ProtoLaser U4/S4

How-to guides

Order code : 10073032
Version: 1.0

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 2017-05-17

Copyright © 2017 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	www.lpkf.com

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: http://www.lpkf.com/support.

Contents

Introduction **8**

1 Basics **10**

1.1 Project placement 10
1.2 Fiducial recognition 16
1.3 Multi-layer process 19
1.4 Design guidelines for the production of a multi-layer PCB with buried vias and blind vias 22

1.4.1 Structure of a multi-layer PCB with buried vias and blind vias 22
1.4.2 Required files/layers 22
1.4.3 Through holes and via sizes before plating 23
1.4.4 Annular rings 23
1.4.5 Staggered vias 24
1.4.6 Stacked vias 25
1.4.7 Inner layers fill 25

2 Producing double-sided PCBs **28**

2.1 How to produce a double-sided PCB without through-hole plating 28
2.2 How to produce a double-sided PCB with galvanic through-hole plating 40
2.3 How to produce a double-sided PCB with non-galvanic through-hole plating 58

3 Producing multi-layer PCBs **64**

3.1 How to produce a multi-layer PCB with galvanic through-hole plating by using a UV laser system only 64
3.2 How to produce a multi-layer PCB with galvanic through-hole plating 76
3.3 How to produce a multi-layer PCB with non-galvanic through-hole plating 89
3.4 How to produce a multi-layer PCB with blind vias and buried vias 100

4 Other how-to examples **120**

4.1 Producing a polyimide stencil 120
4.2 Producing a flex-rigid PCB 123
4.3 Structuring the solder mask with the laser system 155

5 Processing files **162**

5.1 Processing DXF files in CircuitPro PL 3.0 162
5.2 Processing Gerber and Excellon files in CircuitPro PL 3.0 170

6 Appendix **192**

6.1 List of figures 192
6.2 List of tables 195

Introduction

This document describes the most common applications for the production of double-sided and multi-layer PCBs with the LPKF ProtoLaser U4/S4. 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 thoroughly 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.

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.

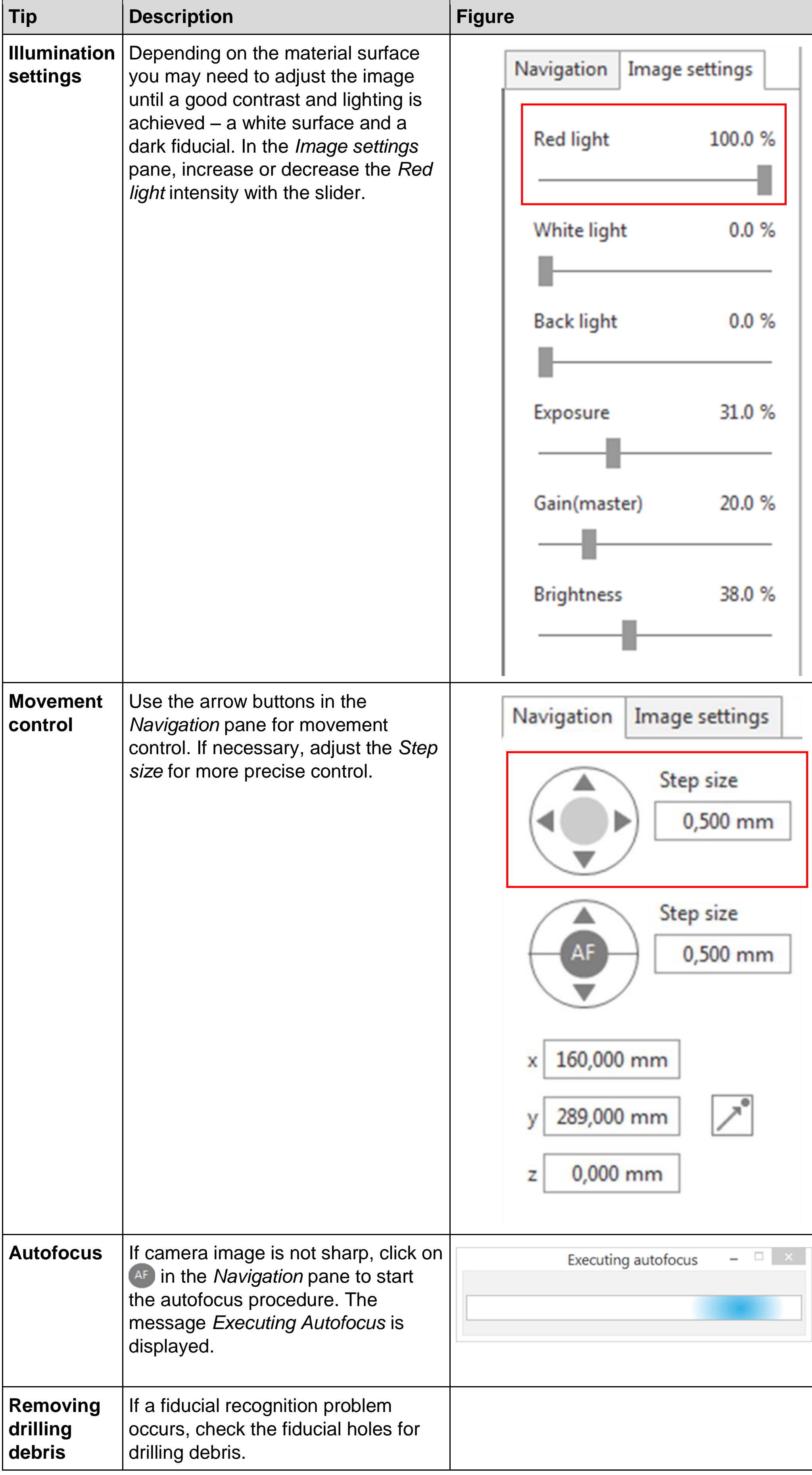
Moreover, this chapter also describes the multi-layer process, as well as specific design guidelines for the production of multi-layer PCBs with blind vias and buried vias. Observe these information, since they provide a base for a successful production of multi-layer PCBs.

1.1 Project placement

This chapter contains helpful information on placing your project easily and quickly in the CircuitPro PL software. The project can be placed correctly either by matching the fiducial positions or by defining the processing area. Both procedures are described below. **Use one** of them.

The following table provides tips for positioning and fiducial reading that can be **used anytime** during procedures:

Tip	Description	Figure
Pilot laser	Switch on the pilot laser by clicking on ⚹ for help with positioning. Follow the movement of the processing table in the system itself by observing the pilot laser position. Match the position of the pilot laser and the fiducial on the material. Switch off the pilot laser off by clicking on ⚹ when position is set.	
Zooming in	Zoom in the processing area by using the scroll wheel of the mouse for more precise positioning.	



Tip	Description	Figure
Illumination settings	Depending on the material surface you may need to adjust the image until a good contrast and lighting is achieved – a white surface and a dark fiducial. In the *Image settings* pane, increase or decrease the *Red light* intensity with the slider.	
Movement control	Use the arrow buttons in the *Navigation* pane for movement control. If necessary, adjust the *Step size* for more precise control.	
Autofocus	If camera image is not sharp, click on AF in the *Navigation* pane to start the autofocus procedure. The message *Executing Autofocus* is displayed.	
Removing drilling debris	If a fiducial recognition problem occurs, check the fiducial holes for drilling debris.	

Tip	Description	Figure
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 reading

Matching the fiducial position

The goal of this procedure is:

- to position the location of the laser head in the system directly above one of the fiducials on the material in order
- to match the location of the red crosshair in CircuitPro PL 3.0 to the fiducial in the layout.

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

In the *Processing* view of the CircuitPro PL 3.0 software:

- a red crosshair ( ) displays the position of the laser head and
- a black and white circle ( ) displays a fiducial.

1. Move the *Placement* dialog off to the left side to get a better overview.
2. Double-click on the processing area.

☐ The processing table starts moving and the red crosshair is displayed at the point of your double click.

3. Click on .

☐ The pilot laser is switched on.

The position of the pilot laser beam on the material represents the position of the laser head.

4. Use the arrow buttons in the *Navigation* pane to move the processing table. If necessary, adjust the *Step size* for more precise movement control.

Use the tips for positioning described at the beginning of this chapter.

5. Observe the movement of the processing table **in the system**.

   Match the position of the pilot laser beam on the material and on **one of the fiducials** on the board.
6. Click on .

- The pilot laser is switched off.
- One of the fiducials is visible in the pane *Camera* (located in the left bottom corner of the software user interface):

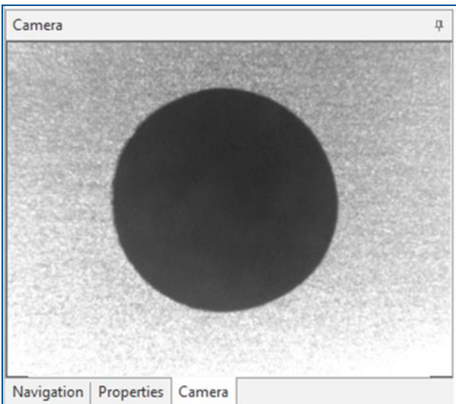


Fig. 1: Fiducial in the *Camera* view

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

By performing the above steps, you have matched the position of the laser head and one of the fiducials on the material. The **same** needs to be **reflected** in the *Processing* view of the **CircuitPro** PL 3.0 **software**.

- The position of the layout in the *Processing* view is displayed as follows (this is an example used for this procedure):

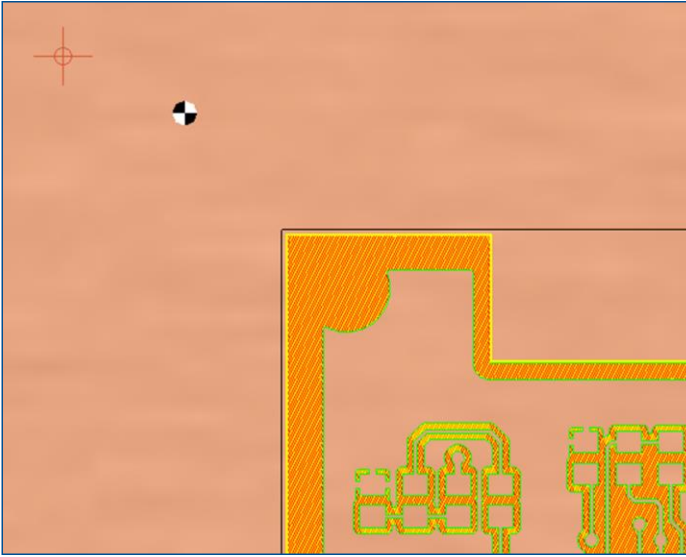


Fig. 2: Inaccurate position of the layout

- The **position** of the **red crosshair** now represents not only the position of the laser head, but also the **actual position of the fiducial on the material**. The remaining step is to move the entire layout so that the fiducial ( ) and the red crosshair overlap.

7. Move the mouse cursor over the layout.

- The mouse cursor turns into the hand symbol - .

8. Drag & drop the layout to match fiducial and red crosshair location.

- ☐ When the red crosshair (laser head) and fiducial position **match**, the *Processing* view is displayed as follows:

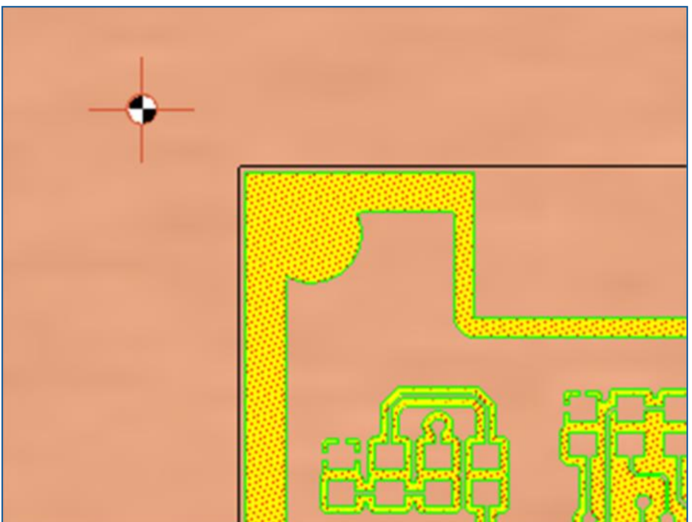


Fig. 3: Matching laser head and fiducial position in the *Processing* view

- ☐ The location of the red crosshair (laser head) in CircuitPro PL 3.0 has been matched to the fiducial in the layout.

9. Click on [Continue].

- ☑ The fiducial position has been matched and the read fiducial phase is initiated.

Determining the processing area

As already mentioned, this procedure is an **alternative** to Matching the fiducial position described above. The aim of this procedure is to precisely match the board position and board size to the processing area used by CircuitPro PL software.

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

1. Move the *Placement* dialog off to the left side in order to get a better overview.
2. Double-click on the processing (brownish) area to start moving the processing table.
3. Use the arrow buttons in the *Navigation* pane for movement control.
4. If necessary, adjust the *Step size* for more precise control.
5. Switch to the *Camera* view.

- ☐ Move the processing table until the processing head is directly above the **left front** corner of the material:

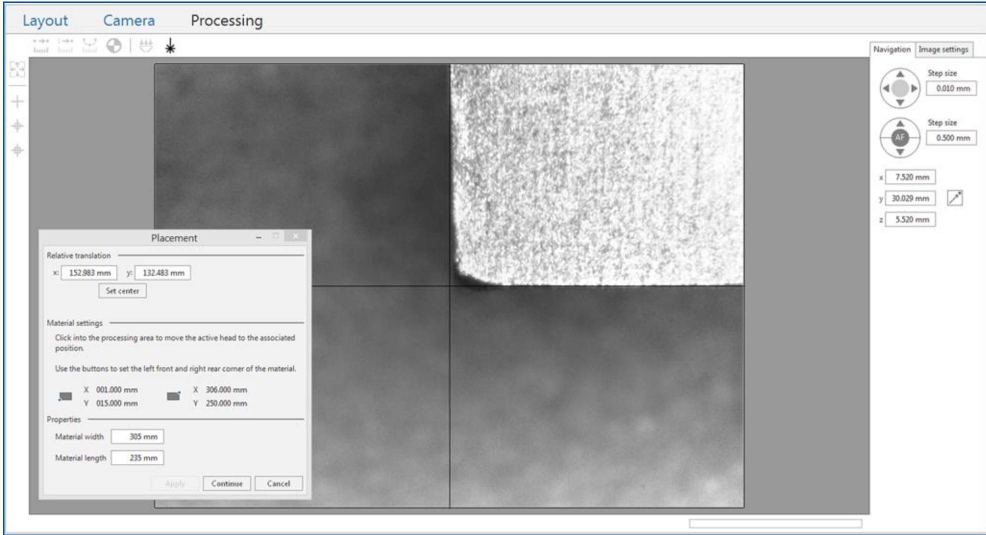


Fig. 4: Matching position of crosshair to board edge

6. Confirm the position by clicking on [icon] in the *Placement* dialog.
7. Switch to the *Processing* view and repeat the above procedure for the **right rear** corner.
8. Confirm the position by clicking on [icon] in the *Placement* dialog.
9. Click on [Set center] in the *Placement* dialog.

- ☐ The data are now precisely aligned and in the center of the board.

10. Click on [Continue].

- ☐ The position of crosshair to board edge has been matched and the reading fiducial phase is initiated.

☑ The processing area has been determined.

1.2 Fiducial recognition

This chapter contains helpful information on fiducial recognition in the CircuitPro PL software. This procedure follows the project placement procedures described in the previous chapter.

Recognizing fiducials

1. Switch to *Camera* view.

- The camera moves to the position of the first fiducial and determines its exact position. The recognized fiducial is selected by a green circle:

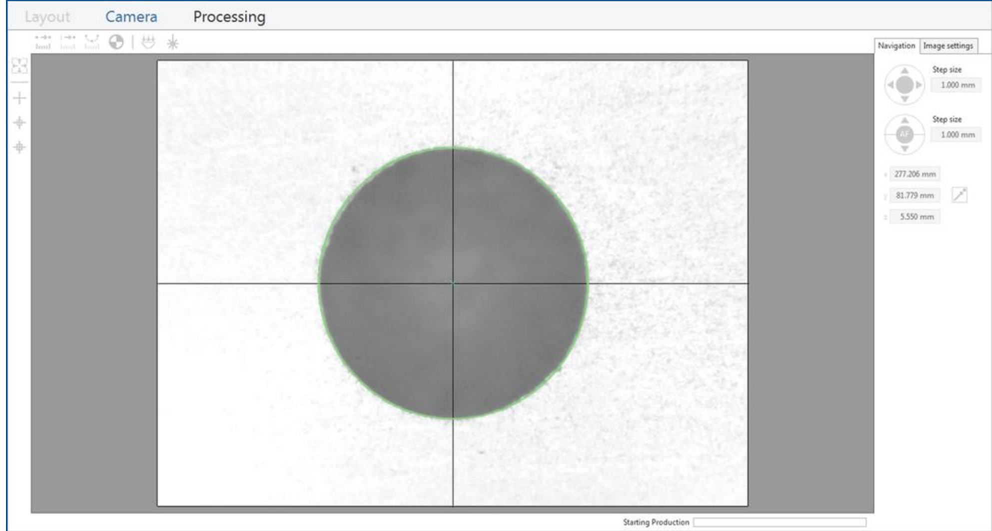


Fig. 5: ***Camera*** **view of a recognized fiducial**

- After the first fiducial has been recognized, the camera proceeds to read the other fiducials.

