step *Import* is displayed.

5. Click on .
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*.

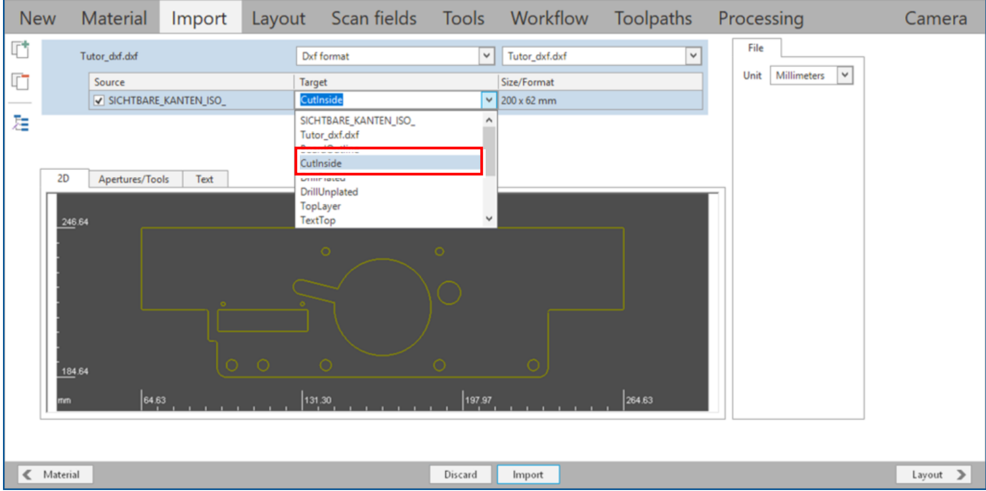


Fig. 296: **Select *CutInside***

- ☐ 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].

- ☐ The imported data is displayed in the user guidance step *Layout*.

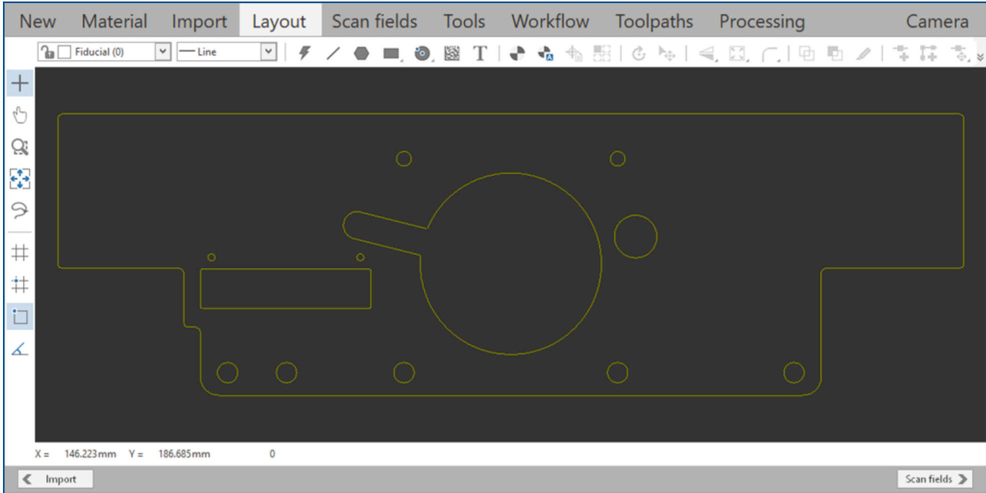


Fig. 297: **Imported data**

- ☑ The DXF file has been imported.

Combining open paths

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.

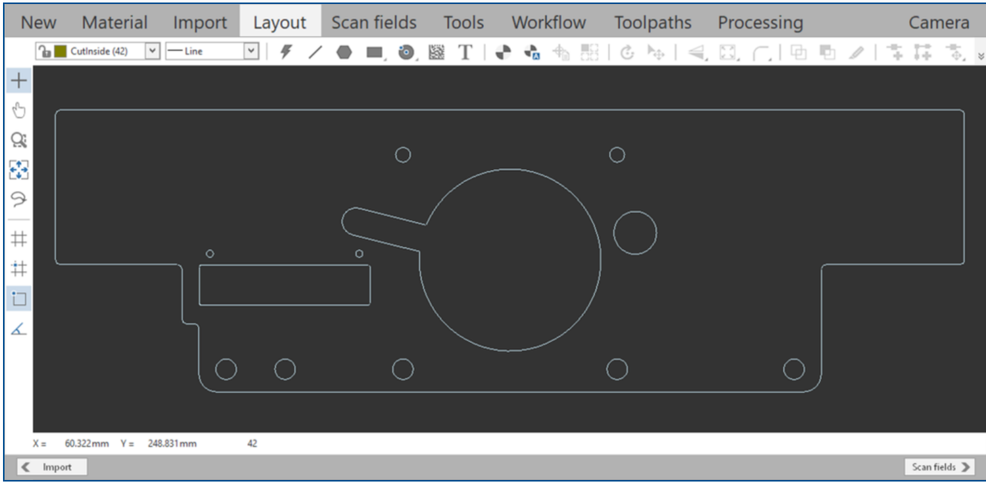


Fig. 298: Layout selected

2. Click on the right mouse button and in the context menu select the entry *Combine open paths*.

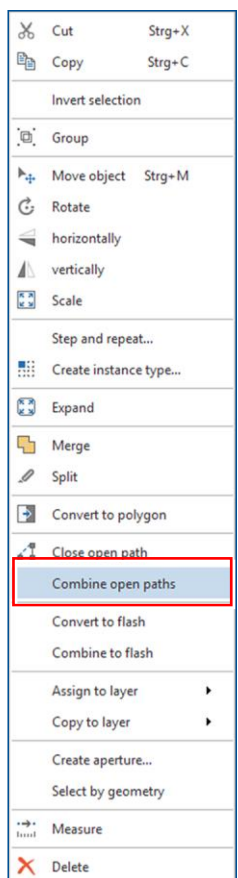


Fig. 299: Combine open paths

☑ The open paths have been combined.

Modifying the workflow settings

Before processing the DXF file, the workflow settings need to be modified.