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

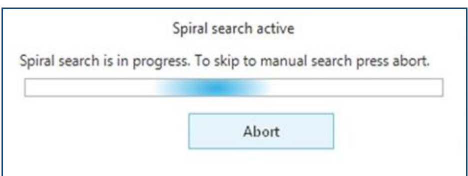


Fig. 6: **Message** ***Spiral search active***

To determine the size of the spiral search area, click on *Processing* > *Positioning settings…* You can enter the search area size in pixels in the corresponding displayed dialog.

☐ If the fiducial is not identified inside the spiral search area, a blue circle appears in the *Camera* view and the following message is displayed:

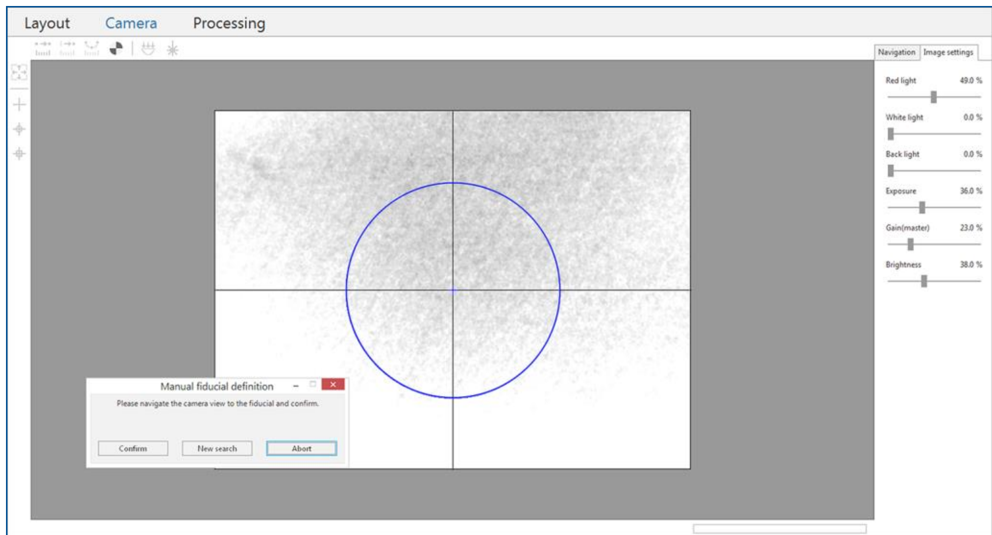


Fig. 7: ***Manual fiducial definition***

2. To move the table position manually, use the arrow buttons in the *Navigation* pane until the fiducial is visible in the *Camera* view.
3. Adjust the *Step size* for more precise movement, if necessary.

☐ The fiducial approximately matches the blue circle and is visible in the *Camera* view:

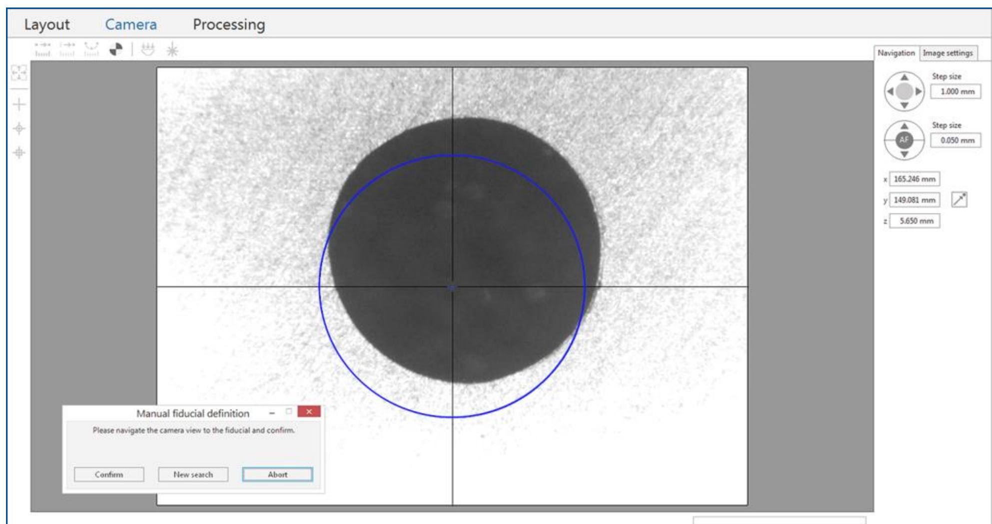


Fig. 8: **Visible fiducial**

4. Click on [New search].

☐ The camera recognizes the fiducial (selected by a green circle) and moves on to the next one.

☐ The *Read fiducial* phase is finished after all fiducials have been read successfully.

☑ The fiducials have been recognized.

Sometimes the fiducial quality is too poor for the camera to find.

Recognizing fiducials of poor quality

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

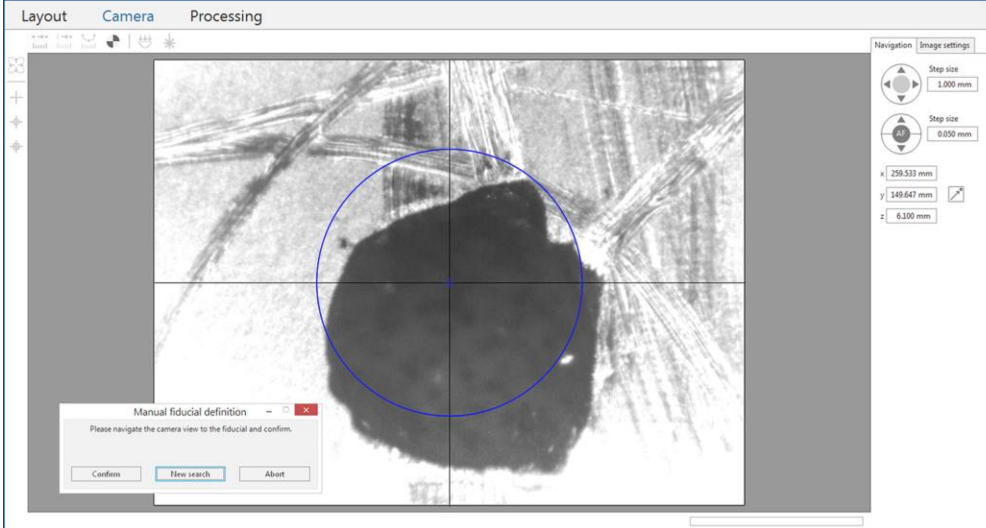


Fig. 9: ***Camera*** **view of a poor quality fiducial**

2. Manually move to the **exact** fiducial position (blue circle **exactly** matches the fiducial position).

- ☐ The fiducial in the Camera view is displayed as follows:

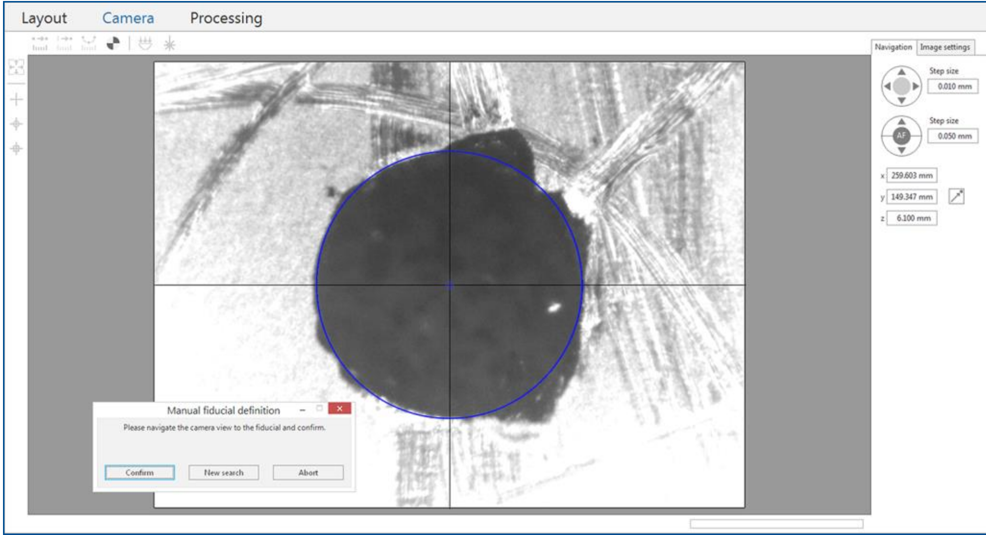


Fig. 10: ***Camera*** **view of the matched position of poor quality fiducial**

5. Click on [Confirm].

- ☐ The system saves the fiducial position.
- ☑ A fiducial of poor quality has been recognized.

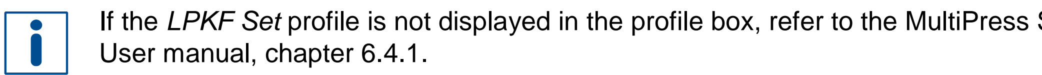
1.3 Multi-layer process

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

For detailed information on the multi-layer process refer to the MultiPress S User manual.

Starting the preheating stage

1. Switch on the MultiPress S.
2. Make sure that the profile *LPKF Set* is displayed in the profile box.



If the *LPKF Set* profile is not displayed in the profile box, refer to the MultiPress S User manual, chapter 6.4.1.

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 **120ºC 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 MultiPress S User manual, chapters 4.2 and 6.6.2.

Assembling the multi-layer stack

Observe the following information: Pins in the press mold align the individual layers. Rings between the bonded layers protect the pinholes 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:

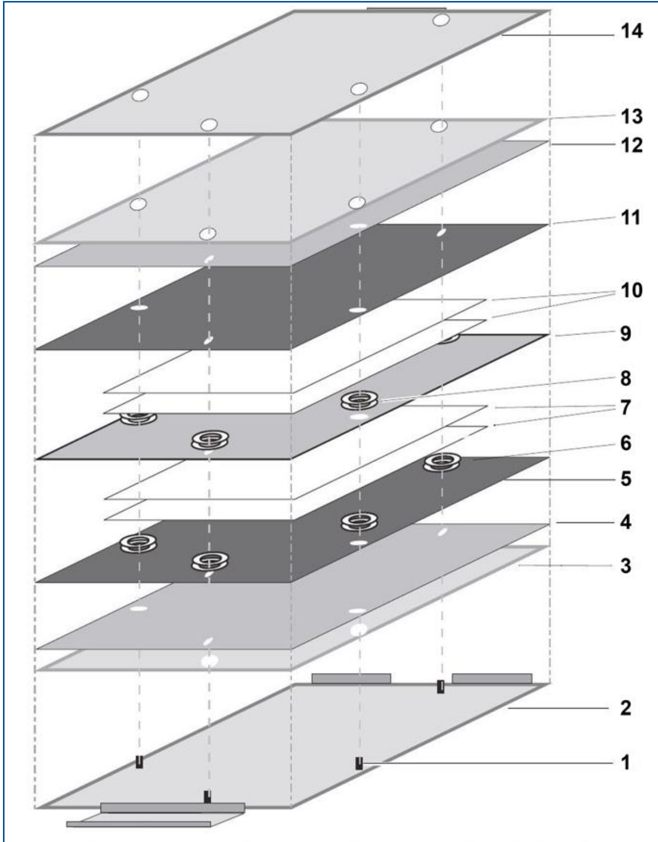


Fig. 11: Assembly of the press mold and the materials

1 Alignment pins
2 Aluminum press mold (lower part)
3 Textile 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 Textile press cushion
14 Aluminum press mold (upper part)

When using one sheet of prepreg, one ring should be used on each pin. When using two sheets of prepreg, two rings should be used on each 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 MultiPress S User manual, chapters 6.6.3 and 6.6.4.

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

1. Press the ENT button 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 ENT button 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.

☐ 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 be allowed to cure completely.

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

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

Use a small flat screwdriver to help removing the multi-layer PCB from the press mold. Use the screwdriver to lift the multi-layer PCB at the base of pins.

☑ The Multi-layer PCB has been pressed and cured.

1.4 Design guidelines for the production of a multi-layer PCB with buried vias and blind vias

This chapter describes basic design guidelines that should be observed when designing a layout for a multi-layer PCB with buried vias and blind vias for production with LPKF systems.

1.4.1 Structure of a multi-layer PCB with buried vias and blind vias

This section explains the structure of a multi-layer PCB with buried vias and blind vias.

The figure below shows the following:

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

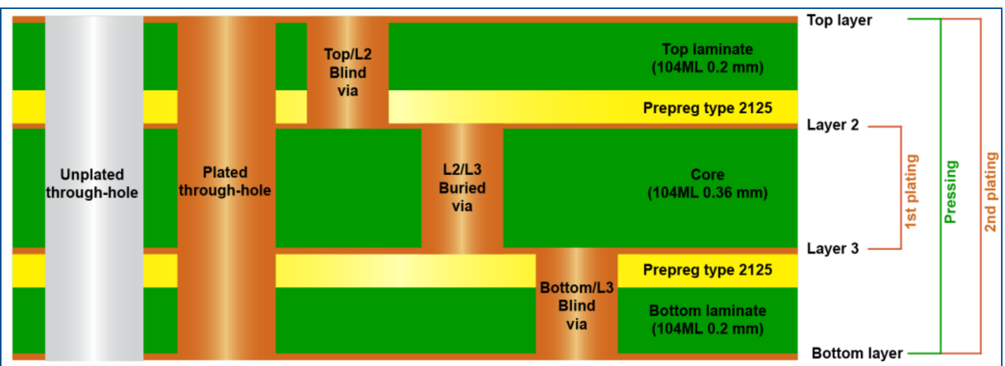


Fig. 12: A 4-layer PCB structure with all hole and via types

The drill layer names indicate which two layers a certain drill hole connects. For example: The drill layer *Top/L2 Blind via* connects the Top layer and layer 2.

1.4.2 Required files/layers

The following table indicates the files you require for the design of a 4-layer PCB with blind vias and buried vias:

Structuring	Drilling	Other
Top layer	DrillPlated (plated through hole)	BoardOutline
Layer 2	DrillUnplated (unplated through hole)	SolderMask Top
Layer 3	Blind vias (Top-L2)	SolderMask Bottom
Bottom layer	Buried vias (L2-L3)	
	Blind vias (Bottom-L3)	

Table 2: Required files/layers

1.4.3 Through holes and via sizes before plating

When designing the layout of a multi-layer PCB with blind vias and buried vias, it is important to observe the size range of the drill holes.

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

Type of drill hole	Minimum size	Maximum size
Through hole	200 µm	/
Buried via	200 µm	500 µm
Blind via	250 µm	1000 µm

Table 3: Size range according to drill hole type

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

1.4.4 Annular rings

An annular ring is a copper ring around a plated drill 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 example of an annular ring:

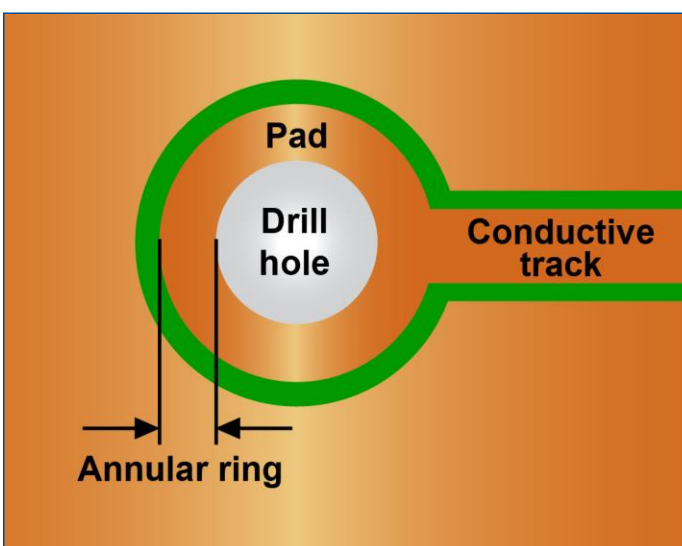


Fig. 13: Annular ring

A minimum 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 to 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 4 layers, as well as its internal and external annular rings:

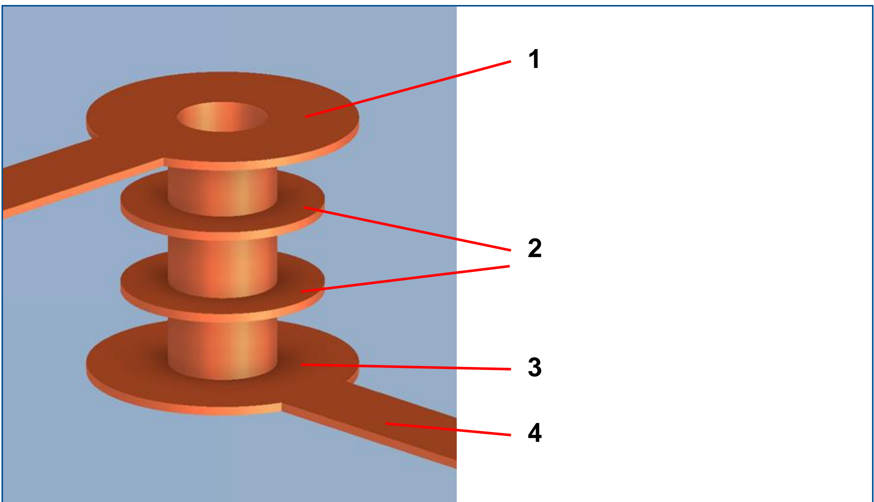


Fig. 14: A through hole with external and internal annular rings

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

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

1.4.5 Staggered vias

Staggered vias are vias on neighboring layers that are located close to each other, but do not overlap.