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

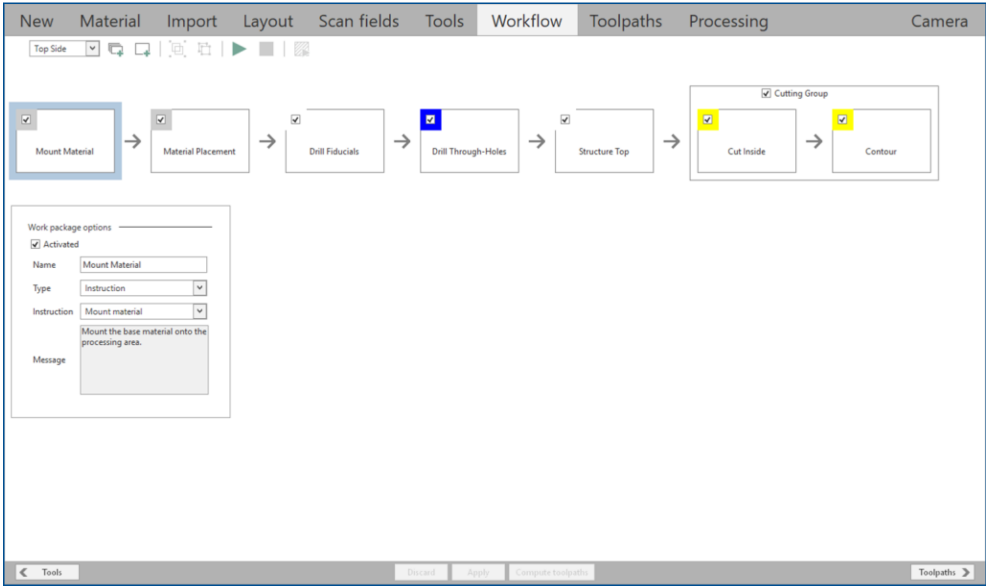


Fig. 300: User guidance step *Workflow*

2. Select the work package *Cut Inside*.

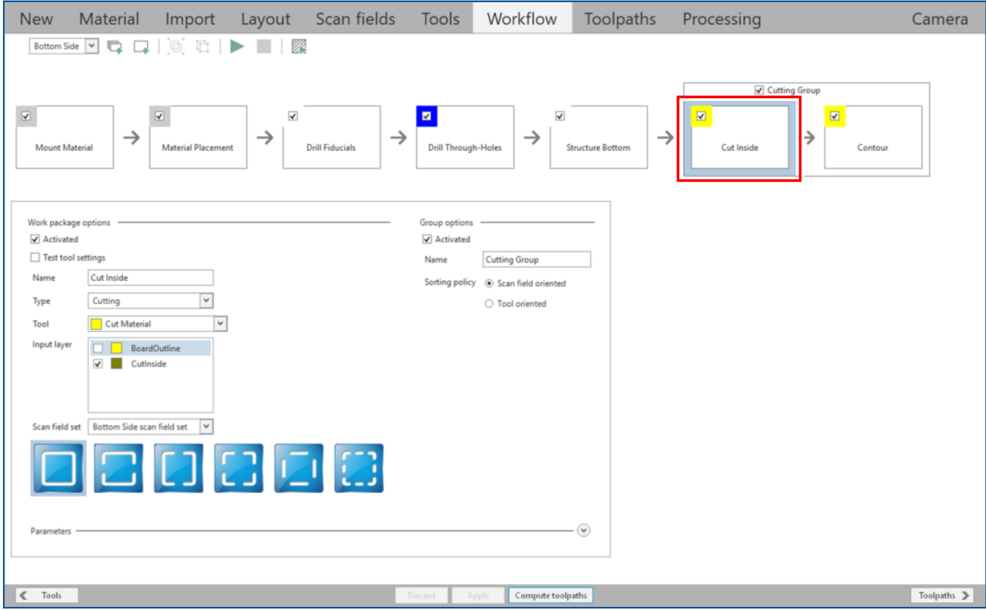


Fig. 301: Selecting the work package *Cut Inside*

3. Expand the group *Parameters*.

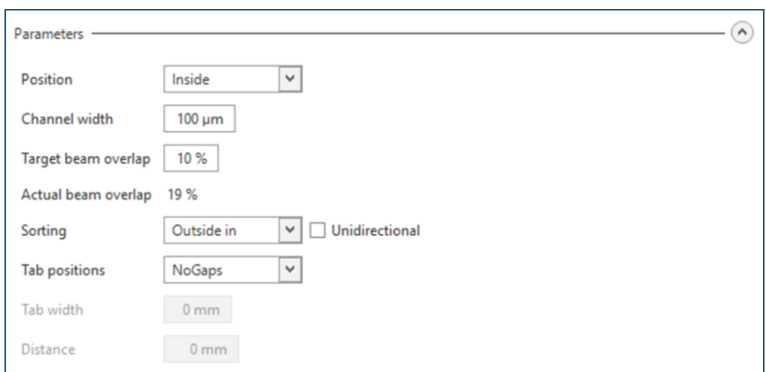


Fig. 302: Group *Parameters* expanded

4. Select the entry *Center* from the drop-down list *Position*.

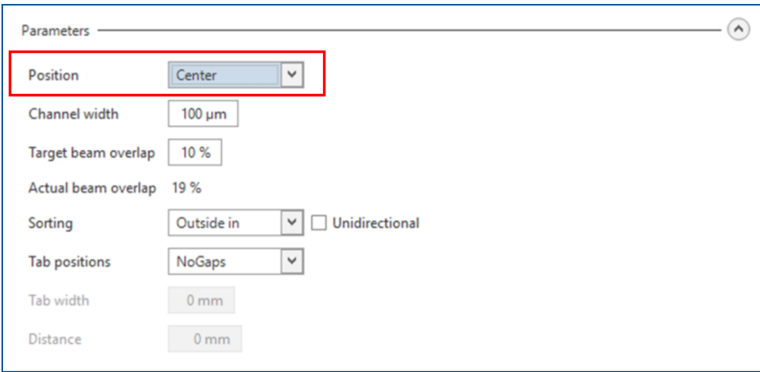


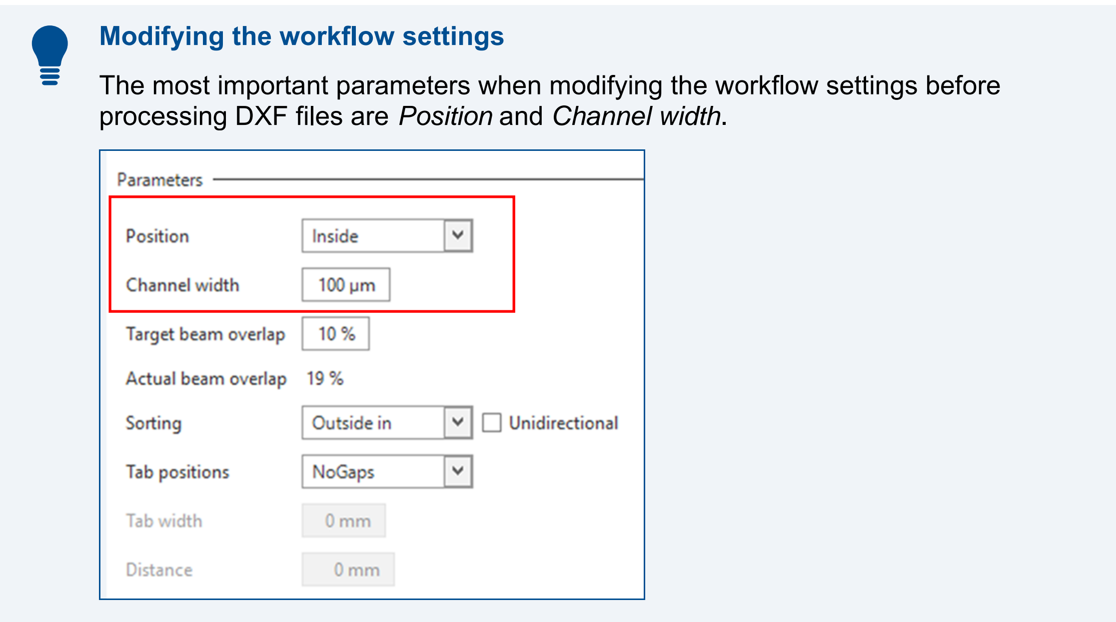
Fig. 303: Entry *Center* selected

5. Click on [Apply].

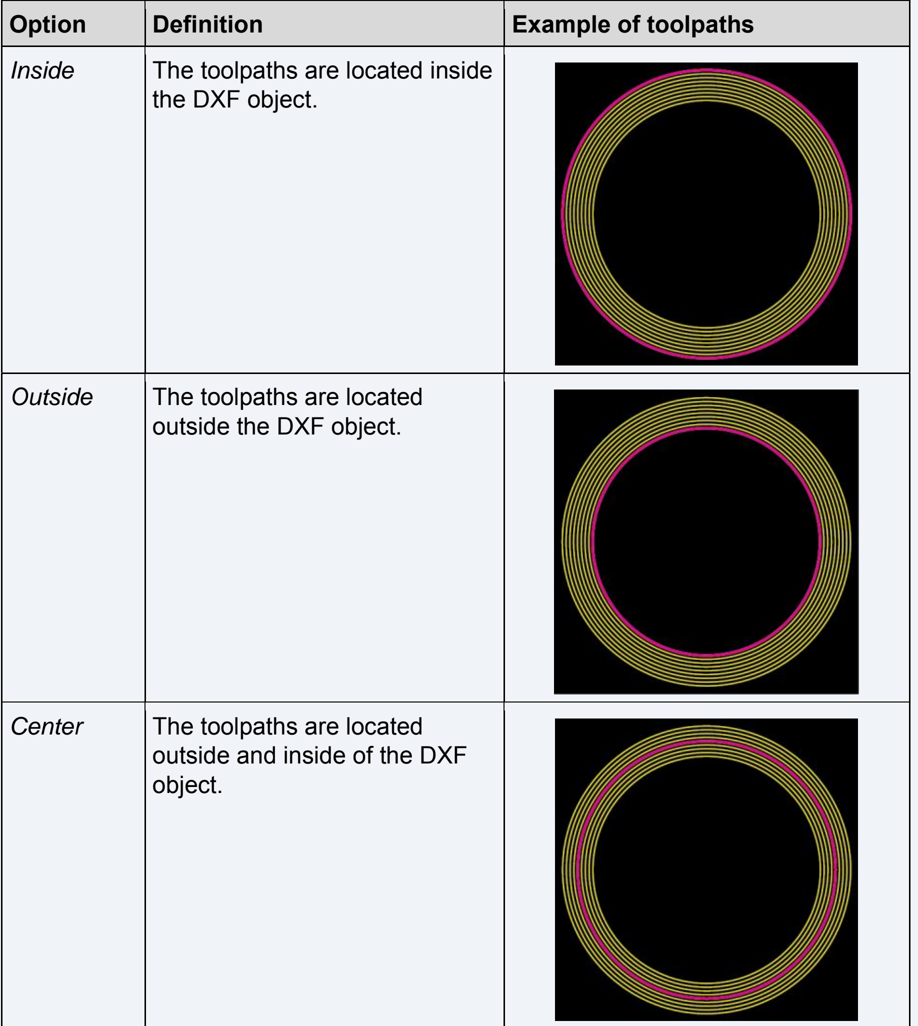
☑ The workflow settings have been modified.

Modifying the workflow settings

The most important parameters when modifying the workflow settings before processing DXF files are *Position* and *Channel width*.

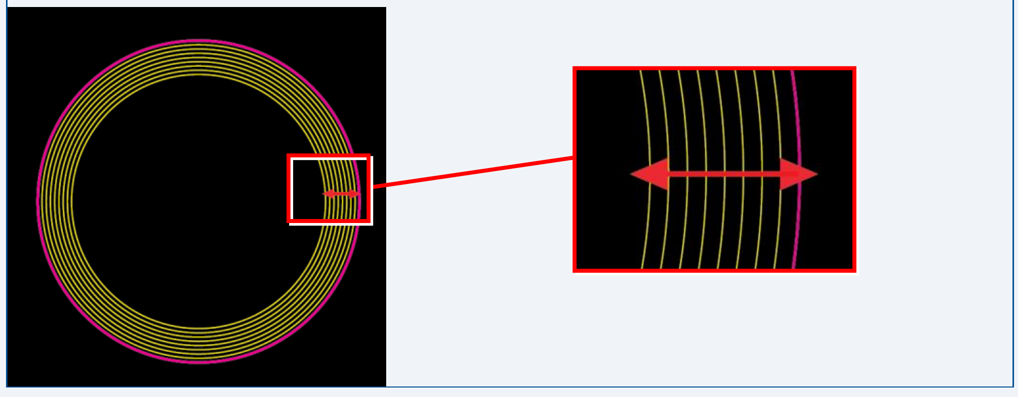


The parameter *Position* defines the position of the toolpaths (in yellow) in comparison to the DXF object (in magenta). There are three options:



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 *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 µm* is used.

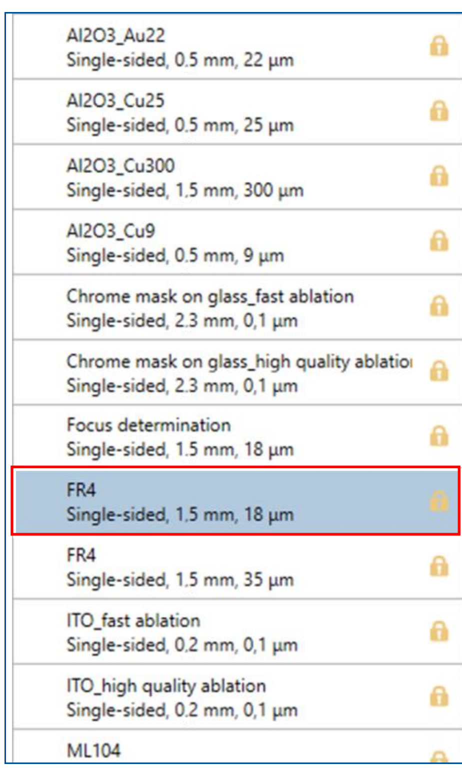


Fig. 304: Selecting the material

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.

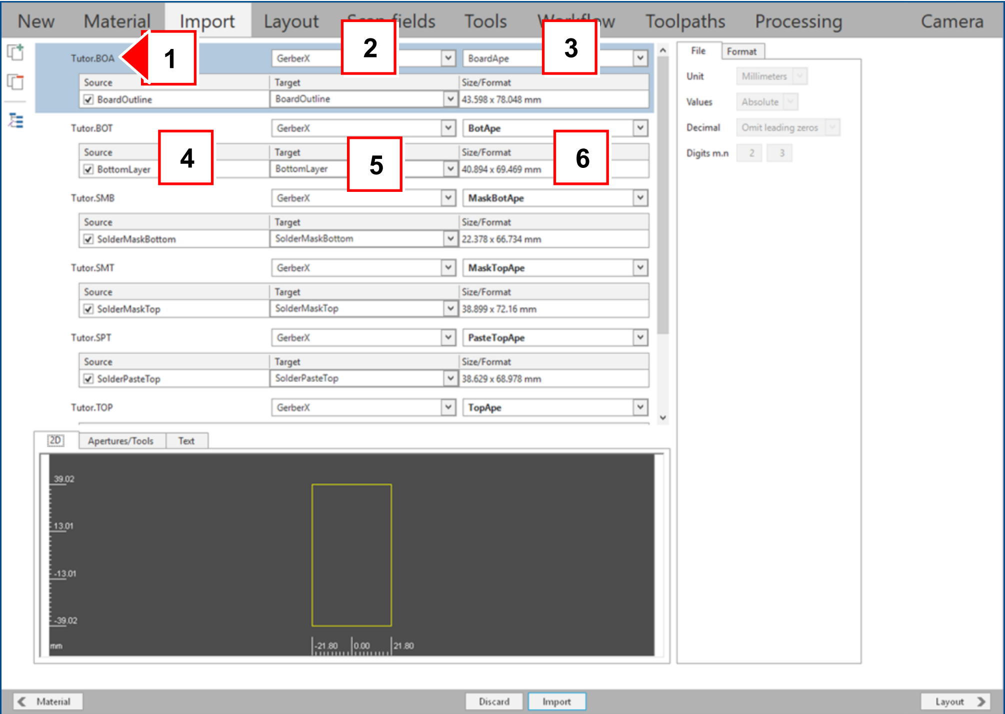


Fig. 305: User guidance step *Import*

The following information on the imported files is displayed:

Item	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 *Apertures/Tools*.
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:

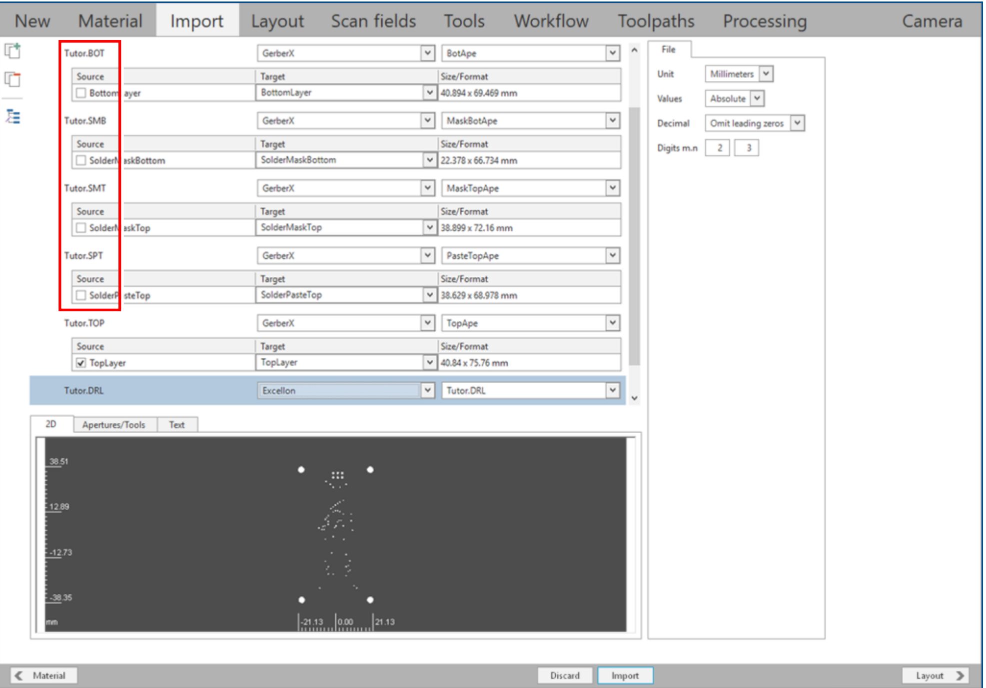


Fig. 306: Deselected files

☑ 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:

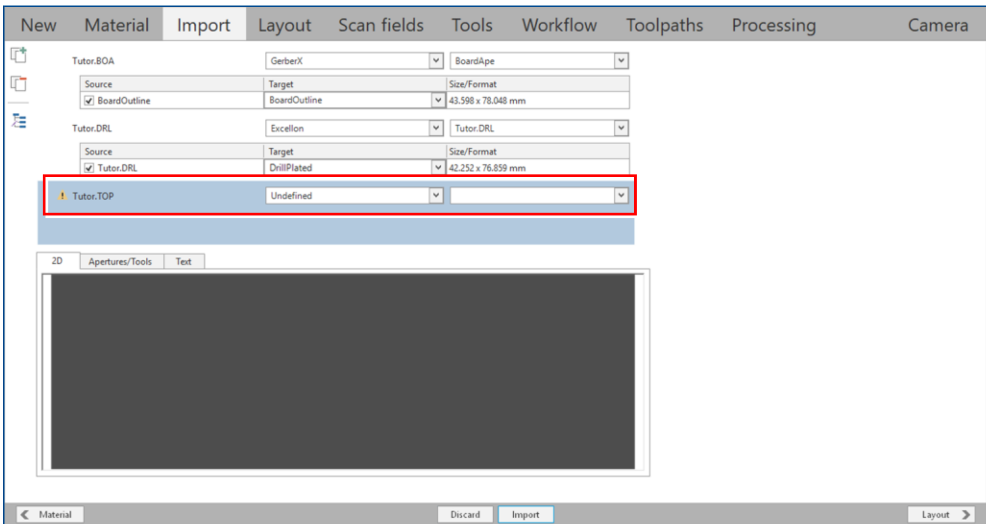


Fig. 307: Format undefined

3. Select the appropriate file format from the drop-down list (in this example *GerberX*):

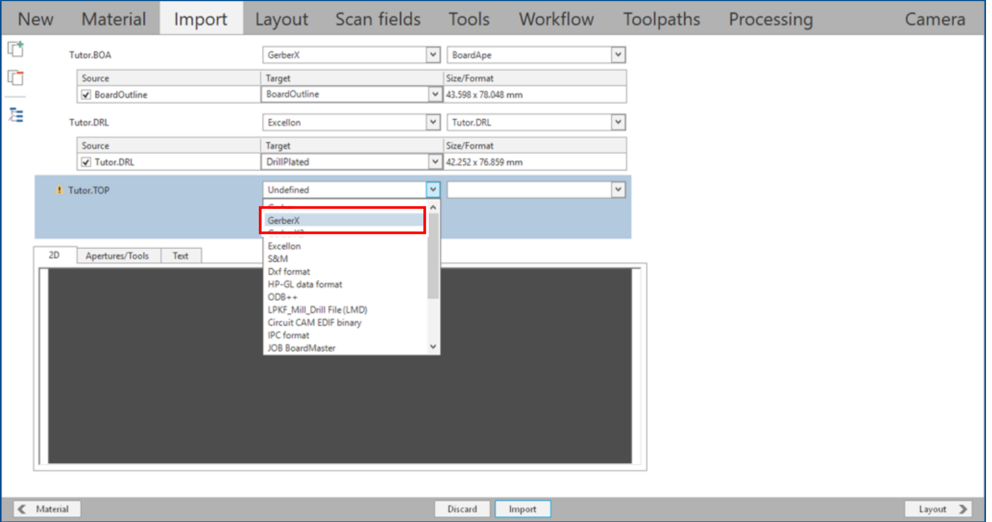


Fig. 308: Selecting the appropriate file format

☐ The missing file-specific information is displayed automatically.

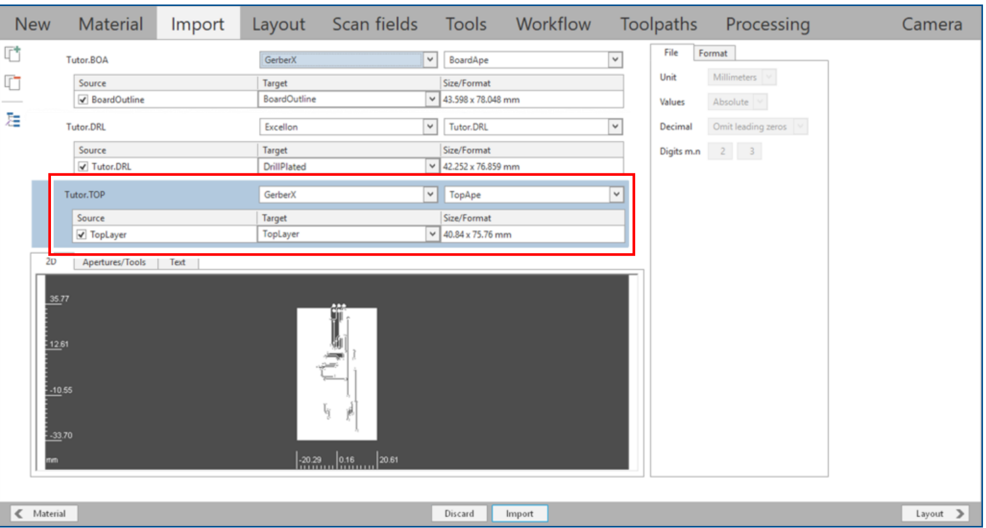


Fig. 309: Appropriate file format selected

☑ 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):

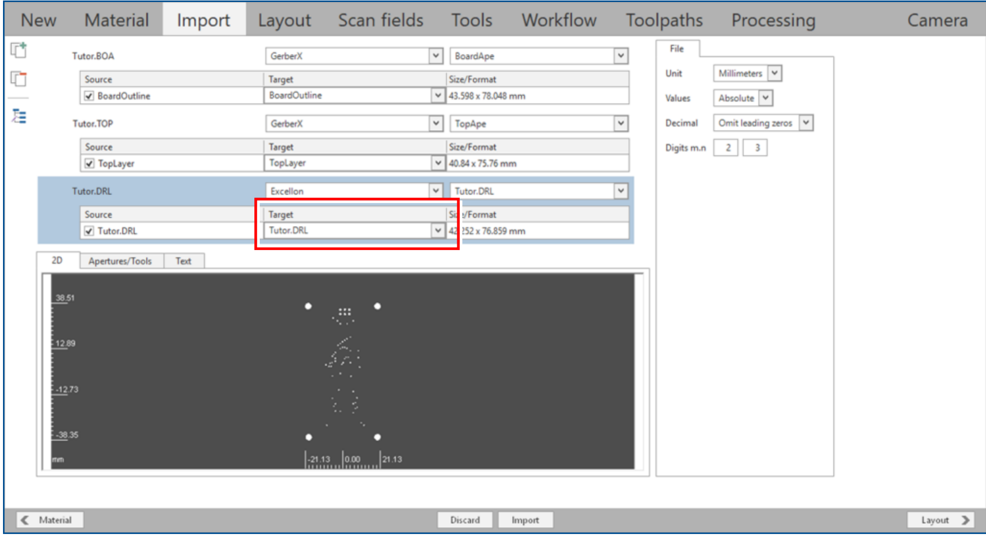


Fig. 310: Unassigned layer

3. Select the desired target layer from the drop-down list (in this example *DrillPlated*):

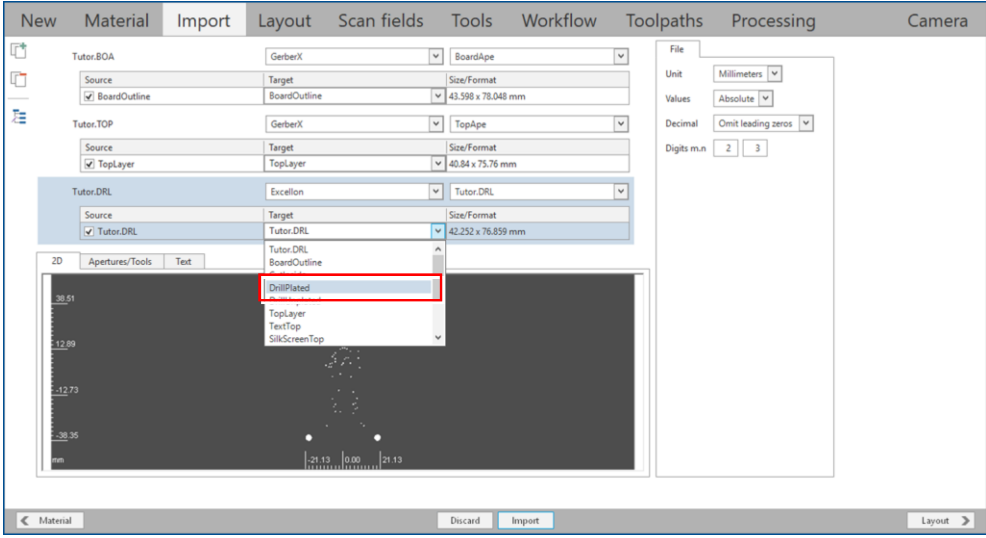


Fig. 311: Selecting the target layer

☐ The user guidance step *Import* changes as follows:

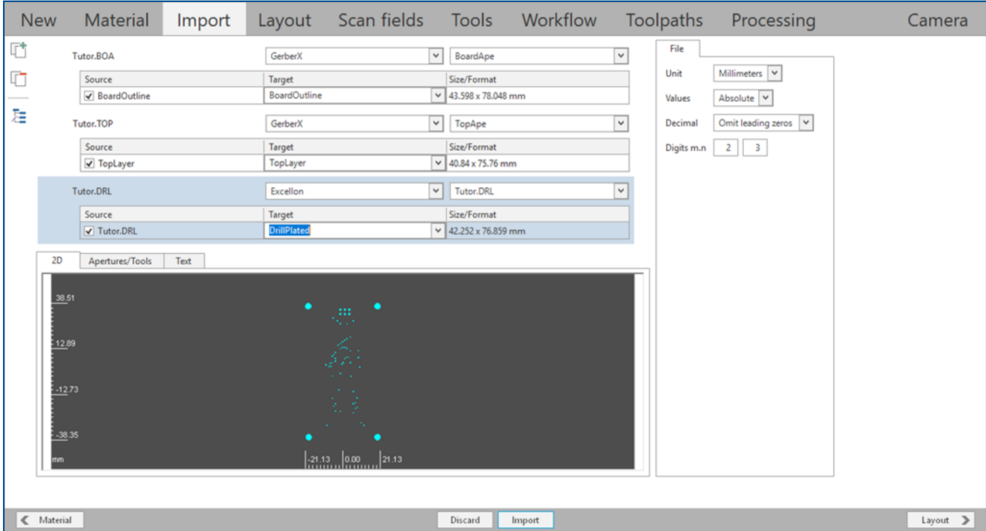


Fig. 312: Selected target layer

4. Click on [Import].

☐ The processing data are displayed in the user guidance step *Layout*.

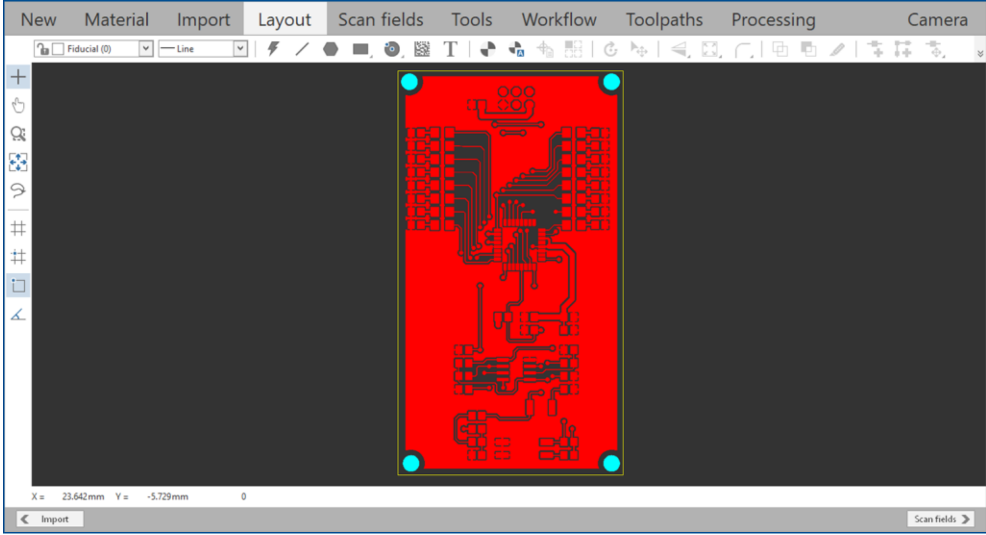


Fig. 313: User guidance step *Layout*

☑ 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:

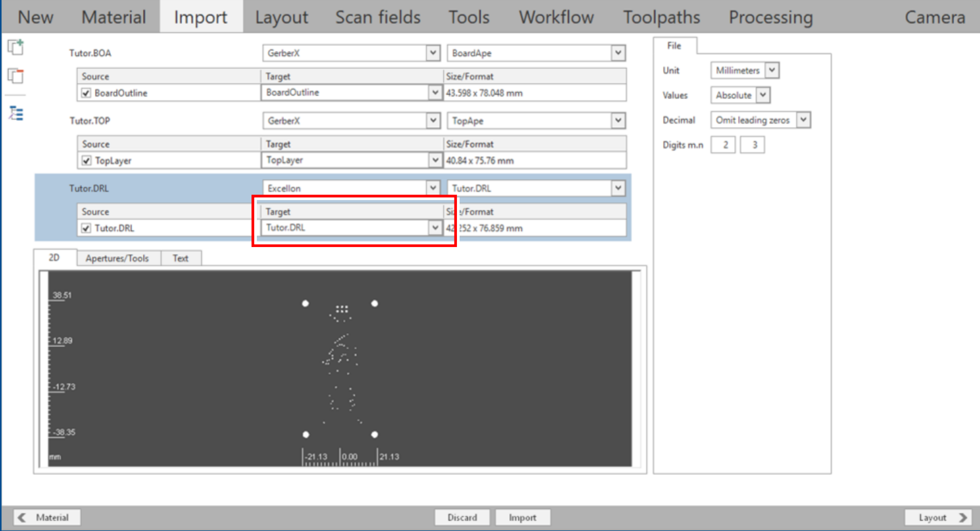


Fig. 314: **Unassigned layer**

3. Click on [Import].

☐ The following message is displayed:

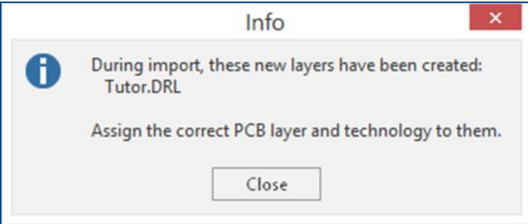


Fig. 315: **Info about new layer created**

4. Click on [Close].

- ☐ A new layer *Tutor.DRL* has been created and the processing data are displayed in the user guidance step *Layout*:

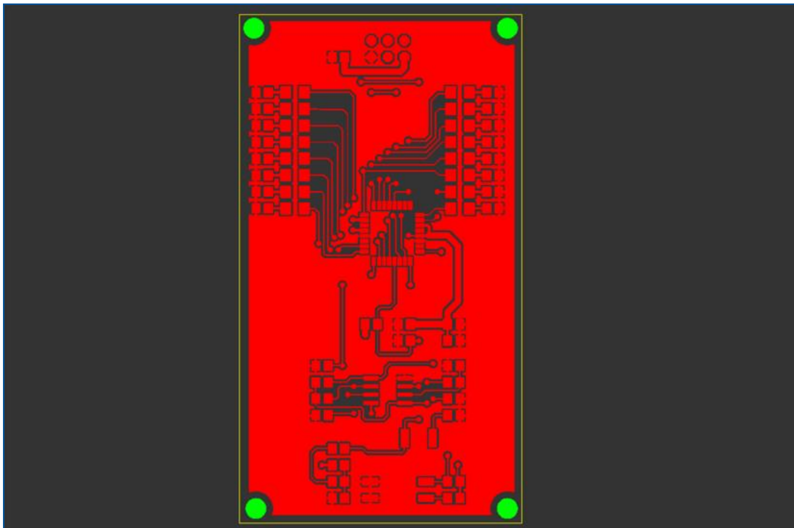


Fig. 316: User guidance step *Layout* | processing data

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*.

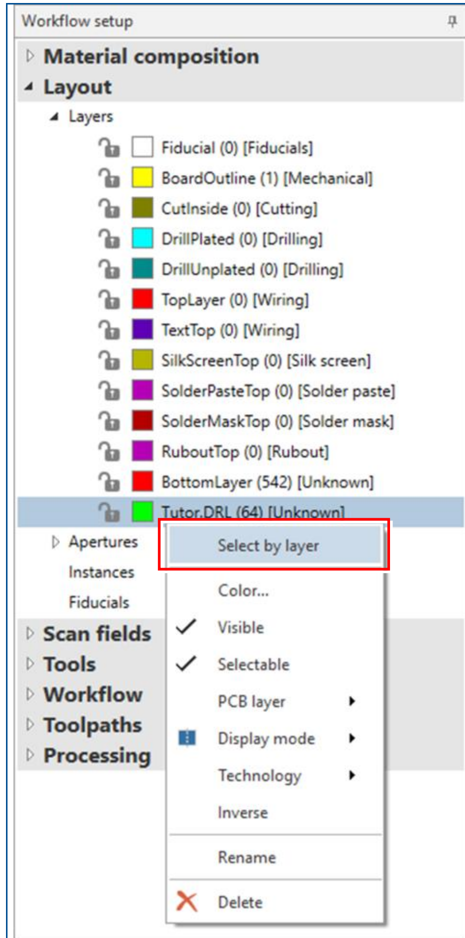
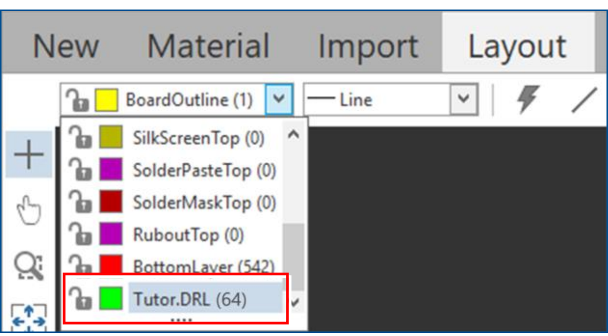


Fig. 317: List of layers | *Select by layer*

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*:

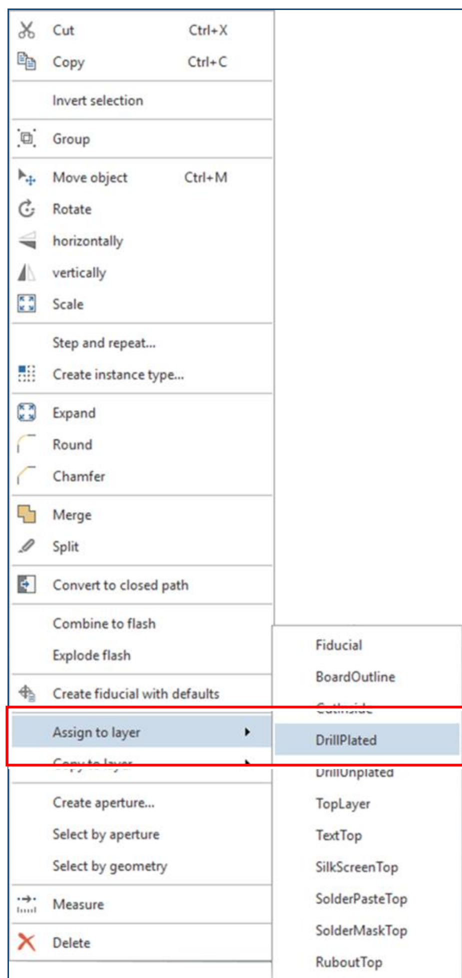


Fig. 318: Assigning objects to target layer

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.

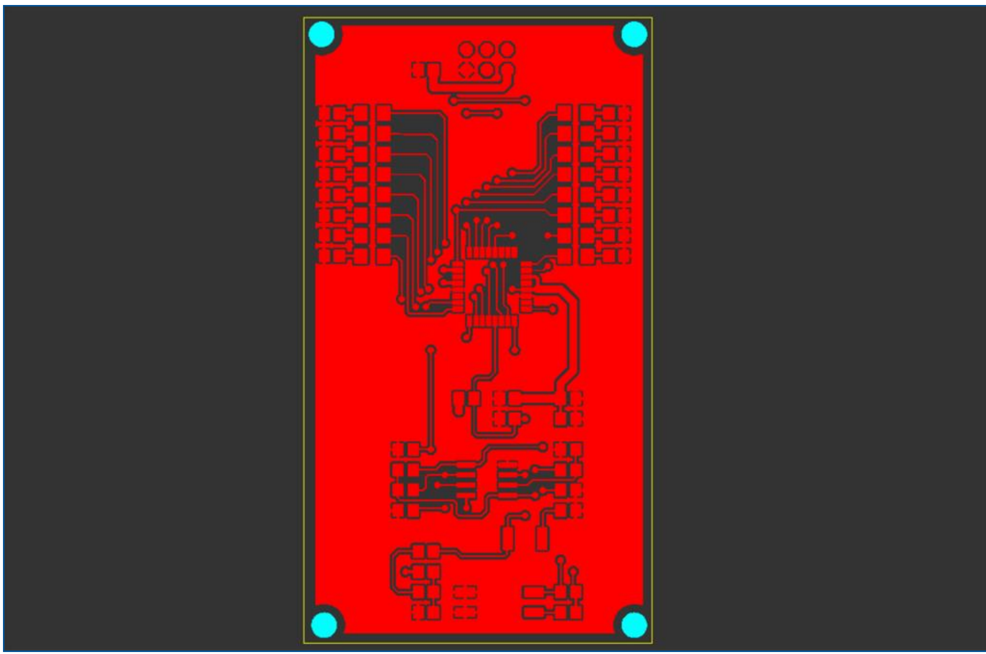


Fig. 319: Assigned holes

☑ 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.

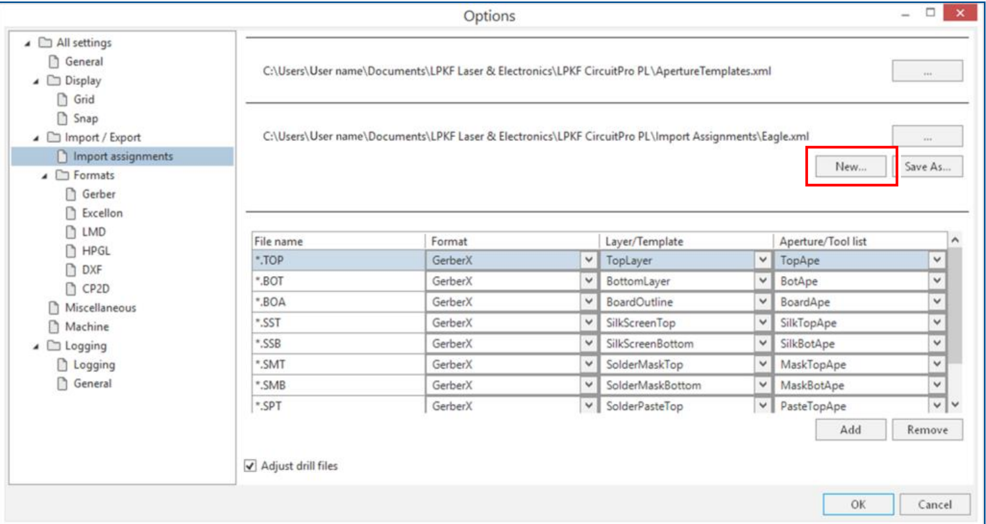
Perform the following steps:

1. Click on *File* > *Options…*

☐ The dialog *Options* is displayed.