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

The following figure displays staggered vias and the minimum offset between them:

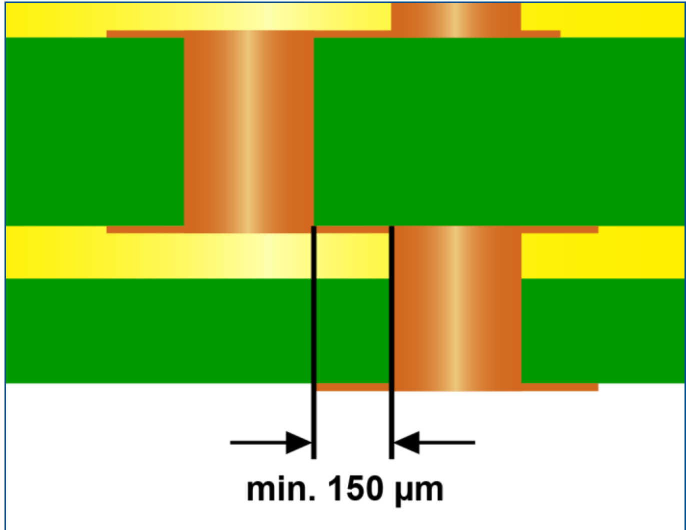


Fig. 15: Staggered vias and minimum offset

1.4.6 Stacked vias

Stacked vias are vias on neighboring layers that overlap. The PCB production process with LPKF systems and software does not support stacking of vias. **Use staggered vias** instead.

The following figure displays stacked vias:

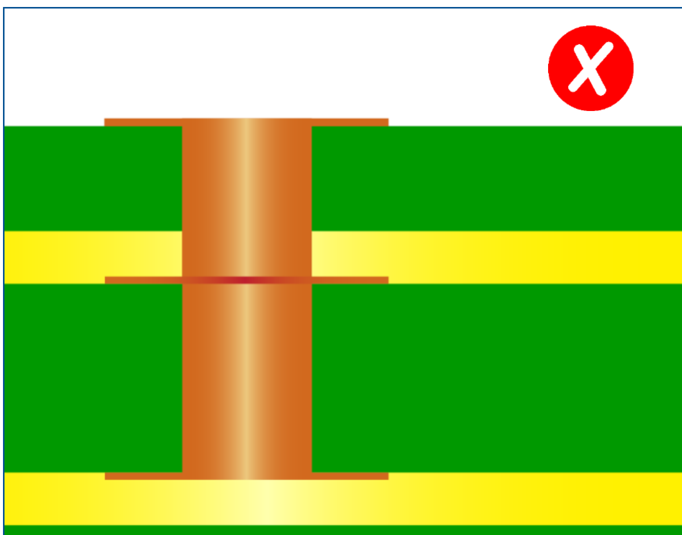


Fig. 16: Stacked vias

1.4.7 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:

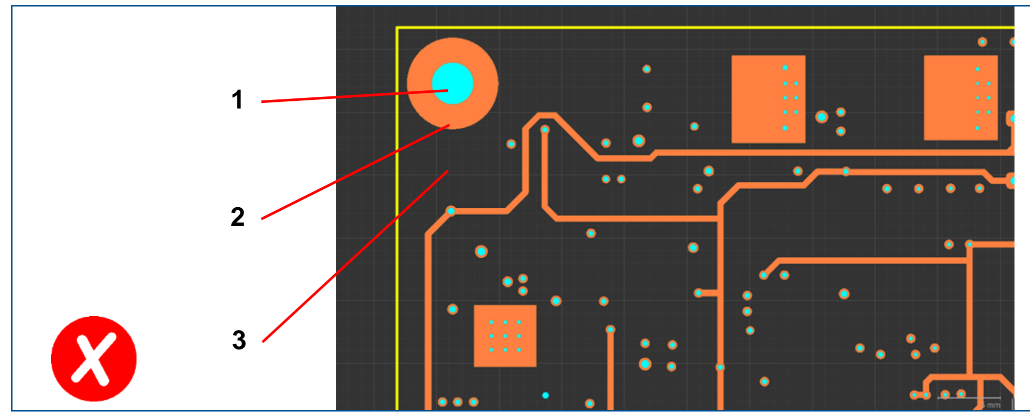


Fig. 17: Incorrect inner layer fill in the *Layout* view

1 Via

2 Annular ring

3 Rub-out (=FR4)

The following figure shows a correct inner layer fill:

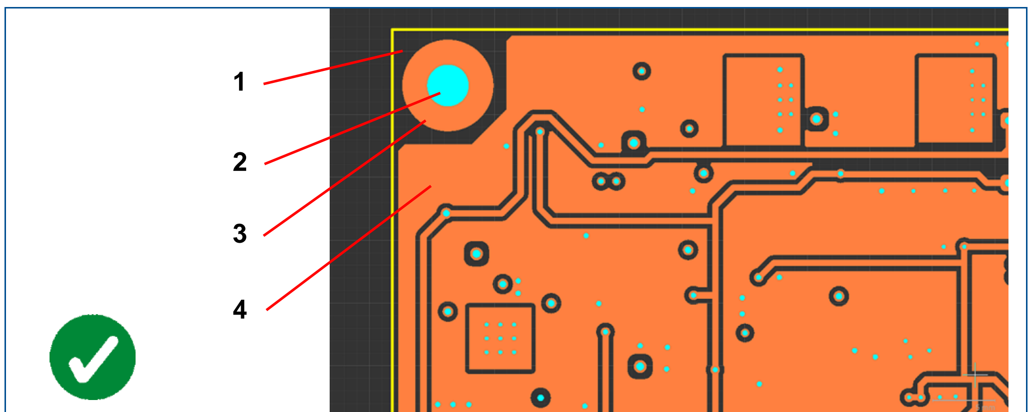


Fig. 18: Correct inner layer fill in the *Layout* view

1 Rub-out (=FR4)

2 Via

3 Annular ring (=copper)

4 Copper pour

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

Double-sided PCBs

2 Producing double-sided PCBs

This chapter describes the production of three different types of double-sided circuit boards:

1. Producing a double-sided circuit board without through-hole plating.
2. Producing a double-sided circuit board with galvanic through-hole plating.
3. Producing a double-sided circuit board with non-galvanic through-hole plating.

The following LPKF systems are required for the different processes:

Procedure	LPKF system
1	ProtoLaser
2	ProtoLaser, ProtoMat S or E, Contac S4
3	ProtoLaser, ProtoMat S or E, ProConduct

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 circuit board without through-hole plating by using a UV laser system only.

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

Consumables

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

Auxiliaries

- LPKF Cleaner (order no. 115891)
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4

The following steps are performed in this tutorial:

- Switching on the ProtoLaser
- Importing the data
- Multiplying the layout
- Inserting fiducials
- Computing toolpaths and scan fields automatically
- Processing the PCB
- Cleaning the ProtoLaser
- Cleaning the PCB

Switching on the ProtoLaser

1. Press the on/off switch at the system front.

☐ The system is switched on. The PC boots automatically.

2. Double-click on the desktop icon of CircuitPro PL 3.0.

Fig. 19: Desktop icon CircuitPro PL 3.0

☐ The following message is displayed:

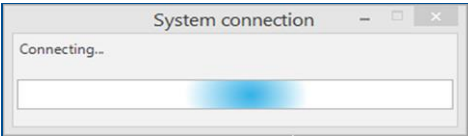


Fig. 20: Message *System connection*

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

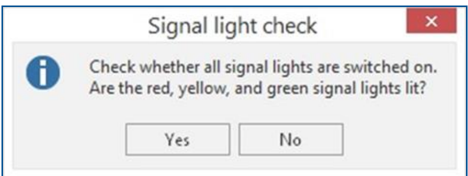


Fig. 21: Dialog *Signal light check*

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

☐ The following dialog is displayed:

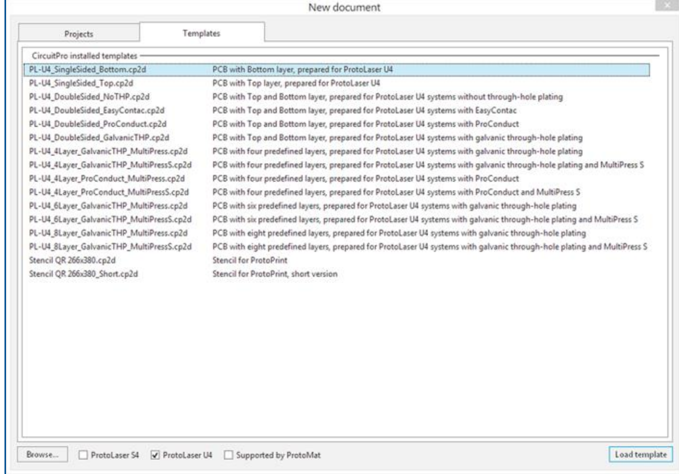


Fig. 22: Dialog *New document*

4. At the bottom of the *New Document* dialog, activate the check box of the laser system you are working with (in this example the ProtoLaser U4 is used):

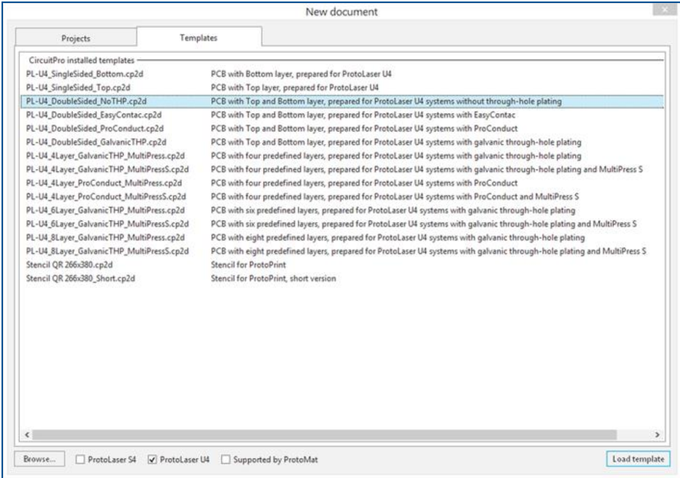


Fig. 23: Dialog *New Document*

5. In the list displayed, select the template *PL-U4_DoubleSided_NoTHP.cp2d*.
6. Click on [Load template].

☐ The following dialog is displayed:

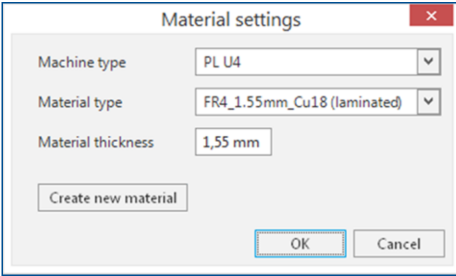


Fig. 24: Dialog *Material settings*

7. Select the *Machine type* and the *Material type* (for this example the material type *FR4_1.55mm_Cu18 (laminated)* is used).
8. Enter the *Material thickness*.
9. Click on [OK].

☑ 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:

1. Click on *Production > Connect with machine…*
2. Select your system in the drop-down list.
3. Click on [Connect].

The system requires a warm-up time of approx. **20** minutes for the laser source to attain a constant diode temperature. The warm-up phase starts automatically with the processing of the first job.
Alternatively, you can start the warm-up phase manually. In the *Processing* view, click on ✳. You can continue working in the *CAM* view during the warm-up phase.

Tips for selecting a template:

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

Selecting the material type

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

Example of the tool library name composition *FR4_1.55mm_Cu18 (laminated)*:

- *FR4* – material type
- *1.55mm* – material thickness
- *Cu18* – type and thickness of the conductor (Copper, 18 µm)
- *laminated* – indicates how the conductor is bonded to the material and determines how the laser toolpaths will be calculated.

Importing the data

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

- The following dialog is displayed:

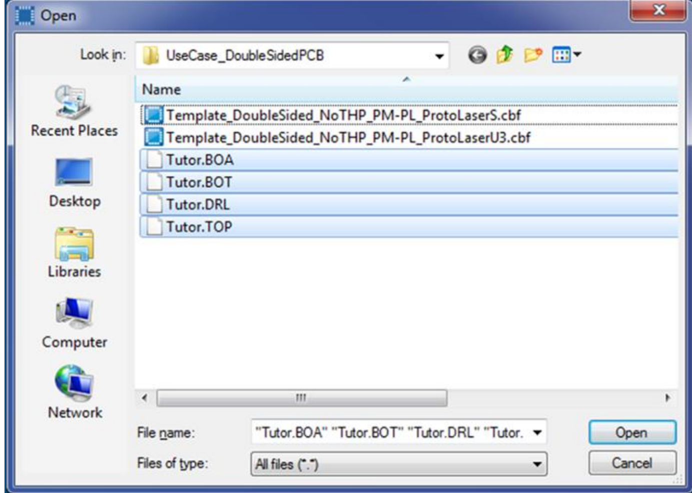


Fig. 25: Dialog *Open*

2. Navigate to the folder that contains the data you wish to import. For the example data that are used for this tutorial refer to the folder: `C:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PL 3.0\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. 26: Dialog *Import* and assigned layers

5. In the column *Layer/Template* select the layer *DrillPlated*.
6. Look at the preview of the layout and check whether the graphics and size (in the *Size/Format* column) of the drilling file is correct. If not, you can adjust the settings in the sub-tab *General*.
7. Click on [OK].

- ☐ The *Layout* view changes as follows:

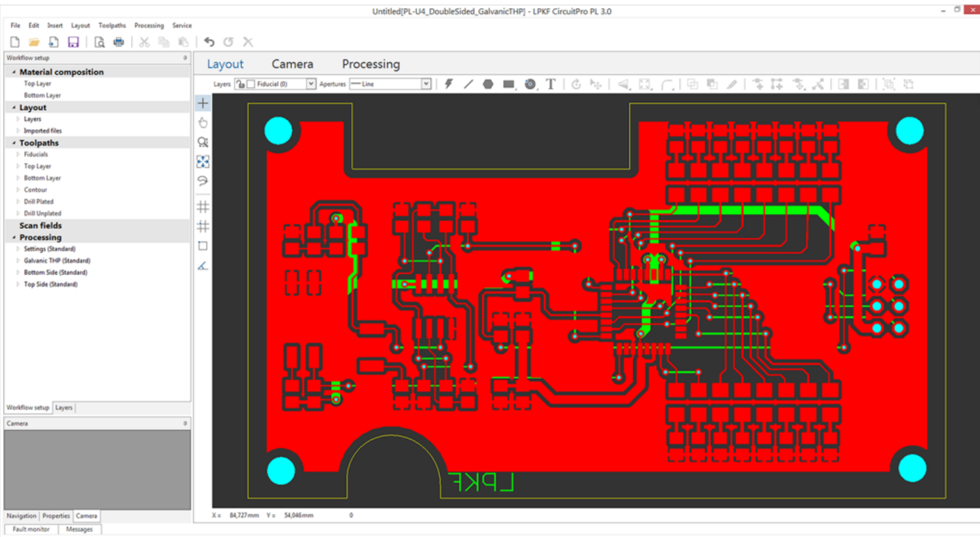


Fig. 27: Imported data in the *CAM* view

☑ 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 of the *Layer/Template* column, select which layer each artwork should be assigned to using the drop-down list.

Multiplying the layout (optional)

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

1. Select the entire layout by pressing `Ctrl` + `A`.

☐ The whole layout is selected.

2. Right-click on the selected layout.

☐ The following context menu is displayed:

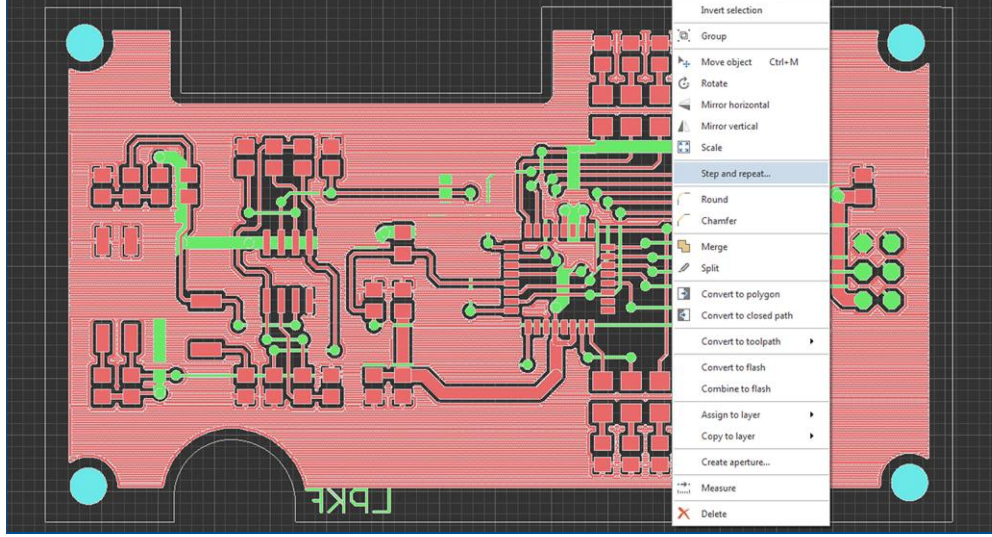


Fig. 28: Context menu *Step and repeat*

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

☐ The following dialog is displayed:

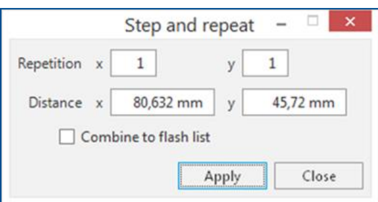


Fig. 29: Dialog *Step and repeat*

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

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

5. Increase the entered values in the *Distance* fields by 2 mm.

☐ After entering all the values, the dialog looks like this:

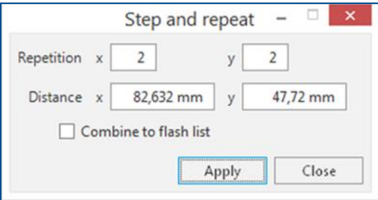


Fig. 30: Dialog *Step and repeat* after entering values

6. Click on [Apply] to multiply the layout.
7. Click on [Close] to close the window.
8. 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].
9. Press [Esc] or click anywhere on the black background to deselect the highlighted layout.