2. Click on *Import assignments*.

☐ The dialog changes as follows:

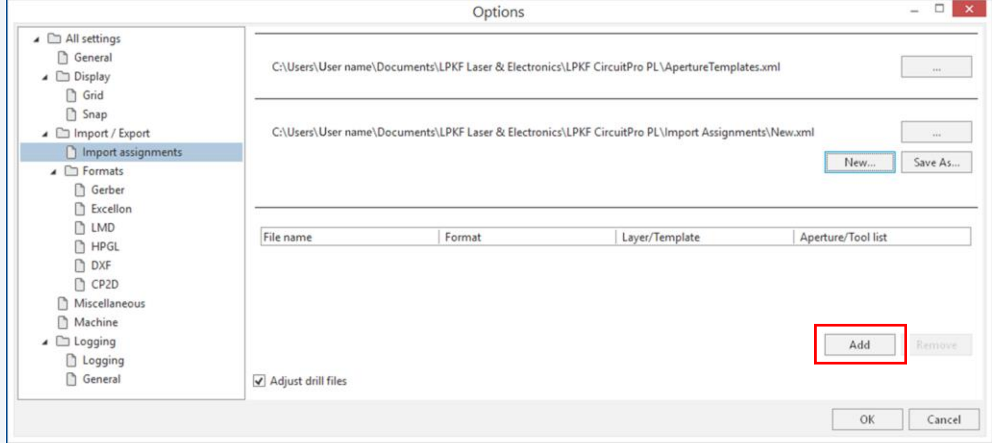


3. Click on [New].

☐ The dialog *Save As* is displayed.

4. Enter the new name of the .xml file and click on [Save].

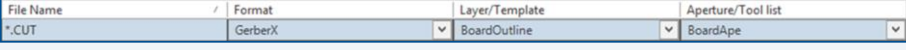
☐ The dialog changes as follows:



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*:



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:

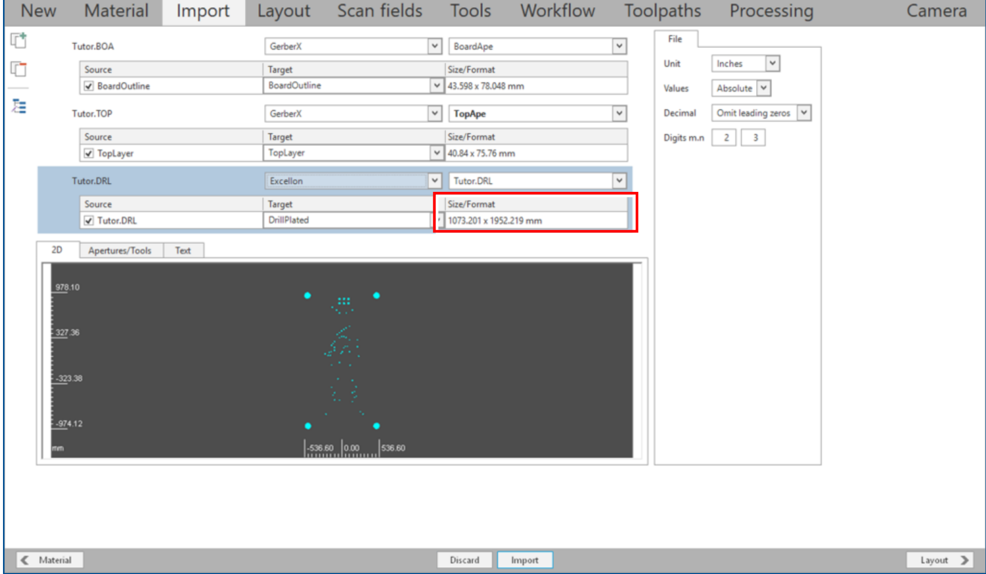


Fig. 320: Large size of the layout

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).
4. Select *Millimeters* from the drop-down list.

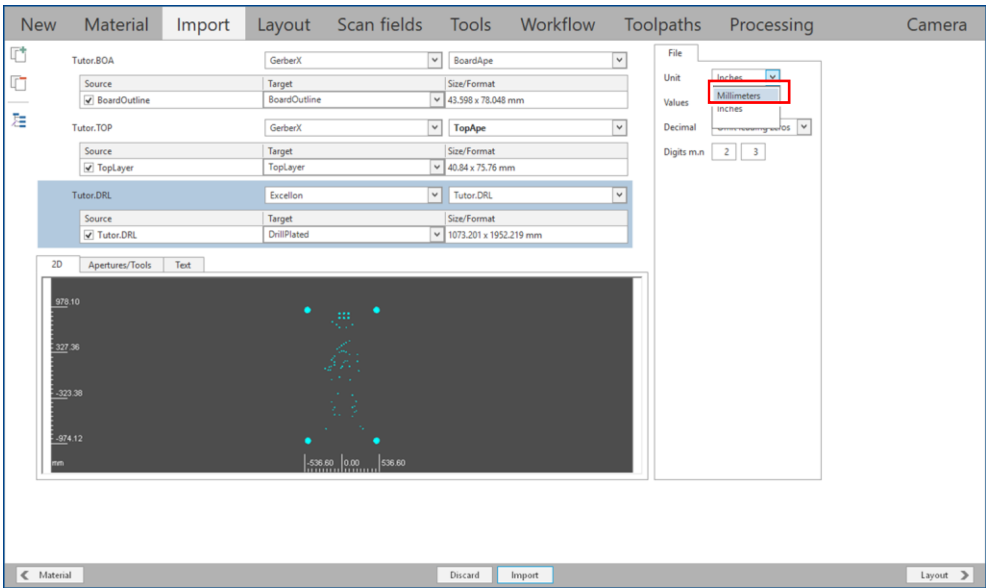


Fig. 321: Selecting the appropriate unit

☐ The dimensions of the layout change automatically:

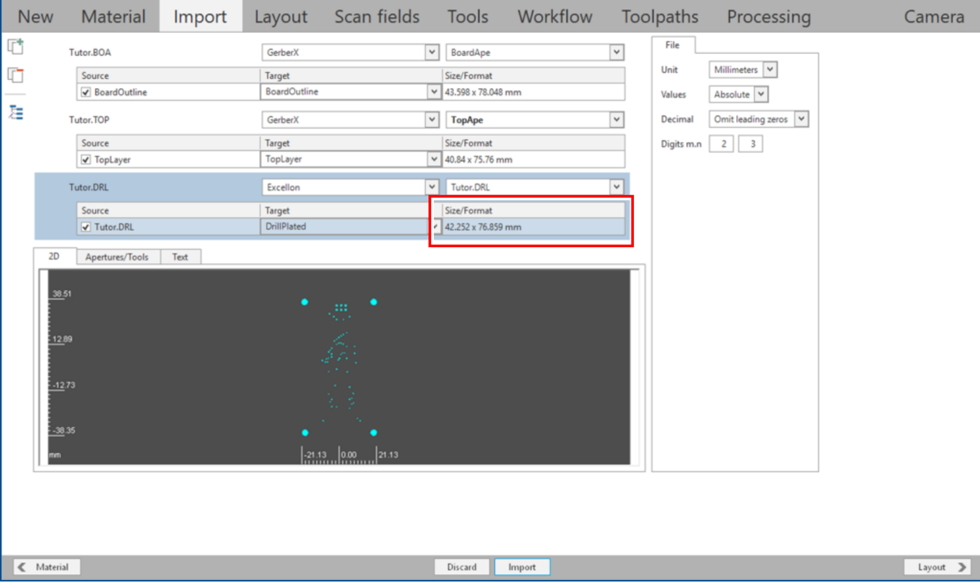


Fig. 322: Changed dimensions of the layout

☐ 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:

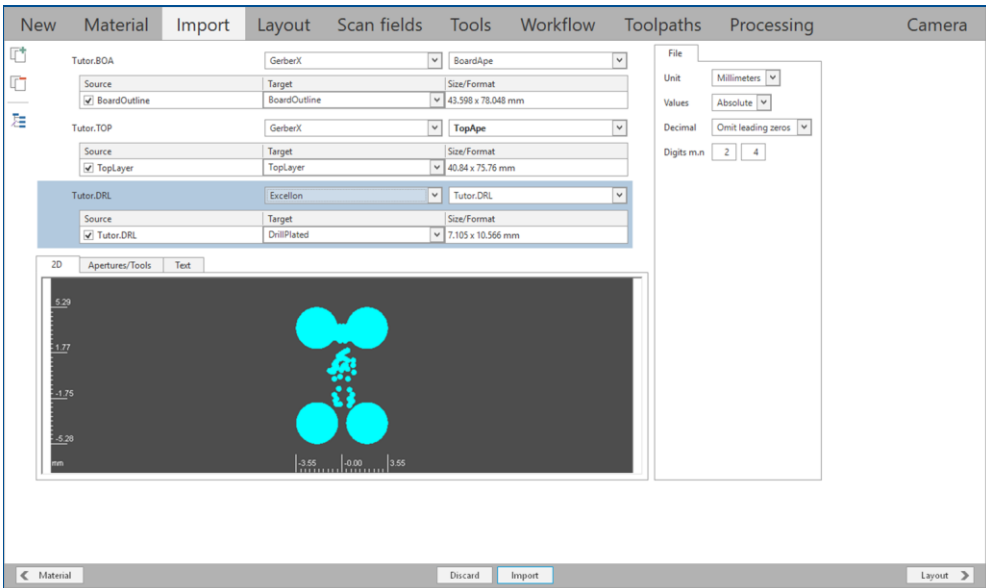


Fig. 323: Incorrect preview of the layout

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:

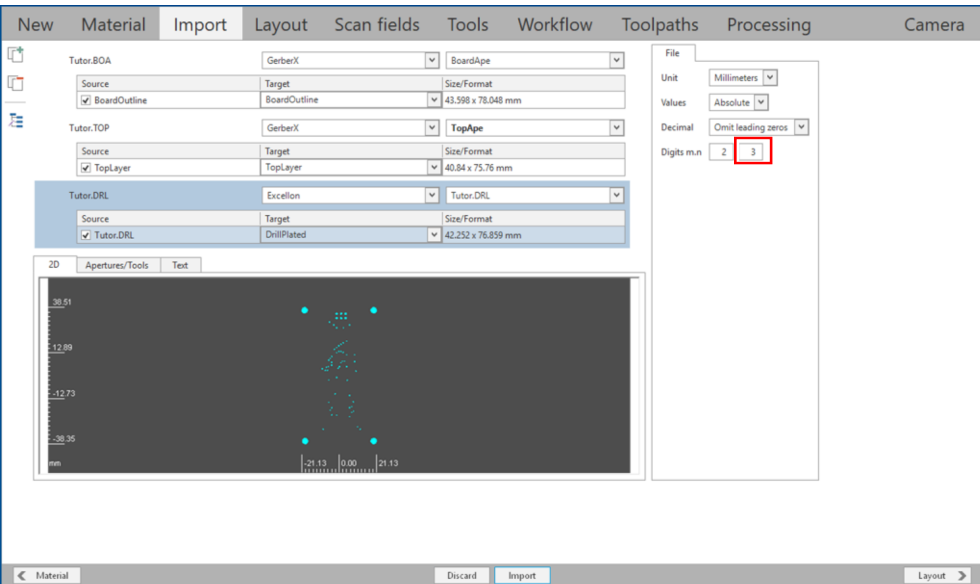


Fig. 324: Correct preview of the layout

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

☑ The size and format have set/corrected.