☐ The *Layout* view looks like this:

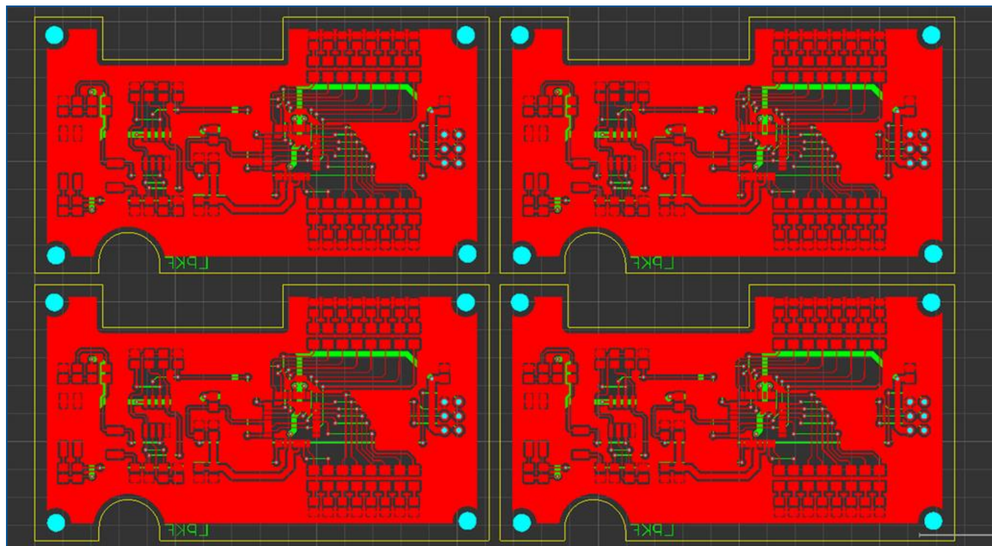


Fig. 31: ***Layout* view of the multiplied layout**

☑ The layout has been multiplied.

Inserting fiducials

1. Click on *Insert* > *Fiducial* > *Fiducial*.
2. Click on four positions in the layout to insert the fiducials there.

Best fiducial positions are just beyond the corners of the board.
A total of four fiducials are recommended for best alignment; using three fiducials is a good way to avoid wrong board orientation. You need at least two fiducials for correct operation of the process.

3. Press ESC.

☐ The function is closed and the *Layout* view changes as follows:

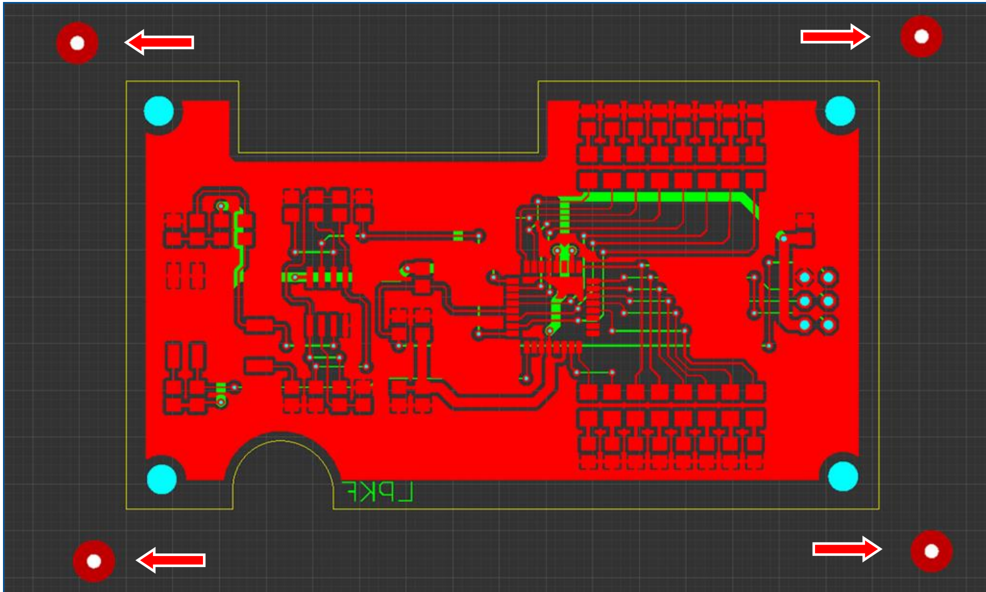


Fig. 32: Inserted fiducials

☑ The fiducials have been inserted.

Computing toolpaths and scan fields automatically

1. In the *Workflow setup* pane right-click on the section *Toolpaths*.

☐ The following context menu is displayed:

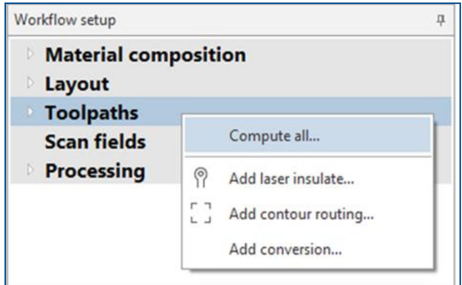


Fig. 33: Context menu *Compute all*

2. Click on the context menu item *Compute all…*

☐ The scan fields and toolpaths are computed automatically with default settings and the following message is displayed:

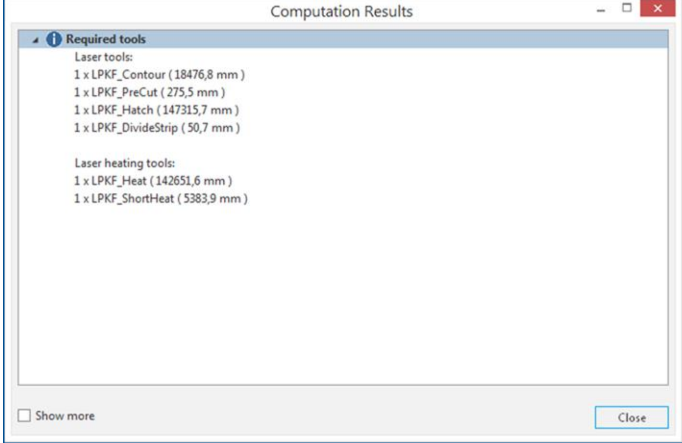


Fig. 34: Message *Computation results*

3. Check the calculation results for any possible warnings or errors and make corrections, if needed.
4. Click on [Close].

☑ The toolpaths and scan fields have been computed.

Processing the PCB

1. Measure the thickness of the base material with a caliper gauge.

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.

2. Click on *Processing* > *Process all* or in the *Processing* view on ▶.

☐ The following message is displayed:

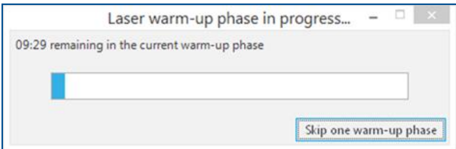


Fig. 35: Message *Laser warm up phase in progress…*

☐ After warm-up the following message is displayed:

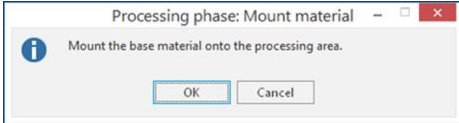


Fig. 36: Message *Processing phase: Mount material*

3. Open the cover.
4. Place the base material onto the processing table.
5. In the *Processing* view, click on .

☐ The base material is fastened to 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:

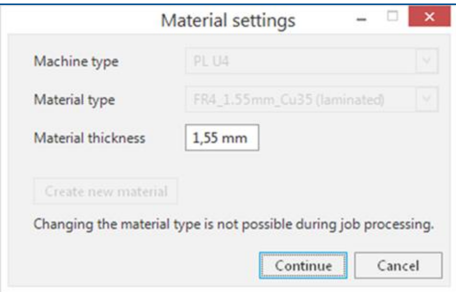


Fig. 37: Dialog *Material settings*

8. Enter the measured thickness in the field *Material thickness*.
9. Click on [Continue].

- The following dialog is displayed:

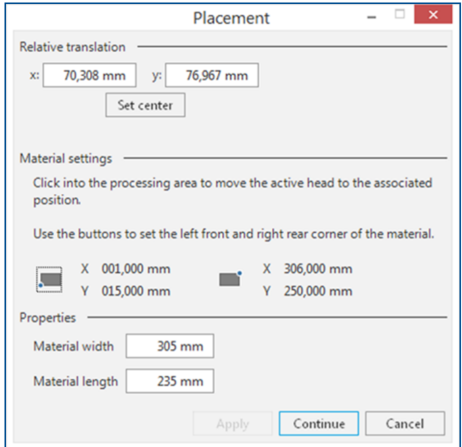


Fig. 38: Dialog *Placement*

10. Move the *Placement* dialog off to the left side.
11. Place the processing data.

    The production data can be moved using drag & drop or by entering the values in the fields *x* and *y*. Click on [Set center] to place the processing data at the center of the base material.

For detailed information on project placement refer to chapter 1.1, Project placement.

12. Click on [Continue].

- The structuring is started and the following message is displayed, when finished:

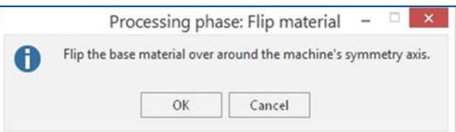


Fig. 39: Message *Processing phase: Flip material*

13. Turn the board over around the symmetry axis of the system.
14. Fasten the material along its edges with adhesive tape.
15. Click on [OK].

- If the fiducials are not found automatically, the following message is displayed:

Fig. 40: Message *Manual fiducial definition*

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

☐ After fiducial recognition is completed, the structuring of the Top side (*TopLayer*) is started. When finished, the following message is displayed:

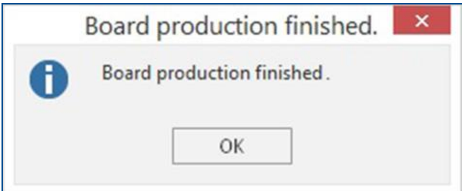


Fig. 41: Message *Board production finished*

☑ The PCB has been processed.

Cleaning the ProtoLaser

The processing area has to be cleaned, if heavily soiled.

- Use a vacuum cleaner to remove chips and residues from the processing area.

Cleaning the PCB

1. Check for any remaining copper strips on the PCB that should have been removed by laser.
2. Spray the board with LPKF Cleaner and use a brush to clean it.
3. Rinse the PCB with tap water and dry it with compressed air.
4. If the board 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 attach to the tape.

☑ The PCB has been cleaned.

Residual copper strips

If despite all cleaning any copper strips still remain on the PCB, check the material type and the tool library used.

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 circuit board using a UV 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

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

Auxiliaries

- 1 set of cutting tools (drills/contour routers for LPKF ProtoMat)
- LPKF Cleaner (order no. 115891)
- Oil-free compressed air
- Tap water

System

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

The following steps are performed in this tutorial:

- Switching on the ProtoMat
- Importing the data in CircuitPro PM 2.3
- Assigning holes to the layer *DrillUnplated* in CircuitPro PM 2.3
- Multiplying the layout in CircuitPro PM 2.3
- Inserting fiducials in CircuitPro PM 2.3
- Saving the file in CircuitPro PM 2.3
- Creating toolpaths in CircuitPro PM 2.3
- Processing a PCB (with ProtoMat)
- Galvanic through-hole plating of the PCB (with Contac S4)
- Preparing the data in CircuitPro PL 3.0
- Structuring the PCB (with ProtoLaser)
- Drilling unplated through-holes and cutting out the PCB (with ProtoMat)
- Cleaning the PCB
- Cleaning the ProtoMat

Switching on ProtoMat

1. Press the on/off switch.

☐ The system is switched on.

2. Turn on the PC that is connected to the system.

☐ Double-click on the desktop icon of CircuitPro PM 2.3.:

Fig. 42: Desktop icon CircuitPro PM 2.3

3. Wait for the system to connect and initialize.

☐ The following dialog is displayed:

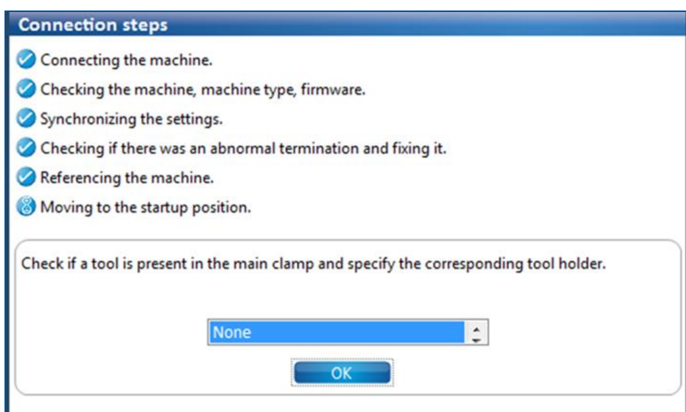


Fig. 43: Dialog *Connection steps*

☐ The following dialog is displayed:

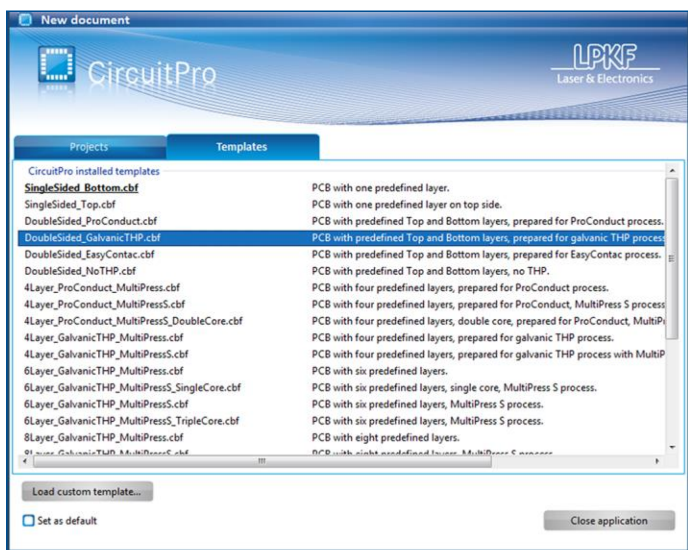


Fig. 44: Dialog *New document*

4. In the tab *Templates* select the template: *DoubleSided_GalvanicTHP.cbf.*

☑ The system has been switched on.

Importing the data in CircuitPro PM 2.3

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

☐ The following dialog is displayed:

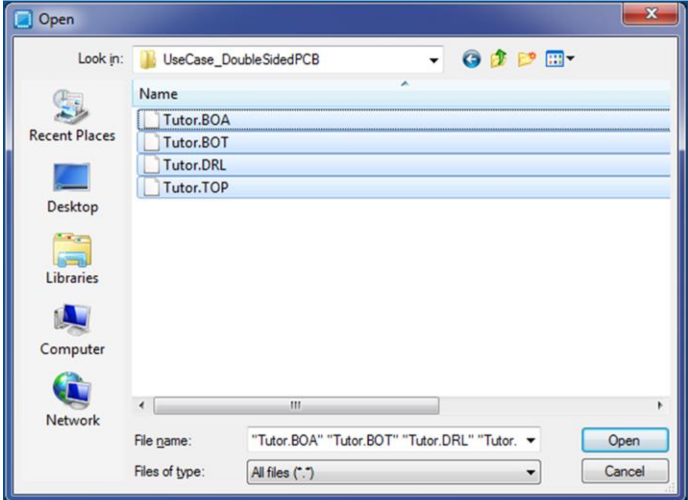


Fig. 45: Dialog *Open*

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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PM 2.3\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:

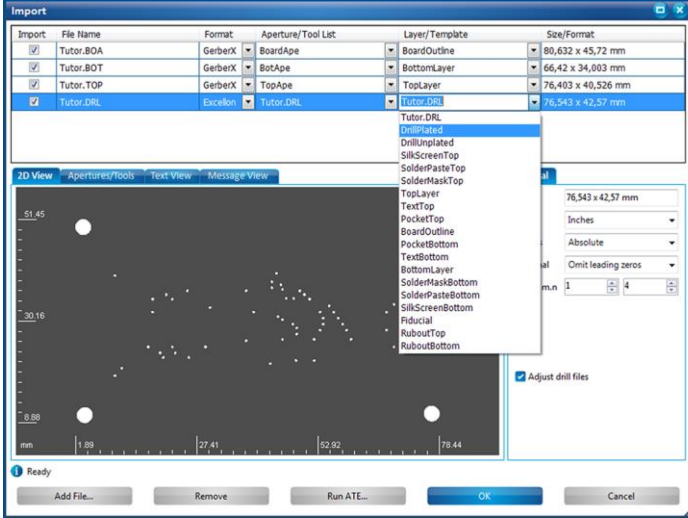


Fig. 46: Dialog *Import* and assigned layers

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

If there are no separate files for plated and unplated drills, they can be manually reassigned to the proper layer later (see following procedure on the assigning of holes).

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, the settings can be adjusted in the sub-tab *General*.
7. Click on [OK].

☐ The *CAM view* changes as follows:

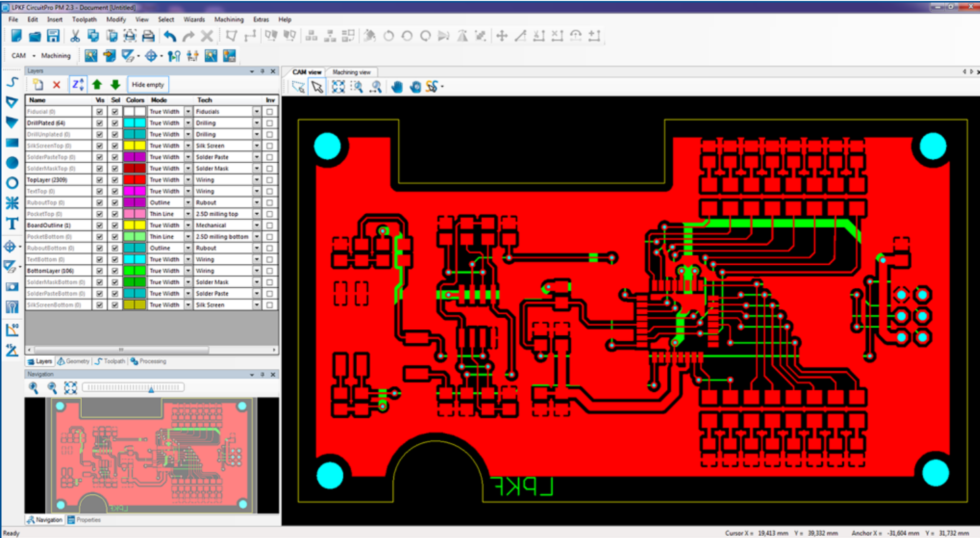


Fig. 47: Imported data in the *CAM view*

☑ The data have been imported in CircuitPro PM 2.3.

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 of the ***Layer/Template*** column, select which artwork should be assigned to which layer respectively.

Assigning holes to the layer *DrillUnplated* in CircuitPro PM 2.3

The holes that do not have a “copper ring” (i.e. 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. Click on all holes that are to be assigned to the ***DrillUnplated*** layer while pressing the `Ctrl` key.

☐ The holes are highlighted in gray:

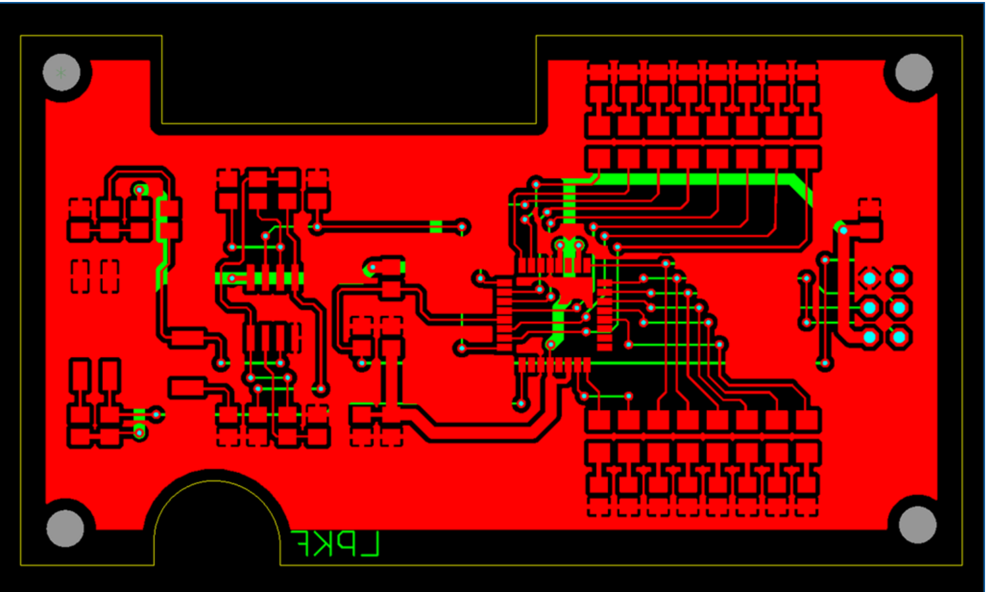


Fig. 48: *CAM view* after selecting holes

☐ Right-click on the layout and under *Assign objects to layer* select *DrillUnplated*:

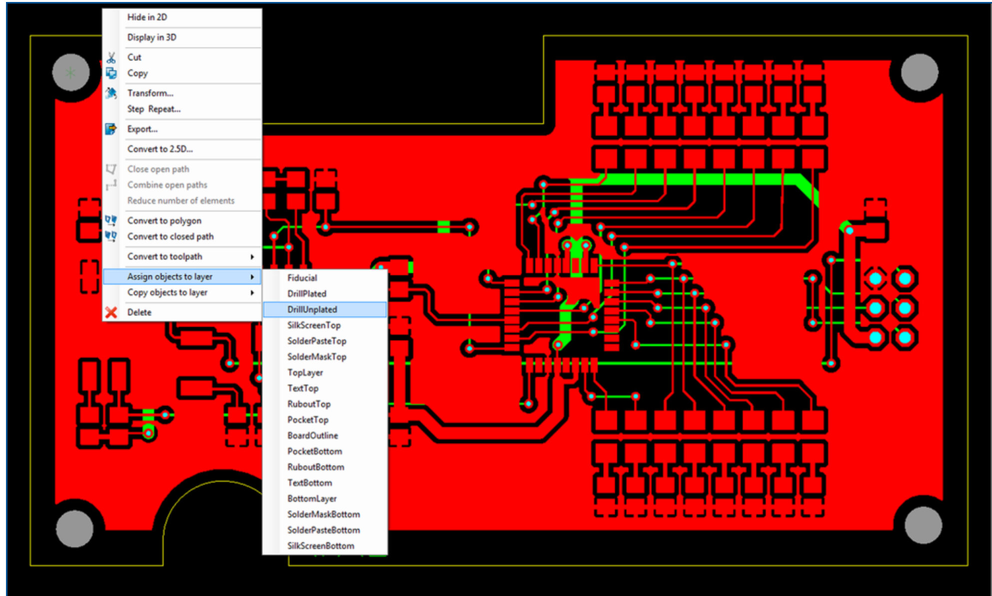


Fig. 49: Context menu *Assign objects* to *DrillUnplated* layer

☐ An additional layer is created automatically.

☑ The holes have been assigned to the layer *DrillUnplated* in CircuitPro PM 2.3.

Multiplying the layout in CircuitPro PM 2.3

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:

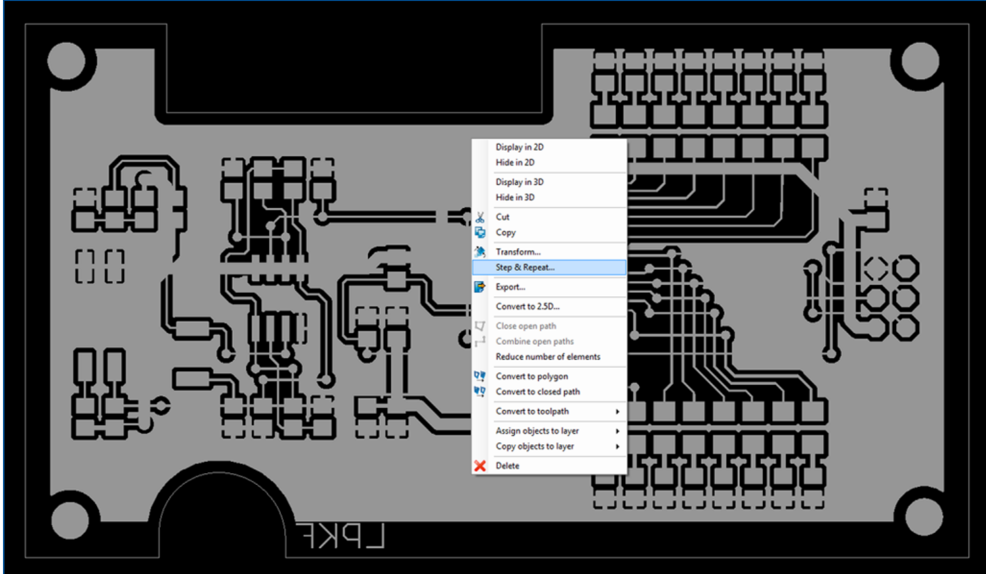


Fig. 50: Context menu *Step & Repeat*

3. Select *Step & Repeat*.
- The following dialog is displayed:

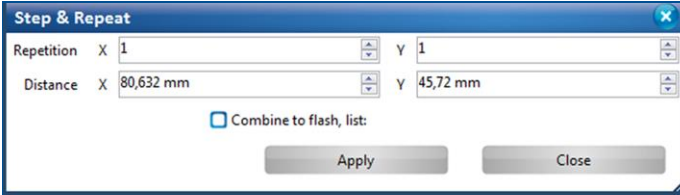


Fig. 51: Dialog *Step & Repeat*

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 needed 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 standard 2 mm contour routing tool.

☐ After entering all the values, the *Step & Repeat* dialog looks like this:

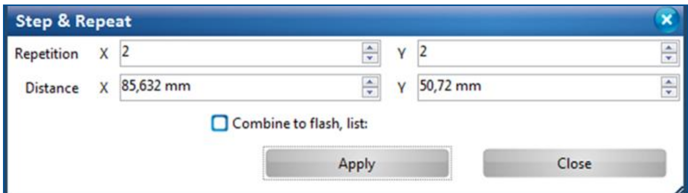


Fig. 52: Dialog *Step & Repeat* after entering values

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 Home key;
   – Click on .

☐ The *CAM view* looks like this:

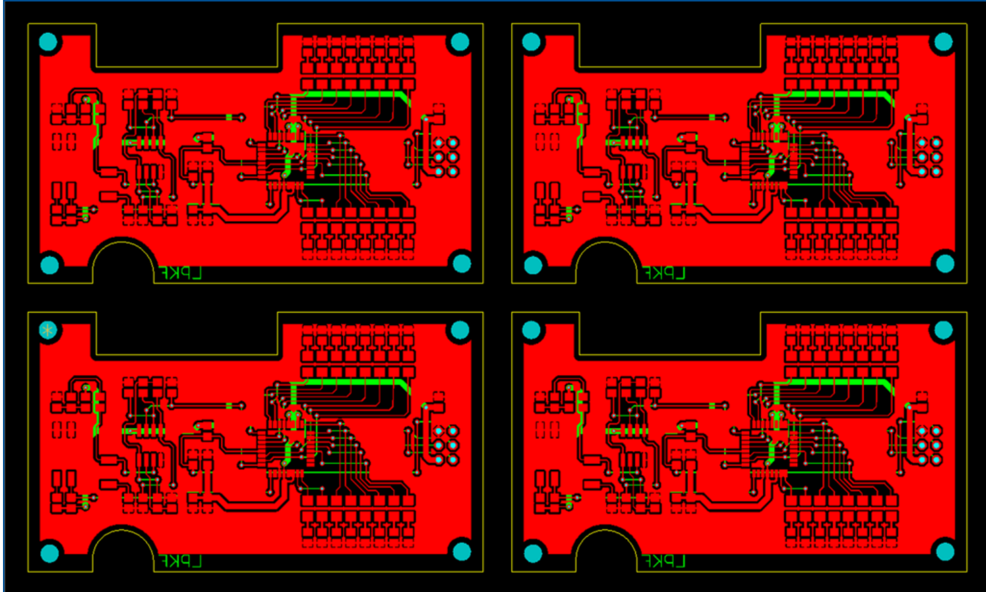


Fig. 53: *CAM view* of the multiplied layout

☑ The layout has been multiplied in CircuitPro PM 2.3.

Inserting fiducials in CircuitPro PM 2.3

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

☐ The following dialog is displayed:

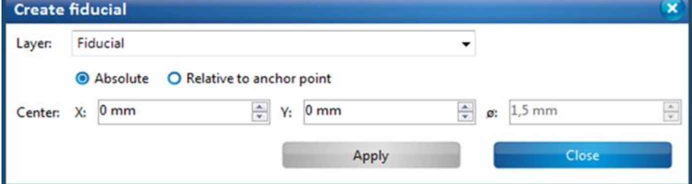


Fig. 54: Dialog *Create fiducial*

2. Click on the desired position in the layout to insert the fiducials there.

The best fiducial positions are just beyond the corners of the board.
A total of four fiducials are recommended for best alignment; using three fiducials is a good way to avoid wrong board orientation. At least two fiducials are required for correct operation of the process.

3. Click on [Close] or press Esc.

☐ The *CAM view* looks like this:

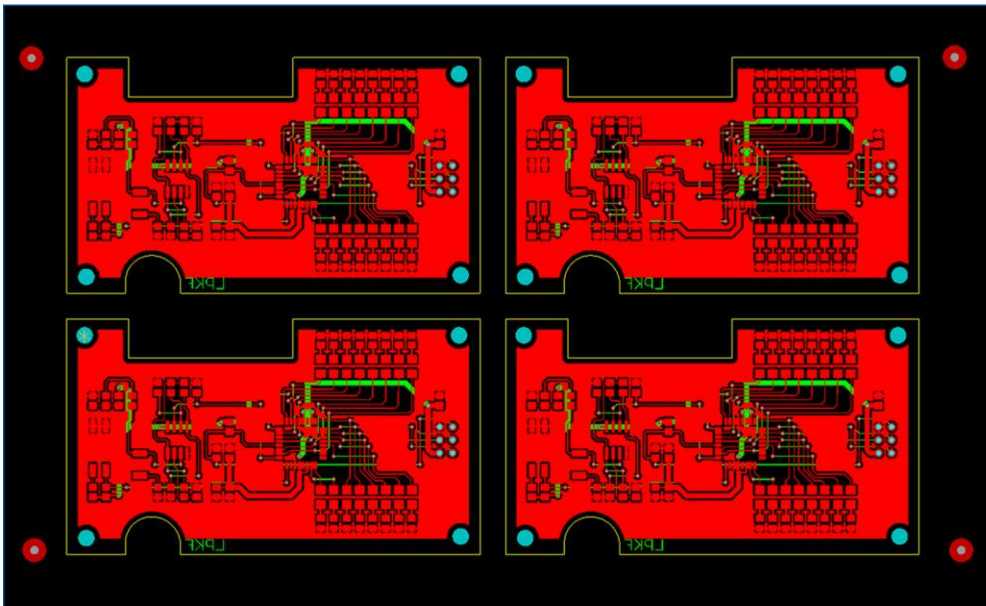


Fig. 55: *CAM view* after inserting fiducials

☑ The fiducials have been inserted in CircuitPro PM 2.3.

Saving the file in CircuitPro PM 2.3

1. Click on *File > Save As…*.
2. Select a suitable folder, name the file with the suffix ***_PL*** and click on [Save].
3. Click on File > Save As….
4. Select a suitable folder, name the file with the suffix ***_PM*** and click on [Save].
5. Continue working with the file with the suffix ***_PM***.

The file with the suffix ***_PL*** will be used in **CircuitPro PL 3.0** for structuring the PCB with the ProtoLaser.
The file with the suffix ***_PM*** will be used in **CircuitPro PM 2.3** for drilling and cutting with the ProtoMat.

☑ The file has been saved in CircuitPro PM 2.3.