Checking/Modifying aperture properties

1. In the user guidance step *Import*, select the file whose apertures you want to check (in this example *Tutor.BOA*).

☐ The user guidance step *Import* changes as follows:

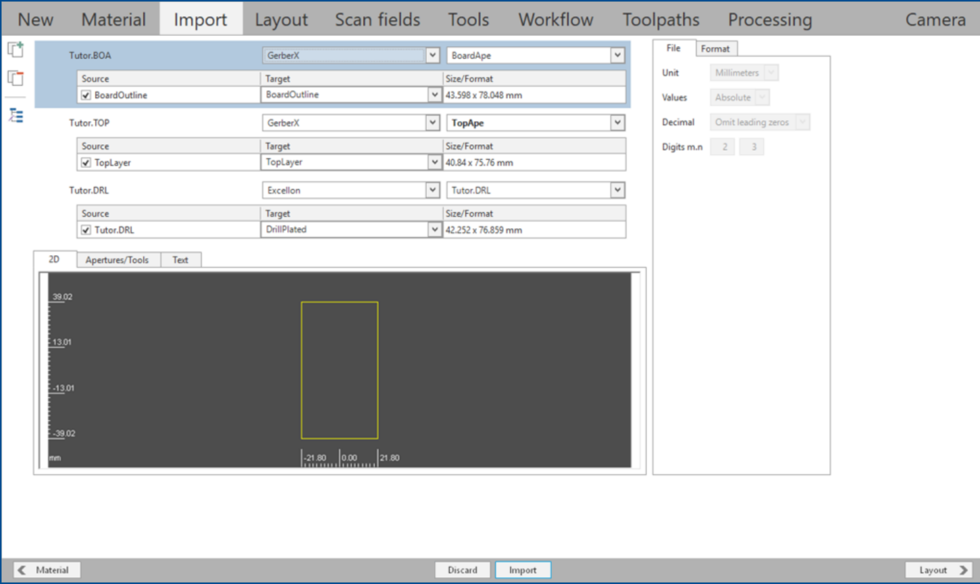


Fig. 325: File *Tutor.BOA* selected

2. Click on the tab *Apertures/Tools* and then on the sub-tab *Attributes*.

☐ The user guidance step *Import* changes as follows:

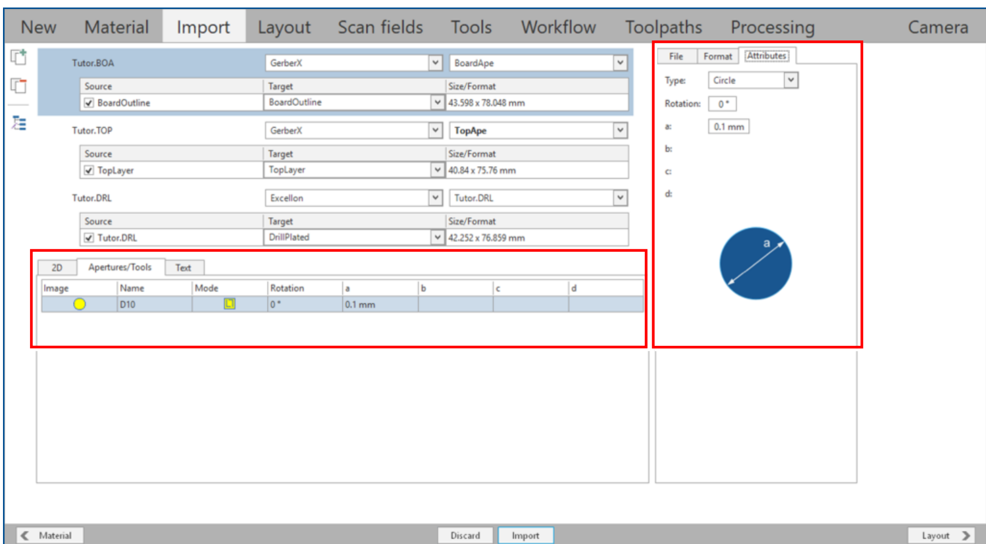


Fig. 326: Tab *Apertures/Tools* and sub-tab *Attributes*

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.

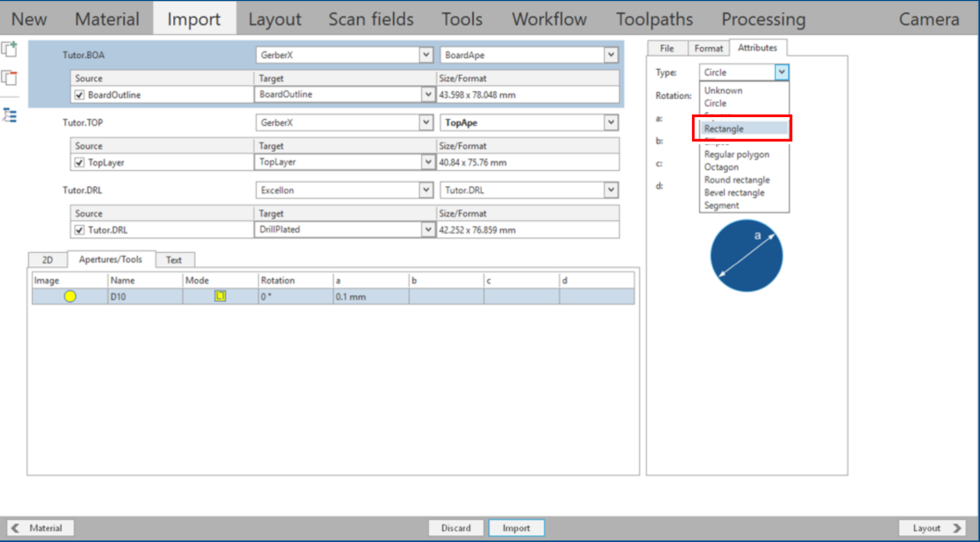


Fig. 327: Selecting the type of aperture

4. Enter *1.5* in the input field *a:*.

☐ A preview of the modified aperture is displayed in the sub-tab *Attributes*:

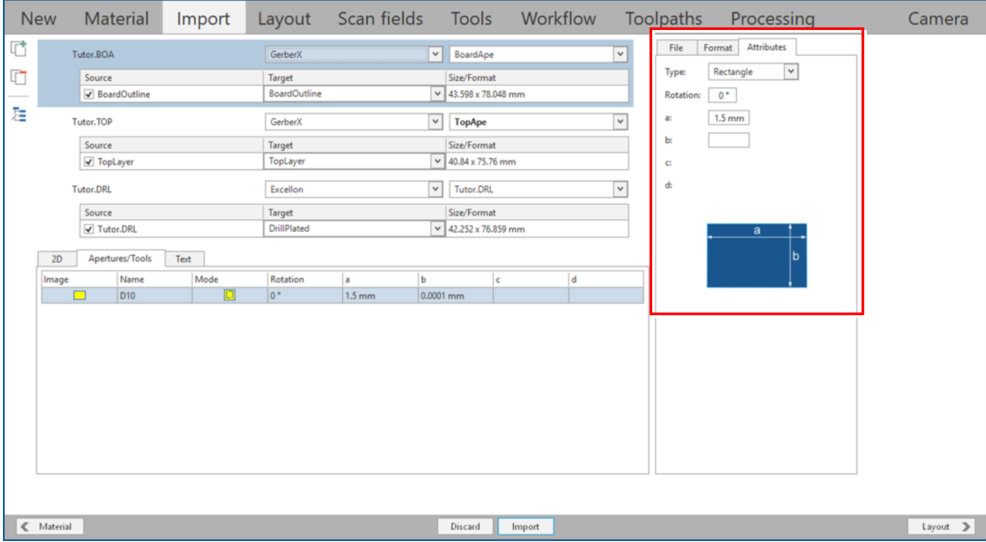


Fig. 328: Preview of the modified aperture

☑ 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):

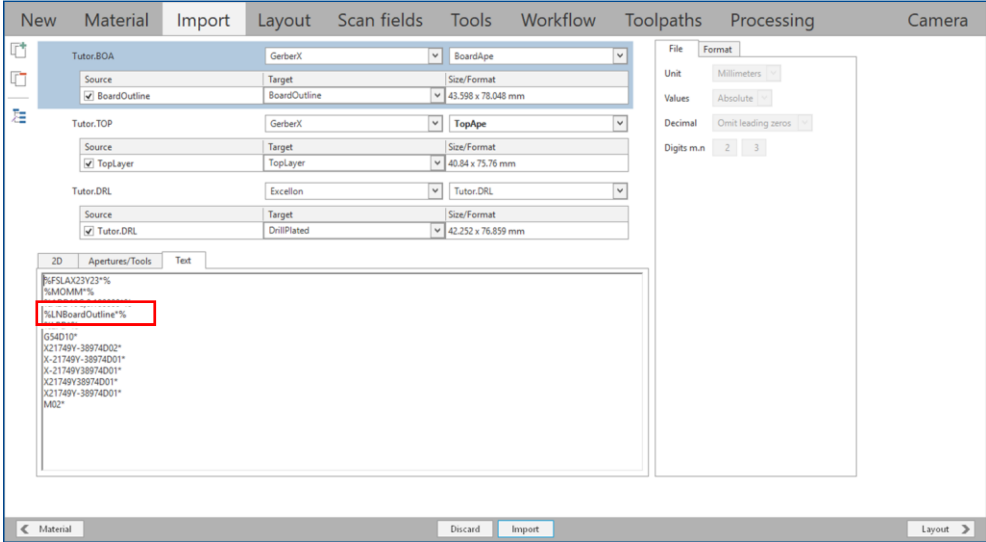


Fig. 329: Layer name in the file *Tutor.BOA*

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*:

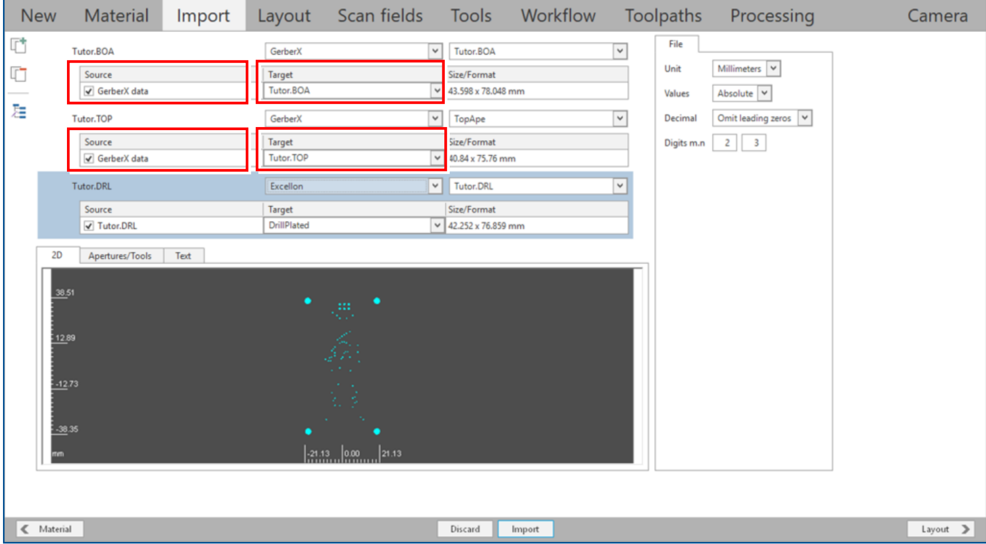


Fig. 330: Layer names not used in the Gerber file

3. Select one of the Gerber files (in this example *Tutor.BOA*) and switch to the sub-tab *Format*.

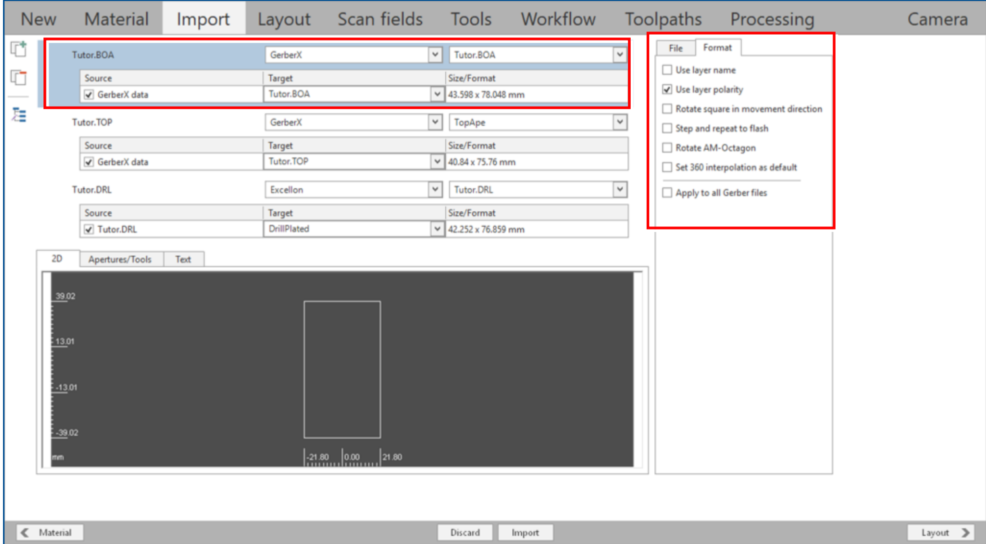


Fig. 331: Gerber file selected

4. Activate the check boxes *Use layer name* and *Apply to all Gerber files*.

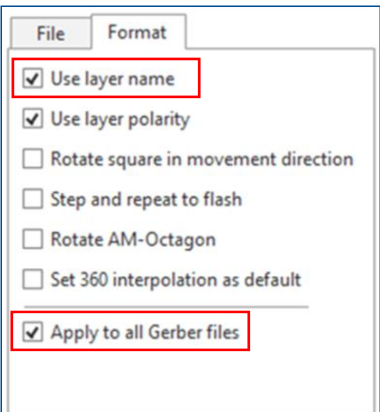


Fig. 332: Activated check boxes

- ☐ The layer names defined in the Gerber files are displayed in the column *Target* and in the column *Source*:

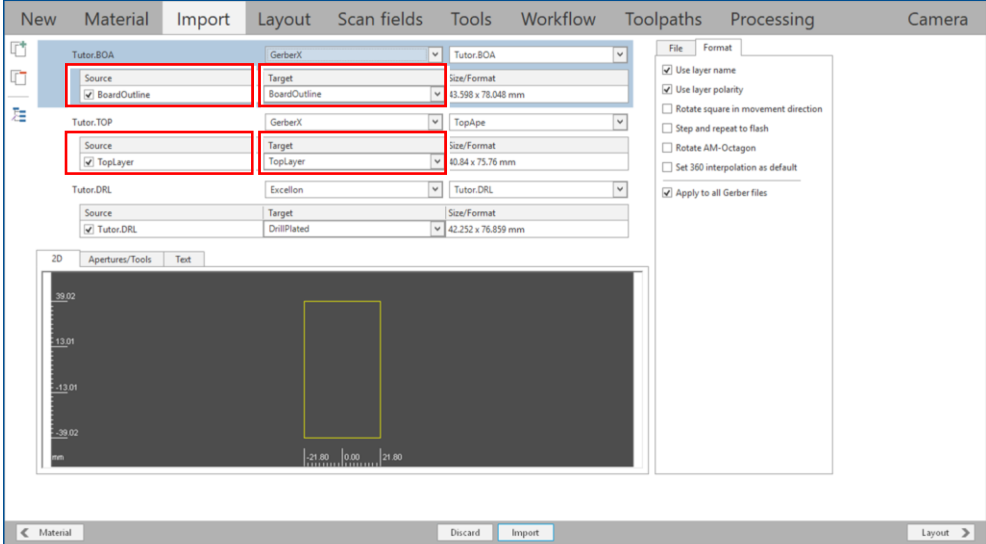


Fig. 333: **Layer names defined in Gerber files displayed**

- ☑ 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*.

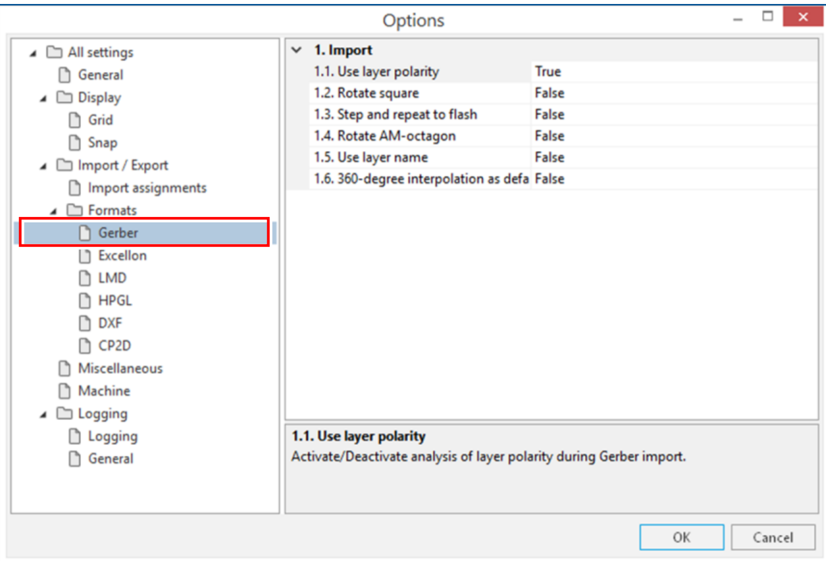


Fig. 334: **Dialog *Options* | *Gerber***

3. Set the value in line *1.5 Use layer name* to *True*.

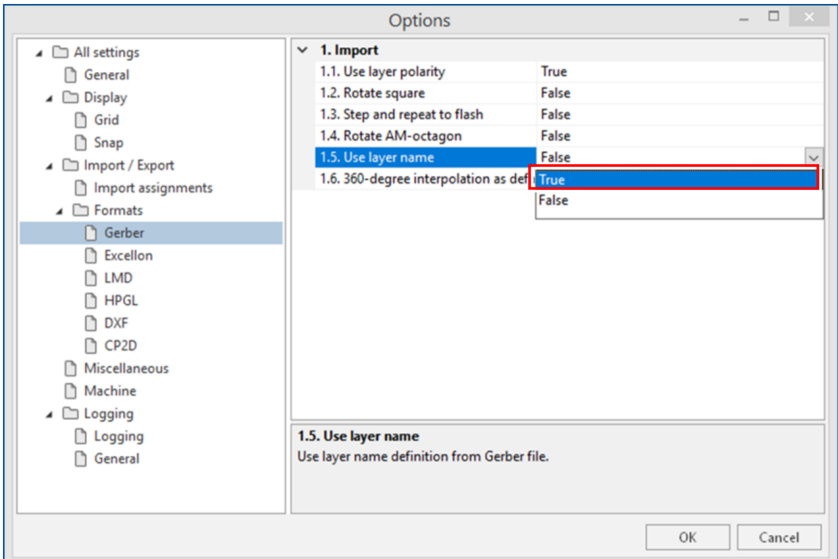


Fig. 335: Selecting value *True* for *Use layer name*

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*:

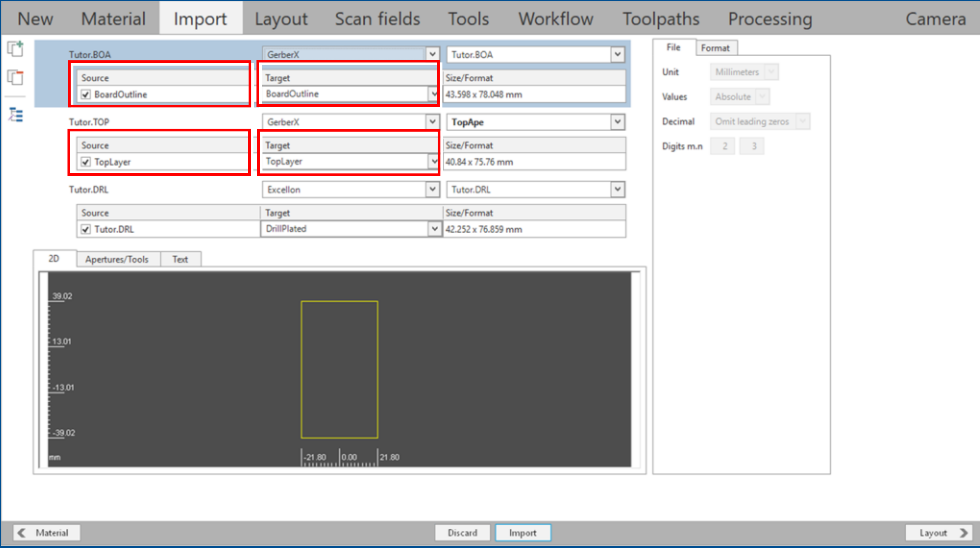


Fig. 336: Layer names defined in Gerber files displayed

☑ 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* ... 12
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 ... 17
Fig. 10: Fiducial of poor quality ... 19
Fig. 11: Matched position of poor quality fiducial ... 19
Fig. 12: Dialog *Material thickness measurement settings* (default settings) ... 20
Fig. 13: Drop-down list *Measurement strategy* ... 22
Fig. 14: Dialog *Placement* ... 23
Fig. 15: Message *Thickness measurement running* ... 23
Fig. 16: Dialog *Measurement result confirmation* ... 24
Fig. 17: Context menu ... 25
Fig. 18: Dialog *Placement* ... 25
Fig. 19: Message *Thickness measurement running* ... 26
Fig. 20: Measured value ... 26
Fig. 21: Entering *Material thickness* ... 27
Fig. 22: Measured values and average value of the reference points ... 28
Fig. 23: Select material ... 29
Fig. 24: User guidance step *Import* ... 31
Fig. 25: Imported PCB layouts ... 32
Fig. 26: One PCB layout selected ... 32
Fig. 27: Menu *Edit instance type* ... 33
Fig. 28: Dialog *Edit instance type* ... 33
Fig. 29: Dialog *Edit instance type* | Values ... 33
Fig. 30: Layout multiplied ... 34
Fig. 31: User guidance step *Processing* ... 34
Fig. 32: Automatic arrangement of PCB layouts ... 35
Fig. 33: Menu item *Placement* ... 35
Fig. 34: Dialog *Placement* | *Processing data* ... 36
Fig. 35: Distance between PCB layouts increased ... 36
Fig. 36: *Place and process "Small test sample 7x7 mm"* ... 40
Fig. 37: Specifying the position ... 40
Fig. 38: Dialog *Test tool settings* ... 41
Fig. 39: Dialog *Check test tool finished* ... 41
Fig. 40: *Tune tool settings* ... 42
Fig. 41: *View Tool Settings* ... 42
Fig. 42: Tool selected ... 43
Fig. 43: Modifying the tool settings ... 43
Fig. 44: Copy of the tool with the modified settings ... 44
Fig. 45: Saving the tool ... 44
Fig. 46: Tool saved ... 45
Fig. 47: Resuming the processing procedure ... 45
Fig. 48: Phase *Bottom Side* selected ... 46
Fig. 49: Work package *Structure Bottom* selected ... 46
Fig. 50: *Test tool settings* disabled ... 47
Fig. 51: Assembly of the press mold and the materials ... 49

Fig. 52: Example of a laminated PCB material ... 51
Fig. 53: Example of a non-laminated PCB material ... 51
Fig. 54: PCB layout ... 52
Fig. 55: PCB layout | isolation channel ... 52
Fig. 56: PCB layout | creating strips ... 53
Fig. 57: PCB layout | delaminating strips ... 53
Fig. 58: RF PCB layout | RF geometry ... 54
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 ... 56
Fig. 63: A 6-layer PCB structure ... 57
Fig. 64: A 6-layer PCB structure with blind vias and buried vias ... 57
Fig. 65: Annular ring ... 58
Fig. 66: A through hole with external and internal annular rings ... 59
Fig. 67: Incorrect inner layer fill in the user guidance step *Layout* ... 59
Fig. 68: Correct inner layer fill in the user guidance step *Layout* ... 60
Fig. 69: Staggered vias and minimum distance ... 60
Fig. 70: Stacked vias ... 61
Fig. 71: Desktop icon CircuitPro PL ... 64
Fig. 72: Message *System connection* ... 64
Fig. 73: Dialog *Signal light check* ... 64
Fig. 74: User guidance step *New* ... 64
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* ... 66
Fig. 79: Dialog *Open* ... 67
Fig. 80: User guidance step *Import* ... 68
Fig. 81: Assigning holes to layer ... 68
Fig. 82: User guidance step *Layout* ... 69
Fig. 83: Context menu *Step and repeat* ... 70
Fig. 84: Dialog *Step and repeat* ... 71
Fig. 85: Dialog *Step and repeat* after entering values ... 71
Fig. 86: Multiplied layout ... 72
Fig. 87: Create fiducials automatically ... 74
Fig. 88: Dialog *Automatic fiducial creation* ... 74
Fig. 89: Fiducials created ... 75
Fig. 90: *Compute scan fields* ... 76
Fig. 91: Dialog *Compute scan fields* ... 76
Fig. 92: Scan fields computed ... 77
Fig. 93: *Computing toolpaths* ... 77
Fig. 94: Message *Toolpaths calculation* ... 77
Fig. 95: Message *Computation results* ... 78
Fig. 96: User guidance step *Processing* ... 78
Fig. 97: Dialog *Processing phase: Mount material* ... 79
Fig. 98: Dialog *Placement* ... 79
Fig. 99: Dialog *Placement* | *Processing data* ... 80
Fig. 100: Message *Thickness measurement running* ... 80
Fig. 101: Message *Laser warm-up phase in progress* ... 80
Fig. 102: Dialog *Test tool settings* ... 81
Fig. 103: Message *Processing phase: Flip material* ... 81
Fig. 104: Message *Board production finished* ... 82
Fig. 105: Desktop icon CircuitPro PM ... 83
Fig. 106: Dialog *Connection steps* ... 84
Fig. 107: Dialog *New document* ... 84
Fig. 108: Dialog *Open* ... 85
Fig. 109: Dialog *Import* | Assigned layers ... 85