Creating toolpaths in CircuitPro PM 2.3

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

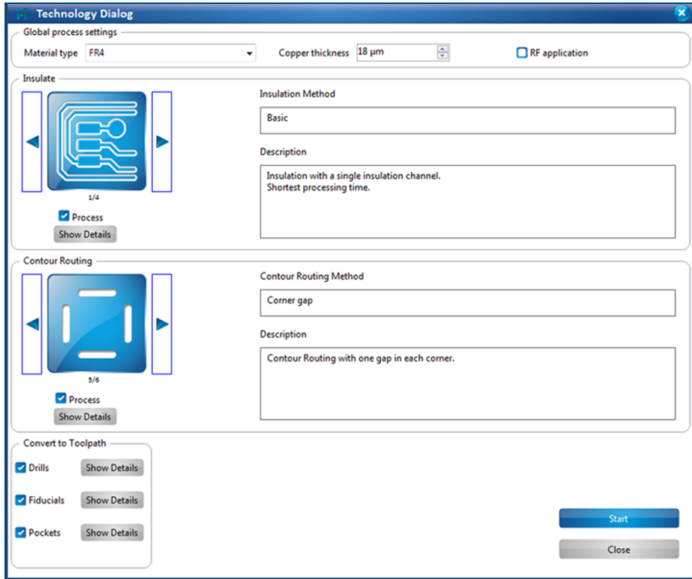


Fig. 56: Dialog *Technology*

2. Deactivate the check box *Process* in the *Insulate* group, as this step is be completed in CircuitPro PL 3.0.
3. Select *Edge gaps* in the *Contour Routing* group using the arrow buttons. This is usually the preferred method of contour routing.
4. Deactivate the check box *Pockets* in the *Convert to Toolpath* group, since there are no “pockets” in this project.
5. Click on [Start].
- ☐ The software creates all toolpaths and identifies all required tools. A report of the required tools is displayed:

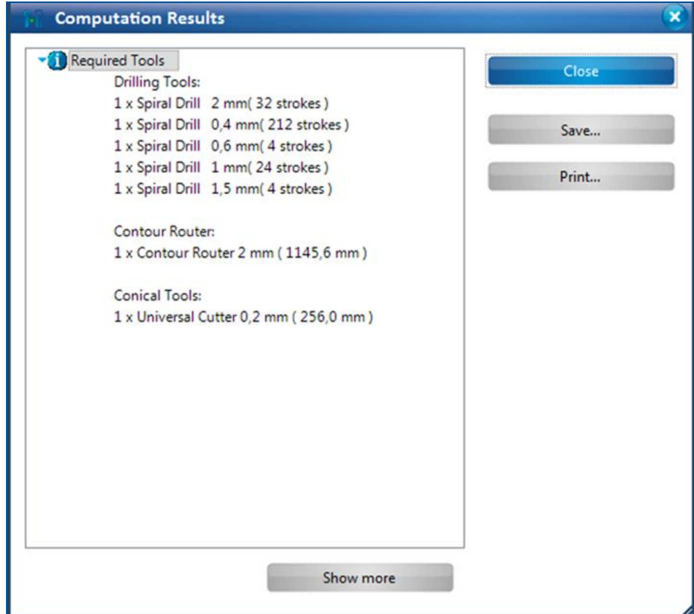


Fig. 57: Message *Computation Results*

6. Check the computation results for any possible warnings or errors and make corrections, if needed.

7. Click on [Close].
8. Click on *File* > *Save* or on to save changes in the file with the suffix _***PM***.

☑ The toolpaths have been created in CircuitPro PM 2.3.

For detailed information on contour routing types and settings refer to the CircuitPro PM 2.3 Compendium, chapter 5.4.3.

Processing a PCB (with ProtoMat)

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

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 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. In the message *Processing Phase: DismountMaterial* click on [OK] and proceed with galvanic plating.

☑ The PCB has been processed.

For detailed information on the ProtoMat phases refer to the CircuitPro PM 2.1 How-to guides, Part I, chapters 1.8 and 1.9.

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

1. Switch on the system.
2. Select a profile.
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. Swipe the activator from the PCB.
9. Copper-plate the PCB.
10. Switch off the system.

☑ The through-holes have been galvanically plated.

For detailed information on the galvanic through-hole plating refer to the chapter 6.3 of the **Contac S4** user manual.

Tips for more efficient galvanic through-hole plating

- Remove the soot around the laser drilled holes using a cloth and isopropyl alcohol.
- In case of oxidation, clean it using a soft brush and LPKF Cleaner.
- Use compressed air to remove drilling debris from the holes.
- Use compressed air to remove the water from the 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 3.0

For detailed information on preparing the data with CircuitPro PL 3.0 refer to chapter 2.1.

1. At the bottom of the *Templates* tab of the *New document* dialog, activate the check box *Supported by ProtoMat* and activate the check box of the laser system you are working with (in this example the *ProtoLaser U4* is used).
2. Select the template *PL-U4_PM_DoubleSided_GalvanicTHP.cp2d*.
3. Click on [Load template].
4. Select the *Machine type* and the *Material type* (in this example the material type *FR4_1.55mm_Cu35 (laminated)* is used).
5. Enter the *Material thickness*.
6. Click on [OK].
7. Click on *File* > *Import old version*.
8. Navigate to the folder that contains the file with the suffix ***_PL*** you previously generated with CircuitPro PM 2.3.
9. Select the *_***PL*** file and click on [Open].

☐ Your layout is displayed in the *Layout* view:

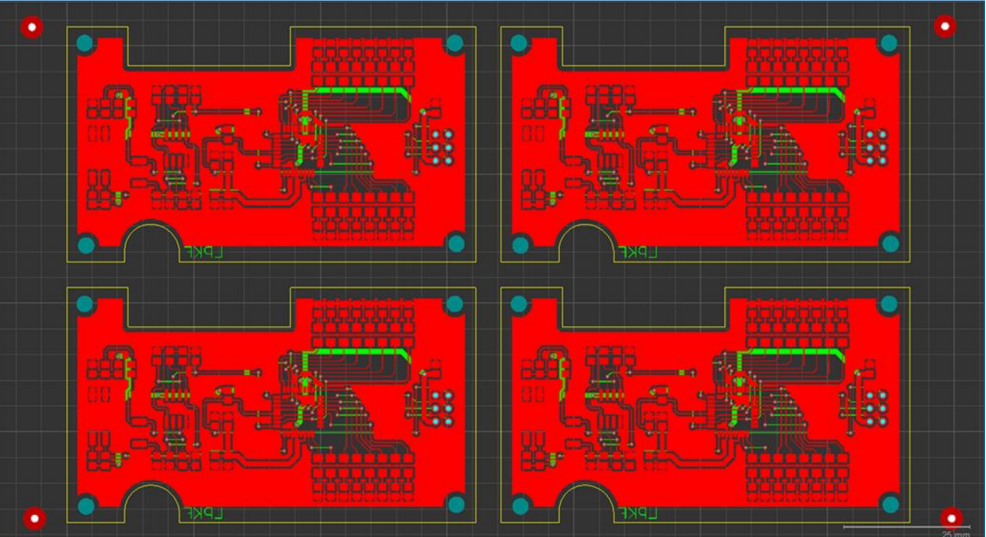


Fig. 58: *Layout* view of the imported layout

10. In the *Workflow setup* pane, right-click on the section *Toolpaths* and click on the context menu item *Compute all…*
11. Check the computation results for any possible warnings or errors and make corrections, if needed.
12. Click on [Close].
13. Save your project by clicking on *File* > *Save As* or on .

☑ The data have been prepared in CircuitPro PL 3.0.

Usual copper thicknesses and tool libraries

If the board was galvanically plated, the additional copper thickness must be considered when choosing a tool library. Choose the tool library according to the final copper thickness.

Examples of usual copper thicknesses and tool libraries:

Original copper thickness on FR4	Added galvanic deposition of copper	Combined copper thickness	Tool Library
5 µm	12 µm	17 µm	FR4_1.55mm_Cu18
5 µm	25 µm	30 µm	FR4_1.55mm_Cu35
18 µm	12 µm	30 µm	FR4_1.55mm_Cu35

Table 5: **Usual copper thicknesses and tool libraries**

Structuring the PCB (with ProtoLaser)

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

1. Measure the **thickness** of the board with a caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing* view on ▶.

☐ After warm-up the following message is displayed:

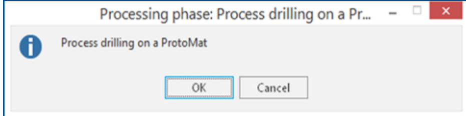


Fig. 59: Message *Processing phase: Process drilling*

3. Click on [OK].

☐ The following message is displayed:

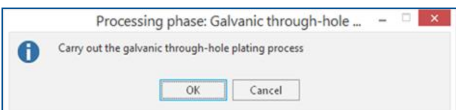


Fig. 60: Message *Processing phase: Galvanic through-hole plating*

4. Click on [OK].

☐ The following message is displayed:

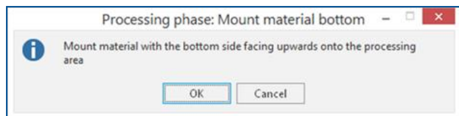


Fig. 61: Message *Processing phase: Mount material bottom*

5. Place the board with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.
6. In the *Processing* view, click on .
7. Click on [OK].
8. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
9. Move the *Placement* dialog off to the left side.
10. 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, Project placement, Matching the fiducial position.

11. After project placement is complete click on [Continue].

☐ The laser system reads the fiducials.

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

☐ When fiducial recognition is completed, the structuring of the Bottom side (*BottomLayer*) is started.

12. When the message *Processing Phase: Flip material* appears, turn the PCB over around the symmetry axis of the system and click on [OK].
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 [Continue].

☐ The laser system reads the fiducials on the Top side (*TopLayer*) and structuring is started.

15. After structuring of the Top side is finished, dismount the PCB.

☑ The PCB has been structured.

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

1. In CircuitPro PM 2.3 click on *File* > *Open* or on .

☐ The following dialog is displayed:

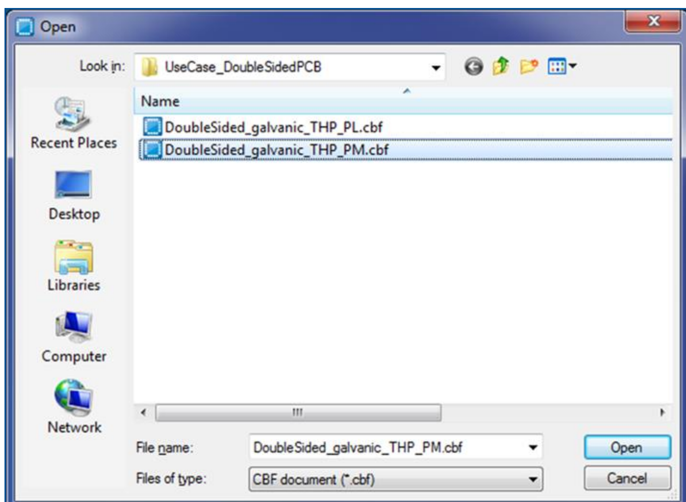


Fig. 62: Dialog *Open*

2. Navigate to the folder that contains the file with the suffix _***PM*** you previously generated with CircuitPro PM 2.3.
3. Select the *_***PM*** file and click on [Open].

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

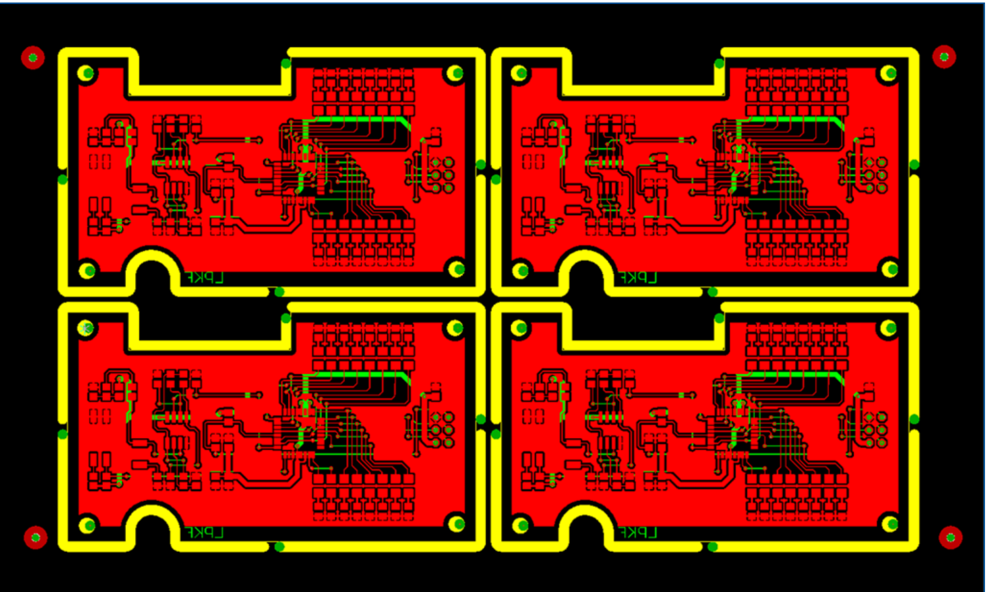


Fig. 63: *CAM view* of the opened project

4. Place the board onto the system's processing table with the **Top side** (*TopLayer*) **facing up** and fasten it using adhesive tape.
5. Change to the *Machining view*.

☐ Your project is displayed in *Machining view*:

Fig. 64: ***Machining view*** **of the opened project**

6. In the pane *Processing*, select *ReadFiducialsTop* in the drop-down list:

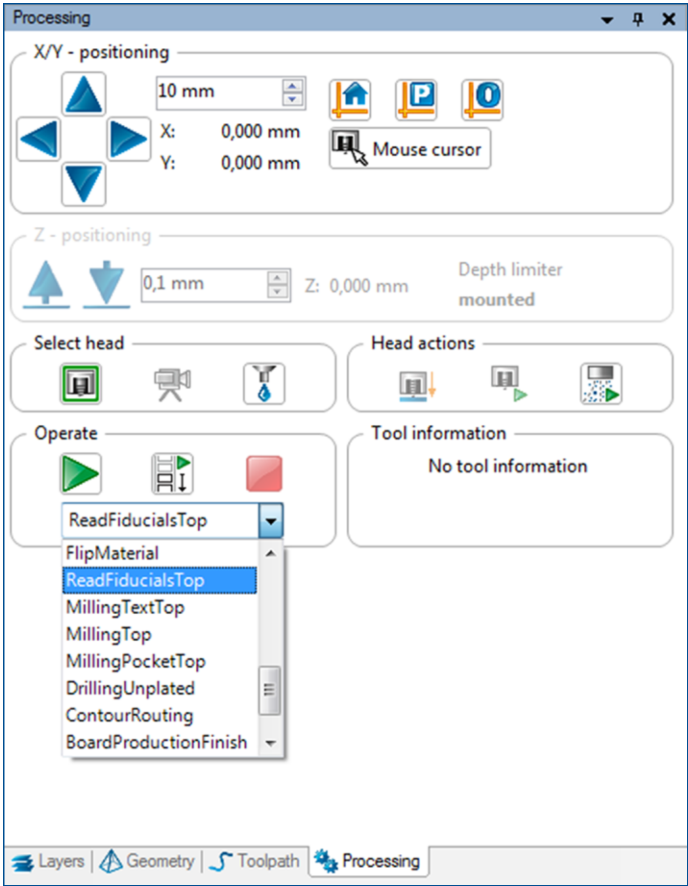


Fig. 65: Drop-down list of the processing phases

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

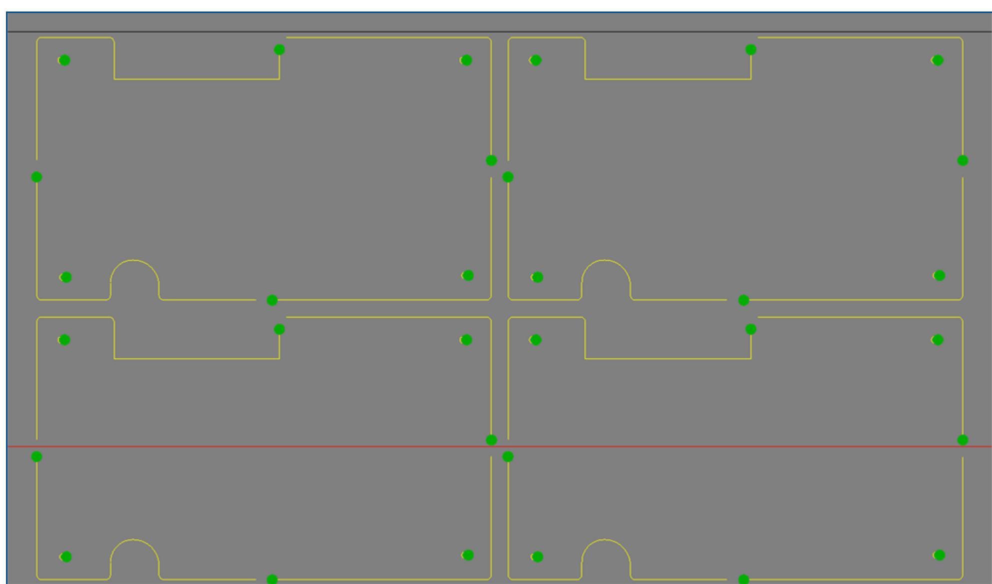


Fig. 66: ***Machining view* of drilling and contour-routing data**

7. Click on .

☐ The *ReadFiducialsTop* phase is performed.

For detailed information on fiducial recognition refer to CircuitPro 2.1 How-to guides, Part I, chapter 1.9.

☐ 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 the PCB, rinse with tap water and dry it with compressed air.
3. Break or cut the breakout tabs.

☑ The process has been completed.

The PCB production is finished.

Cleaning the ProtoMat

The processing area has to be cleaned if heavily soiled.

▶ Use a vacuum cleaner to remove chips and residues from the processing area.

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

This chapter describes how to create a double-sided circuit board with the use of a UV laser system, circuit board plotter and non-galvanic through-hole plating system.

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

Consumables

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

Auxiliaries

- 1 set of cutting tools (drills/contour routers for LPKF ProtoMat)
- LPKF Cleaner (order no. 115891)
- Hot air oven (order no. 115877)
- Oil-free compressed air
- Tap water

System

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

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PM 2.3
- Drilling fiducials (with ProtoMat)
- Preparing the data in CircuitPro PL 3.0
- Structuring the PCB (with ProtoLaser)
- Drilling unplated through-holes and cutting out the PCB (with ProtoMat)
- Drilling plated through-holes (with ProtoMat)
- Plating the through-holes (with ProConduct)

Preparing the data in CircuitPro PM 2.3

For detailed information on preparing the data in CircuitPro PM 2.3 refer to chapter 2.2.

1. In the *Templates* tab of the *New document* dialog, select the template: *ProtoLaser – ProtoMat – 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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PM 2.3\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. Save two copies of the file. Name one file with the suffix ***_PL*** and the other one with the suffix ***_PM***.
11. Continue working on the file with the suffix ***_PM***.
12. Click on *Toolpath* > *Technology Dialog* or on .
13. Click on [Start].

☐ The computation results are displayed in a dialog.

14. Click on [Close].
15. Save the changes in the ****_PM*** file.

☑ The data have been prepared in CircuitPro PM 2.3.

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: FlipMaterial* 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 ProtoMat phases refer to the CircuitPro 2.1 How-to guides, Part I, chapters 1.8 and 1.9.

Preparing the data in CircuitPro PL 3.0

For detailed information on preparing the data in CircuitPro PL 3.0 refer to chapter 2.1.

1. In the *Templates* tab of the *New document* dialog, select the template *PL-U4_PM_DoubleSided_ProConduct.cp2d*.
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type, Material type* and enter the *Material thickness*. In this example the material type *FR4_1.55mm_Cu18 (laminated)* is used.
4. Click on [OK].
5. Click on *File* > *Import old version*.
6. Select the ***_PL*** file you previously generated with CircuitPro PM 2.3.
7. Click on [Open].
8. In the *Workflow setup* pane, right-click on the section *Toolpaths* and click on the context menu item *Compute all…*
9. Check the computation results for any possible warnings or errors and make corrections, if needed.
10. Click on [Close].
11. Save your project.

☑ The data have been prepared in CircuitPro PL 3.0.

Structuring the PCB (with ProtoLaser)

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

1. Measure the **thickness** of the board with a caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing* view on ▶.
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 board with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table and click on [OK].
5. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
6. Place the processing data **matching** 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, Project placement, Matching the fiducial position.

7. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials.

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

☐ The Bottom side is being structured.

8. When the message *Processing Phase: Flip material* is displayed, turn the board over around the symmetry axis of the system and click on [OK].
9. Place the processing data **matching** the location of the PCB and fiducials on the processing table.
10. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side (*TopLayer*) and starts structuring.

11. After structuring of the top side is finished, dismount the board.

☑ 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 board with ProtoMat refer to chapter 2.2.

1. Open the ***_PM*** file you previously generated with CircuitPro PM 2.3.
2. Place the board onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** and fasten it using adhesive tape.
3. Switch to the pane *Processing*.
4. Select *ReadFiducialsTop* from the drop-down list and click on .

☐ The phases *ReadFiducialsTop, MarkingDrills, DrillingUnplated* and *ContourRouting* are performed.

5. When the message *Processing Phase: DismountMaterial* is displayed, remove the PCB from the system and click on [OK].
6. Rinse the PCB with tap water and dry it using compressed air.

☑ The unplated through-holes have been drilled and the board has been cut out.

The PCBs **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* 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 *Placement_6* phase.

☐ 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 film and expose them if necessary.

☑ The plated through-holes have been drilled.

Plating the 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 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 5-10 on the other side.
9. Slowly peel off the 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 board from the board.

☑ The through-holes have been plated.

For detailed information on non-galvanic through-hole plating refer to the chapter 3 of the **ProConduct** process description.

The PCB production is finished.

Multi-layer PCBs

3 Producing multi-layer PCBs

This chapter describes the production of multi-layer circuit boards. Four different processes are applied:

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

The following LPKF systems are required for the different processes:

Process	LPKF system
1	ProtoLaser, MultiPress, Contac S4
2	ProtoLaser, MultiPress, ProtoMat S or E, Contac S4
3	ProtoLaser, MultiPress, ProtoMat S or E, ProConduct
4	ProtoLaser, MultiPress, ProtoMat S or E, Contac S4

Table 6: Required LPKF systems

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

This chapter describes how to create a **4-layer** circuit board using a UV 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

- Base plate FR4 18/18 µm, 305 (k) mm x 229 mm x 1 mm (order no. 119574)
- Thin Laminate 104 ML 0/5 µm, 305 (k) x 229 x 0.2 mm with protection foil (order no. 119571)
- Prepreg Type 2125, 275 mm (k) x 200 mm x 0.1 mm (order no. 119572)

Auxiliaries

- LPKF Cleaner (order no. 115891)
- Isopropyl alcohol
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4
- MultiPress S
- Contac S4

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PL 3.0
- Structuring the core (with ProtoLaser)
- Preparing the laminate materials (with ProtoLaser)
- Assembling and pressing the multi-layer stack (with the MultiPress S)
- Drilling plated through-holes into the multi-layer PCB (with ProtoLaser)
- Galvanic through-hole plating of the multi-layer PCB (with Contac S4)
- Structuring the outer layers of the multi-layer PCB (with ProtoLaser)

Preparing the data in CircuitPro PL 3.0

For detailed information on preparing data with CircuitPro PL 3.0 refer to chapter 2.1.

1. In the *Templates* tab of the *New document* dialog, select the template *PL-U4_4Layer_GalvanicTHP_MultiPressS.cp2d*.
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type, Material type* and enter the *Material thickness*. For this procedure the material type *FR4_1.00mm_Cu18 (laminated)* was used.

A multi-layer PCB consists of different materials. For **each material** used during production, a **corresponding *Material type*** has to be selected.

4. Click on [OK].
5. Click on *File* > *Import* or on .
6. 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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PL 3.0\Example Data\UseCase_Multilayer`.
7. 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*.
8. Click on [Open].

☐ The data should automatically be assigned to the correct layers and the following dialog is displayed:

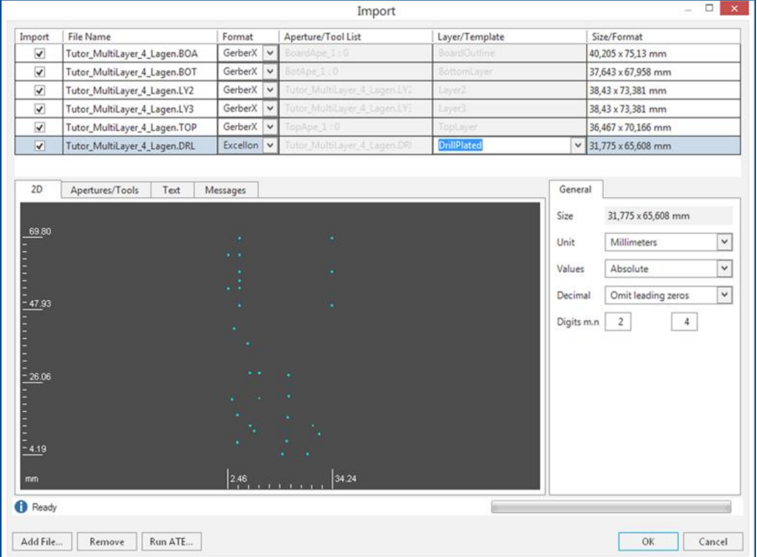


Fig. 67: Dialog *Import* and assigned layers

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

9. In the column *Layer/Template* select the layer *DrillPlated* from the drop-down list.
10. Click on [OK].

☐ The *Layout* view changes as follows:

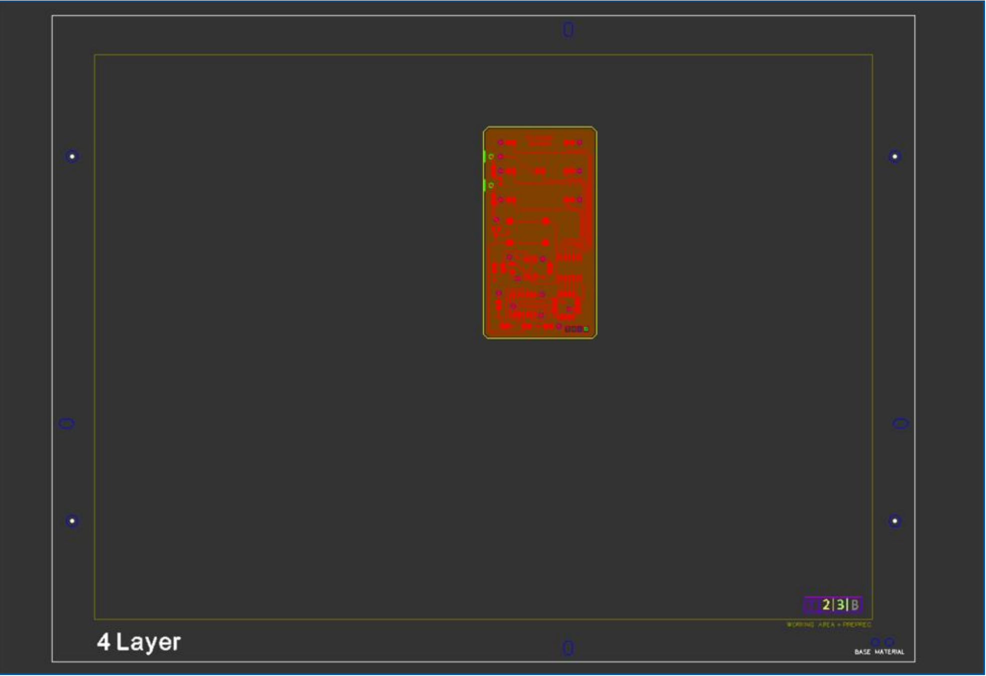


Fig. 68: *Layout* view of the imported data

11. If desired, multiply the layout. In this example the layout is not multiplied.
12. Select the entire layout.

☐ The layout is highlighted and changes its color.

13. Right-click on the highlighted layout.

☐ The following context menu is displayed:

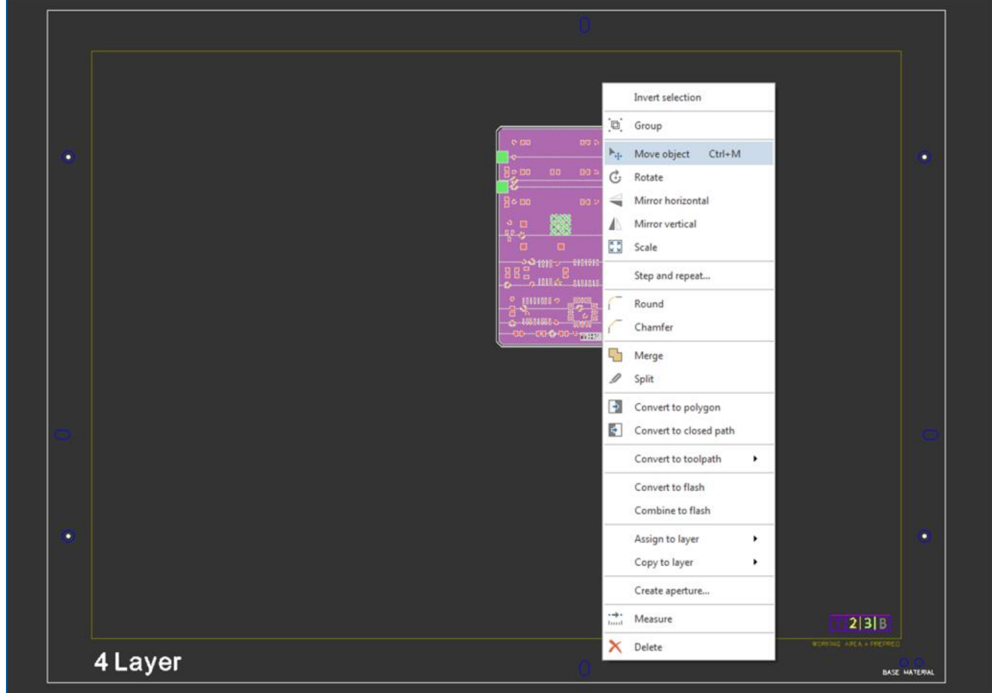


Fig. 69: Context menu *Move object*

14. Select *Move object* or click on [icon].

☐ A copy of the layout (in orange color) and the corresponding enter fields for specifying the reference point are displayed:

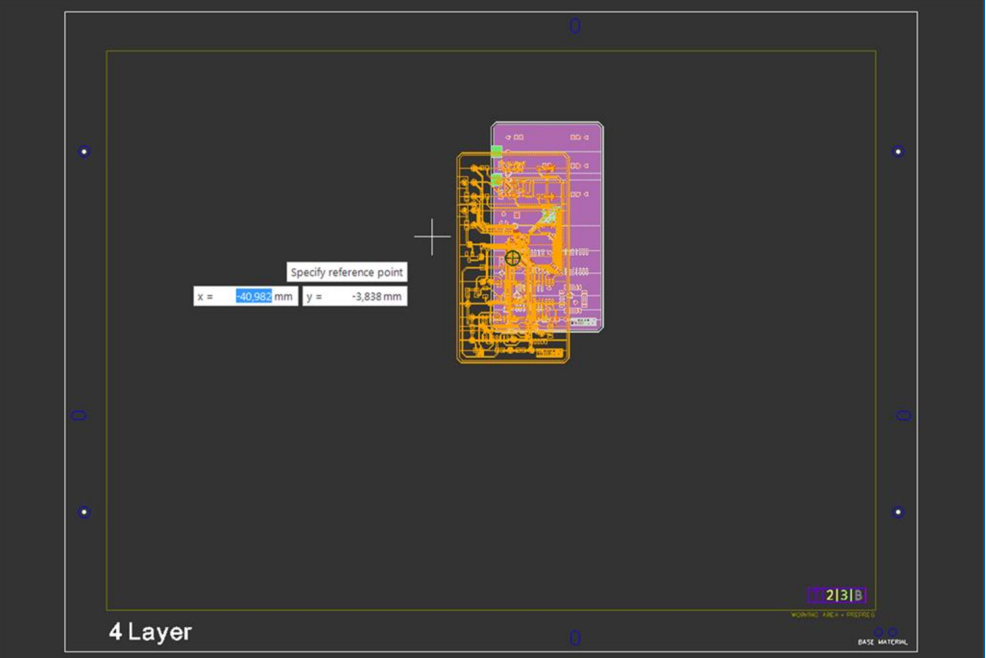


Fig. 70: Copy of the layout

15. Click on a point **in the original layout** to specify the **reference point**.

☐ The *Layout* view changes as follows:

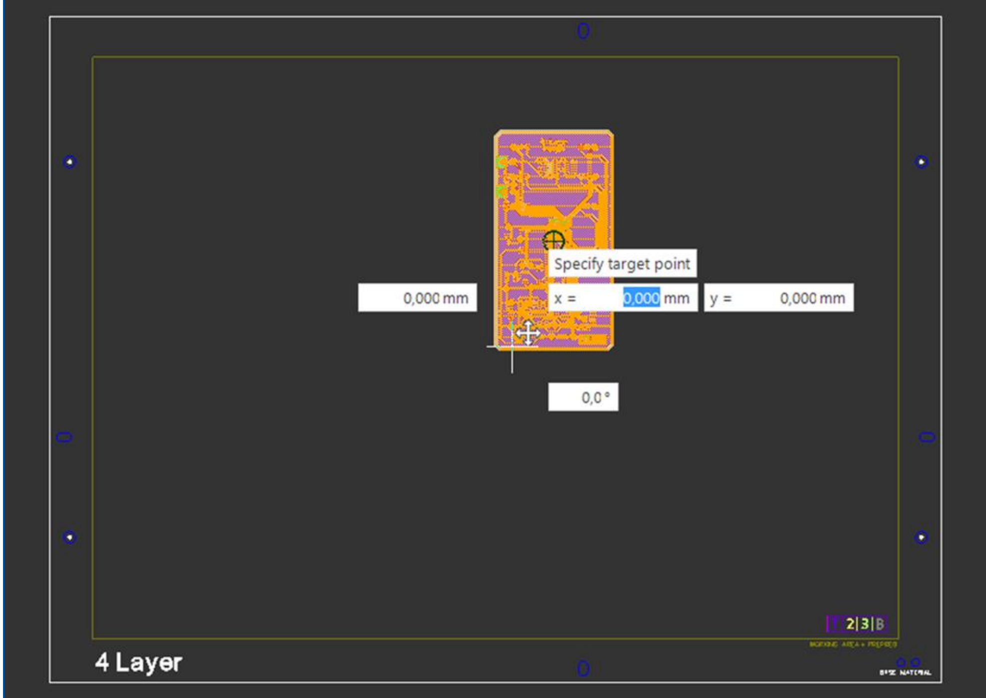


Fig. 71: Layout after specifying the reference point

☐ Move the copied layout with the mouse to the desired position:

Fig. 72: Desired position of the copied layout

16. When you reach the desired position, left-click to specify the **target point**.
17. In the Layout view click anywhere on the black area or press Esc in order to disable the *Move object* function.