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
Fig. 113: Context menu *Step & Repeat*... 88
Fig. 114: Dialog *Step & Repeat* 88
Fig. 115: Dialog *Step & Repeat* after entering values 89
Fig. 116: *CAM view* of the multiplied layout 89
Fig. 117: Dialog *Create fiducial* 90
Fig. 118: *CAM view* after creating fiducials 90
Fig. 119: Tab *Isolate*.. 91
Fig. 120: Tab *Contour routing*.. 91
Fig. 121: Tab *Pockets*... 92
Fig. 122: Message *Computation Results* 92
Fig. 123: Open project 94
Fig. 124: Dialog *Upgrade project*.. 95
Fig. 125: Upgrade selected 95
Fig. 126: Material selection.. 96
Fig. 127: Opened project in the user guidance step *Toolpaths*................................ 96
Fig. 128: Dialog *Computing toolpaths* 97
Fig. 129: Message *Processing phase: Process drilling on a ProtoMat* 97
Fig. 130: Message *Processing phase: Galvanic through-hole plating* 97
Fig. 131: Message *Processing phase: Mount material bottom* 97
Fig. 132: Dialog *Copper thickness* 98
Fig. 133: Dialog *Placement* 98
Fig. 134: Message *Processing phase: Flip material* 99
Fig. 135: Dialog *Open*.. 100
Fig. 136: *CAM view* of the opened project 100
Fig. 137: *Machining view* of the opened project 101
Fig. 138: Drop-down list of the processing phases 101
Fig. 139: *Machining view* of drilling and contour-routing data 102
Fig. 140: Upgrade selected 106
Fig. 141: List of templates 111
Fig. 142: Material selection.. 111
Fig. 143: User guidance step *Import* 112
Fig. 144: User guidance step *Layout*.. 112
Fig. 145: Dialog *Compute scan fields*... 113
Fig. 146: Scan fields computed | *Overlap* displayed 114
Fig. 147: *Computing toolpaths*.. 114
Fig. 148: Flexible material fastened to the processing table 116
Fig. 149: Dialog *Placement* 116
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
Fig. 153: Pane *Workflow setup*... 119
Fig. 154: Context menu *Process from here*.. 119
Fig. 155: Dialog *Copper thickness* 120
Fig. 156: Dialog *Test tool settings* 120
Fig. 157: List of multi-layer templates.. 125
Fig. 158: Material selection.. 126
Fig. 159: User guidance step *Import* 126
Fig. 160: Layout of a multi-layer PCB.. 127
Fig. 161: Context menu *Move object*... 127
Fig. 162: *Specify reference point*.. 128
Fig. 163: *Specify distance* 128
Fig. 164: Desired position of the copied layout 129
Fig. 165: Layout moved to the desired position.. 129
Fig. 166: Opened project 130
Fig. 167: Dialog *Computing toolpaths* 130

Fig. 168: User guidance step *Processing* ... 131
Fig. 169: Message *Processing phase: Mount laminate* ... 131
Fig. 170: Dialog *Placement* ... 131
Fig. 171: Message *Processing phase: Mount core* ... 133
Fig. 172: Message *Processing phase: Press all layers* ... 134
Fig. 173: Pane *Workflow setup* ... 135
Fig. 174: Context menu *Process from here* ... 136
Fig. 175: Message *Processing phase: Mount material top* ... 136
Fig. 176: Dialog *Placement* ... 137
Fig. 177: Message *Processing phase: Galvanic through-hole plating* ... 137
Fig. 178: Dialog *Copper thickness* ... 138
Fig. 179: Dialog *Import* | Assigned layers ... 141
Fig. 180: *CAM view* multi-layer PCB ... 141
Fig. 181: Upgrade selected ... 144
Fig. 182: Material selection ... 145
Fig. 183: Opened project ... 145
Fig. 184: Dialog *Computing toolpaths* ... 145
Fig. 185: User guidance step *Processing* ... 146
Fig. 186: Message *Processing phase: Custom instruction* ... 146
Fig. 187: Message *Processing phase: Mount core* ... 146
Fig. 188: Dialog *Placement* ... 147
Fig. 189: Message *Processing phase: Press all layers* ... 148
Fig. 190: Context menu *Process from here* ... 150
Fig. 191: Dialog *Copper thickness* ... 150
Fig. 192: Message *Processing phase: Process drilling on a ProtoMat* ... 152
Fig. 193: Dialog *Import* | Assigned layers ... 154
Fig. 194: *CAM view* multi-layer PCB ... 154
Fig. 195: Upgrade selected ... 157
Fig. 196: Material selection ... 158
Fig. 197: User guidance step *Processing* ... 158
Fig. 198: Message *Processing phase: Custom instruction* ... 159
Fig. 199: Message *Processing phase: Mount core* ... 159
Fig. 200: Dialog *Placement* ... 159
Fig. 201: Message *Processing phase: Press all layers* ... 160
Fig. 202: Context menu *Process from here* ... 161
Fig. 203: Dialog *Import* | Assigned layers ... 166
Fig. 204: *CAM view* multi-layer PCB with blind vias and buried vias ... 167
Fig. 205: Upgrade selected ... 169
Fig. 206: Material selection ... 170
Fig. 207: Opened project ... 170
Fig. 208: Dialog *Computing toolpaths* ... 170
Fig. 209: User guidance step *Processing* ... 171
Fig. 210: Message *Processing phase: Custom instruction* ... 171
Fig. 211: Dialog *Copper thickness* ... 171
Fig. 212: Dialog *Placement* ... 172
Fig. 213: Message *Processing phase: Press all layers* ... 173
Fig. 214: Assembly of the materials in the press mold ... 174
Fig. 215: Positioning markers core material ... 174
Fig. 216: Copper foil on the lower steel press sheet ... 175
Fig. 217: Prepreg material on the copper foil ... 175
Fig. 218: Core material on the dowel pins ... 176
Fig. 219: Prepreg material on the core material ... 176
Fig. 220: Copper foil on the prepreg material ... 177
Fig. 221: Pane *Workflow setup* ... 179
Fig. 222: Context menu *Process from here* ... 180
Fig. 223: Dialog *Test tool settings* ... 181
Fig. 224: Blind via of good quality under the microscope ... 182
Fig. 225: Context menu *Process from here* ... 184

Fig. 226: Dialog *Copper thickness* ... 184
Fig. 227: Message *Processing phase: Process drilling on a ProtoMat* ... 186
Fig. 228: List of single-sided templates ... 188
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
Fig. 233: User guidance step *Material* ... 194
Fig. 234: Copy of selected material ... 194
Fig. 235: View *Modify parameters* ... 195
Fig. 236: Entering the material name ... 195
Fig. 237: Deleting the materials ... 195
Fig. 238: Materials deleted ... 196
Fig. 239: Adding the material ... 196
Fig. 240: Material added to the new set of materials ... 196
Fig. 241: Selecting the material ... 197
Fig. 242: User guidance step *Import* ... 198
Fig. 243: Assigning board outline to layer *BoardOutline* ... 198
Fig. 244: Deselecting the source *BoardOutline* ... 198
Fig. 245: Assigning the holes to layer *DrillPlated* ... 198
Fig. 246: Message *Info* ... 198
Fig. 247: User guidance step *Layout* ... 199
Fig. 248: Highlighted layout ... 199
Fig. 249: Layout moved ... 200
Fig. 250: Context menu *PCB layer > Prepreg 1* ... 201
Fig. 251: Context menu *Display mode > True Width* ... 201
Fig. 252: Context menu *Technology > Cutting* ... 202
Fig. 253: Menu item *Add new layer* ... 203
Fig. 254: Context menu item *Rename* ... 203
Fig. 255: Renaming layer ... 204
Fig. 256: Input fields *Specify center point* ... 204
Fig. 257: Input fields *Specify diameter* ... 205
Fig. 258: Specifying the next center point ... 205
Fig. 259: Flexible material positioning holes ... 206
Fig. 260: Context menu *Copy to layer* ... 207
Fig. 261: Input fields *Non-uniform scaling (in percent)* ... 208
Fig. 262: Values for *Non-uniform scaling (in percent)* ... 208
Fig. 263: Preview of the scaled positioning holes ... 209
Fig. 264: Prepreg positioning holes ... 209
Fig. 265: Rename ... 210
Fig. 266: Work package selected ... 210
Fig. 267: Modified settings ... 211
Fig. 268: Work package selected ... 211
Fig. 269: Settings of the work package *Cut Reference Holes* ... 212
Fig. 270: *Channel width* modified ... 212
Fig. 271: Work package deselected ... 213
Fig. 272: Work package selected ... 213
Fig. 273: Modified settings ... 214
Fig. 274: Phase and work package selected ... 214
Fig. 275: Modified settings ... 215
Fig. 276: User guidance step *Processing* ... 216
Fig. 277: Context menu *Process from here* ... 219
Fig. 278: Dialog *Test tool settings* ... 220
Fig. 279: Dialog *Copper thickness* ... 221
Fig. 280: Dialog *Test tool settings* ... 222
Fig. 281: User guidance step *Layout* ... 225
Fig. 282: Material selection ... 226
Fig. 283: *Specify target point* ... 227

Fig. 284: Pasted layout ... 227
Fig. 285: Material selection ... 230
Fig. 286: User guidance step *Import* ... 231
Fig. 287: Layout of an RF PCB ... 231
Fig. 288: User guidance step *Workflow* ... 232
Fig. 289: Work package *Structure Top* selected ... 232
Fig. 290: Group *Parameters* ... 233
Fig. 291: Sub-group *Hatching* ... 233
Fig. 292: Tool selected ... 234
Fig. 293: Tool moved ... 234
Fig. 294: User guidance step *Processing* ... 235
Fig. 295: Select material ... 238
Fig. 296: Select *CutInside* ... 239
Fig. 297: Imported data ... 239
Fig. 298: Layout selected ... 240
Fig. 299: Combine open paths ... 240
Fig. 300: User guidance step *Workflow* ... 241
Fig. 301: Selecting the work package *Cut Inside* ... 241
Fig. 302: Group *Parameters* expanded ... 242
Fig. 303: Entry *Center* selected ... 242
Fig. 304: Selecting the material ... 245
Fig. 305: User guidance step *Import* ... 246
Fig. 306: Deselected files ... 247
Fig. 307: Format undefined ... 248
Fig. 308: Selecting the appropriate file format ... 249
Fig. 309: Appropriate file format selected ... 249
Fig. 310: Unassigned layer ... 250
Fig. 311: Selecting the target layer ... 250
Fig. 312: Selected target layer ... 251
Fig. 313: User guidance step *Layout* ... 251
Fig. 314: Unassigned layer ... 252
Fig. 315: Info about new layer created ... 252
Fig. 316: User guidance step *Layout* | processing data ... 253
Fig. 317: List of layers | *Select by layer* ... 253
Fig. 318: Assigning objects to target layer ... 254
Fig. 319: Assigned holes ... 255
Fig. 320: Large size of the layout ... 257
Fig. 321: Selecting the appropriate unit ... 258
Fig. 322: Changed dimensions of the layout ... 258
Fig. 323: Incorrect preview of the layout ... 259
Fig. 324: Correct preview of the layout ... 259
Fig. 325: File *Tutor.BOA* selected ... 260
Fig. 326: Tab *Apertures/Tools* and sub-tab *Attributes* ... 260
Fig. 327: Selecting the type of aperture ... 261
Fig. 328: Preview of the modified aperture ... 261
Fig. 329: Layer name in the file *Tutor.BOA* ... 262
Fig. 330: Layer names not used in the Gerber file ... 262
Fig. 331: Gerber file selected ... 263
Fig. 332: Activated check boxes ... 263
Fig. 333: Layer names defined in Gerber files displayed ... 264
Fig. 334: Dialog *Options* | *Gerber* ... 264
Fig. 335: Selecting value *True* for *Use layer name* ... 265
Fig. 336: Layer names defined in Gerber files displayed ... 265

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