☐ The layout has been moved to the desired location:

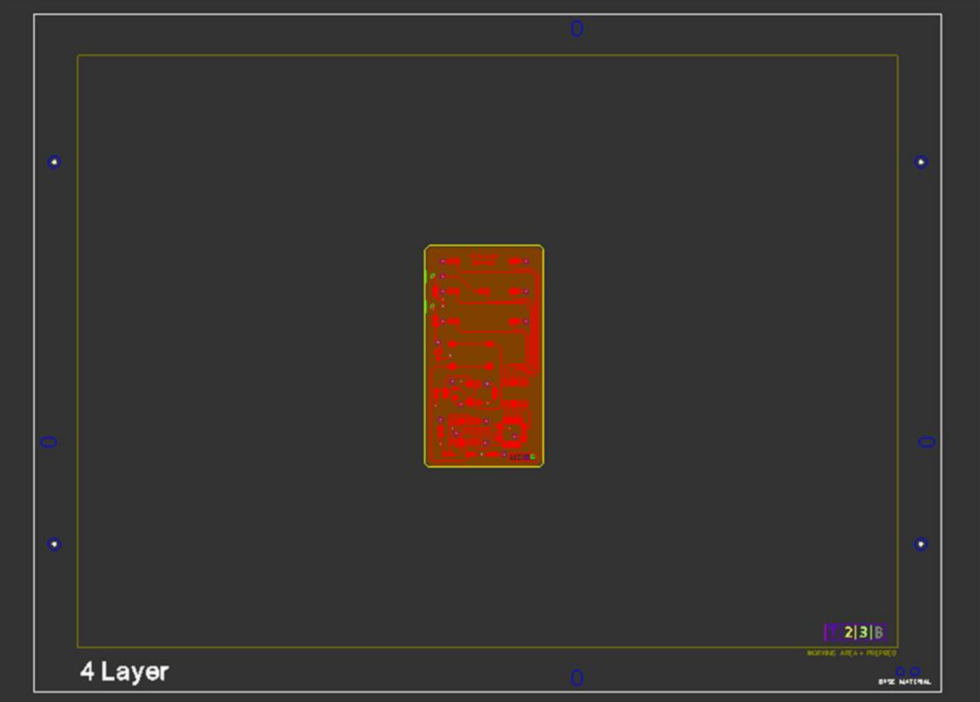


Fig. 73: Layout moved to the desired location

18. In the *Workflow setup* pane, right-click on the group *Toolpaths.*
19. Click on the displayed context menu item *Compute all…*
20. Check the *Computation Results* for any possible warnings or errors and make corrections, if needed.

21. Click on [Close].
22. Save the file with the suffix _***PL***.

☑ The data have been prepared in CircuitPro PL 3.0.

Structuring the core (with ProtoLaser)

1. Measure the thickness of the core with a caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing* view on ▶.

☐ After warm-up, the following message is displayed:

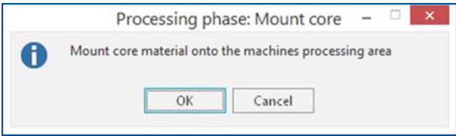


Fig. 74: Message *Processing phase: Mount core*

3. Place the core in the center of the processing table.
4. Click on [OK].
5. In the *Material settings* dialog, enter the *Material thickness.*
6. Click on [Continue].
7. Place the processing data **matching** the board position and size to the processing area used by the CircuitPro PL software.

For detailed information on project placement according to board position and size refer to chapter 1.1, Project Placement, Determining the processing area.

8. When project placement is complete, click on [Set center].

☐ The layout data are now precisely aligned and in the center of the core.

9. Click on [Continue].

☐ The laser system drills the fiducials into the core, the Bottom side of the core (*Layer3*) is being prepared (positioning holes and pin holes are being drilled) and the Bottom side of the core (*Layer3*) is being structured.

10. When the message *Processing phase: Flip material* is displayed, turn the core over around the symmetry axis of the system (the Top side of the core is facing upwards).

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

☐ The following figure displays the positioning holes:

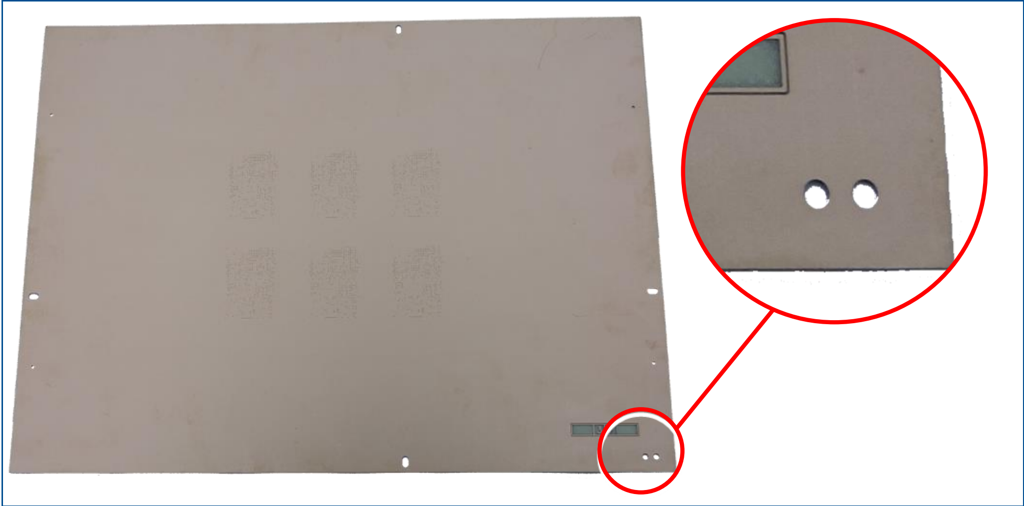


Fig. 75: Positioning holes in the right front corner of the core

11. Click on [OK].
12. Place the processing data **matching** the location of the core and fiducials on the processing table.

For detailed information on project placement according to fiducial positions refer to chapter 1.1, Project placement, Matching the fiducial position.

13. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side of the core (*Layer2*).

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

☐ The Top side of the core (*Layer2*) is being structured. When finished, the following message is displayed:

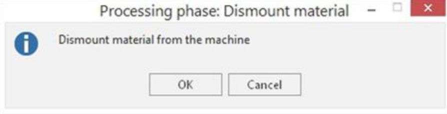


Fig. 76: Message *Processing phase: Dismount material*

14. Click on [Cancel] to temporarily stop the process.
15. Dismount the core from the system.
16. Spray the PCB with LPKF Cleaner and use a brush to clean it.
17. Rinse the PCB with tap water and dry it with compressed air.

☑ The core has been structured.

Preparing the laminate materials (with ProtoLaser)

1. Measure the thickness of the laminate with a caliper gauge.
2. Click on *Processing > Material Settings* or on .

☐ The *Material settings* dialog is displayed.

3. Select the *Machine type, Material type* and enter the *Material thickness* (in this example the material type *ML104_0.20mm_Cu05 (laminated)* is used).
4. Click on [OK].
5. Switch to the *Processing* view.
6. In the *Workflow setup* pane expand the group *Processing*.

☐ A list of processing phases is displayed:

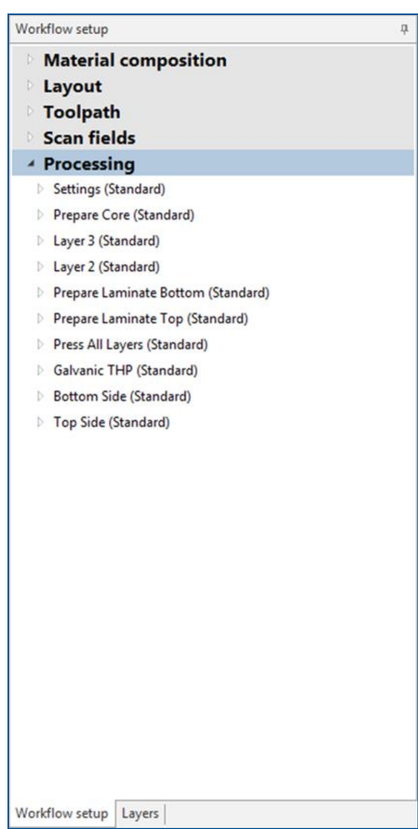


Fig. 77: Group *Processing* and list of processing phases

7. Right-click on *Prepare Laminate Bottom (Standard)*.

☐ The following context menu is displayed:

Workflow setup
Material composition
Layout
Toolpath
Scan fields
Processing
Settings (Standard)
Prepare Core (Standard)
Layer 3 (Standard)
Layer 2 (Standard)
Prepare Laminate Bottom (Standard)
Prepare Laminate Top
Press All Layers (Stand
Galvanic THP (Standar
Bottom Side (Standard
Top Side (Standard)
Compute toolpaths
Compute scan fields
Add scan field
Hide scan fields
Process
Process from here
Set phase ID
Set mirrored
Change sorting
Add work package
Export... Ctrl+E
Rename
Delete
Workflow setup Layers

Fig. 78: Context menu *Processing*

8. Select *Process from here*.

☐ The following message is displayed:

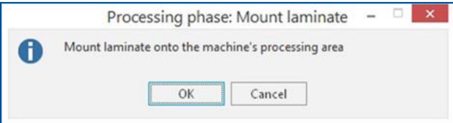


Fig. 79: Message *Processing phase: Mount laminate*

9. Place the laminate with the **copper side facing upwards** on the center of the processing table.

The laminate is very thin and for this reason it is usually bent. Click on and fasten the laminate along its edges with adhesive tape.

10. Click on [OK].
11. In the *Material settings* dialog, enter the *Material thickness.*
12. Click on [Continue].
13. Place the processing data **matching** the board position and size to the processing area used by the CircuitPro PL software.
14. When project placement is complete, click on [Set center].

☐ The layout data are now precisely aligned and in the center of the laminate.

15. Click on [Continue].

☐ The laminate (*BottomLayer*) is being prepared (positioning holes and pin holes are being drilled).

16. When the message *Processing phase: DismountMaterial* is displayed, remove the laminate from the system and click on [OK].
17. When the message *Processing phase: MountLaminate* is displayed, place the second laminate with the **copper side facing upwards** on the center of the processing table.

18. Click on [OK].
19. Place the processing data **matching** the board position and size to the processing area used by the CircuitPro PL software.
20. When project placement is complete, click on [Set center].

☐ The layout data are now precisely aligned and in the center of the second laminate.

21. Click on [Continue].

☐ The laminate (*TopLayer*) is being prepared (positioning holes and pin holes are being drilled).

22. When the message *Processing phase: DismountMaterial* is displayed, remove the laminate from the system and click on [OK].

☑ The laminate materials have been prepared.

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

For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.3, Multi-layer process.

1. Start the pre-heating process of the MultiPress S.
2. Prepare the materials.
3. Assemble the multi-layer stack in the press mold.
4. Press the multi-layer stack.
5. Cure the multi-layer stack.

After pressing of the multi-layer stack, it must **cure** at ambient temperature for at least **12 to 18 hours**.

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

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

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

1. Measure the thickness of the multi-layer PCB with a caliper gauge.
2. Click on *File* > *Open or* on .
3. Select the ***_PL** file you previously saved in CircuitPro PL 3.0.
4. Click on [Open].

☐ Your layout is displayed in the *Layout* view.

5. Click on *Processing* > *Material Settings* or on .
6. In the *Material settings* dialog select the *Machine type, Material type* and enter the *Material thickness* (in this example the material type *FR4_1.55mm_Cu35 (laminated)* is used).
7. Click on [OK].
8. Switch to the *Processing* view.
9. In the *Workflow setup* pane expand the group *Processing*.
10. Right-click on *Galvanic THP (Standard)* and select *Process from here*.

☐ The following message *Processing phase: Mount material top* is displayed:

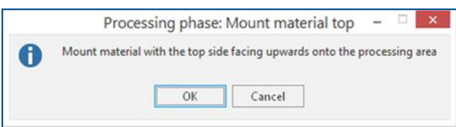


Fig. 80: Message *Processing phase: Mount material top*

11. Place the multi-layer PCB with the **Top side** (*TopLayer*) **facing upwards** (the positioning holes should be located in the right front corner) onto the processing table.
12. Click on [OK].
13. In the *Material settings* dialog, enter the *Material thickness.*
14. Click on [Continue].
15. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
16. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side (*TopLayer*) and the plated through-holes are being drilled.

17. When the message *Processing phase: Dismount material* is displayed, dismount the multi-layer PCB from the system.
18. Remove the soot around the laser drilled holes using a cloth and isopropyl alcohol.
19. Click on [OK].
20. When the message *Processing phase: Galvanic through-hole plating* is displayed, click on [OK] and proceed with galvanic through-hole plating.

☑ The plated through-holes have been drilled.

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

1. Switch on the system.
2. Select a profile.
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. Swipe the activator from the multi-layer PCB.
9. Copper-plate the multi-layer PCB.

   The multi-layer PCB must remain in **tank 5 for 160 minutes** in order to achieve the **final copper thickness** of approximately **30 µm** on **all layers**. This way, the surfaces of the holes are sufficiently plated and the laser rubout process functions properly.
10. Switch off the system.

☑ The multi-layer PCB has been galvanically through-hole plated.

For detailed information on the galvanic through-hole plating process refer to the Contac S4 User manual, chapter 6.3.

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

1. When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*BottomLayer*) **facing upwards** (the positioning holes should be located in the right rear corner) onto the processing table.
2. Click on [OK].
3. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
4. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Bottom side (*BottomLayer*) and starts structuring.

5. When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system (the positioning holes should be located in the right front corner) and click on [OK].
6. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
7. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side (*TopLayer*), unplated through-holes are being drilled, top side is being structured and the multi-layer PCB is being cut out.

8. After structuring of the Top side (*TopLayer*) is finished, dismount the multi-layer PCB from the system.
9. Spray the board with LPKF Cleaner and use a brush to clean the board.
10. Rinse the board under tap water and dry it with compressed air.

☑ The outer layers of the multi-layer PCB have been structured.

The 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** circuit board using a UV 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

- Base plate FR4 18/18 µm, 305 (k) mm x 229 mm x 1 mm (order no. 119574)
- Thin Laminate 104 ML 0/5 µm, 305 (k) x 229 x 0.2 mm with protection foil (order no. 119571)
- Prepreg Type 2125, 275 mm (k) x 200 mm x 0.1 mm (order no. 119572)

Auxiliaries

- 1 set of cutting tools (drills/contour routers for LPKF ProtoMat)
- LPKF Cleaner (order no. 115891)
- Oil-free compressed air
- Tap water

System

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

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PM 2.3
- Preparing the core and laminate materials (with ProtoMat)
- Preparing the data in CircuitPro PL 3.0
- Structuring the core (with ProtoLaser)
- Assembling and pressing the multi-layer stack (with the MultiPress S)
- Drilling plated through-holes into the multi-layer PCB (with ProtoMat)
- Galvanic through-hole plating of 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 2.3

For detailed information on preparing data with CircuitPro PM 2.3 refer to chapter 2.2.

1. In the *Templates* tab of the *New document* dialog, select the template: *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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PM 2.3\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 should automatically be assigned to the correct layers and the following dialog is displayed:

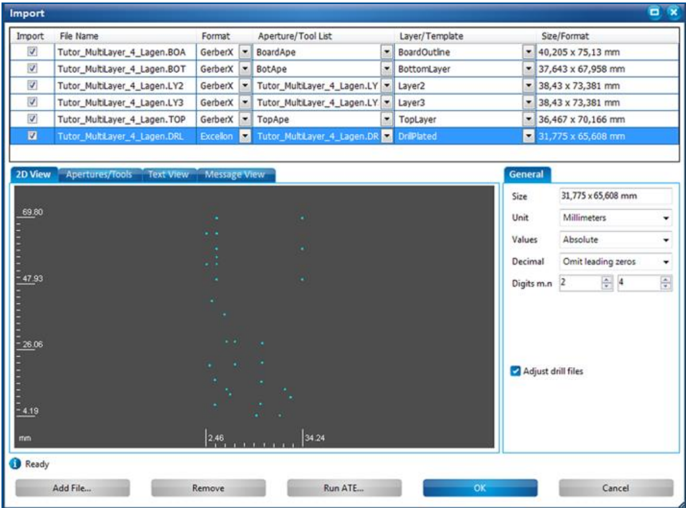


Fig. 81: Dialog *Import* and assigned layers

6. Assign the drill holes to the layer *DrillPlated* (in the *Layer/Template* column).
7. Click on [OK].

- The *CAM view* changes as follows:

Fig. 82: Imported data in the *CAM view*

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

8. Select all holes that are not to be plated (i.e. that are to be assigned to the *DrillUnplated* layer) by clicking on them while pressing `Ctrl` key.
- Selecting the holes changes their color.
9. Right-click on the layout and under *Assign objects to layer* select *DrillUnplated*.
- An additional layer is created automatically and the holes that are not to be plated are assigned to the *DrillUnplated* layer.
10. If desired, multiply the layout (in this example the layout is not multiplied).

Usually it is necessary to move the layout within the multi-layer base material frame. It is preferable to move it **to the center** of the multi-layer base material frame.

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.

Before creating toolpaths, all ***Index* toolpaths and layers** need to be **deleted**, as they are already included in the template for ProtoLaser.

14. Switch to the pane *Layers*.

☐ The view changes as follows:

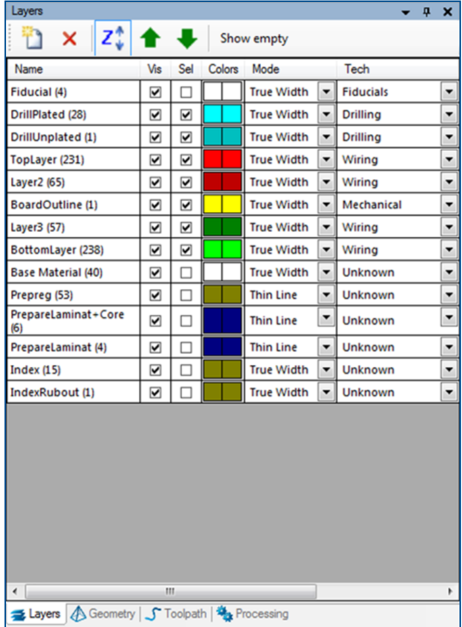


Fig. 83: Pane *Layers*

15. Activate all check boxes in the in the columns *Vis* and *Sel*.
16. Select the index layers and toolpaths in the layout.

☐ The index layers and toolpaths are highlighted in gray:

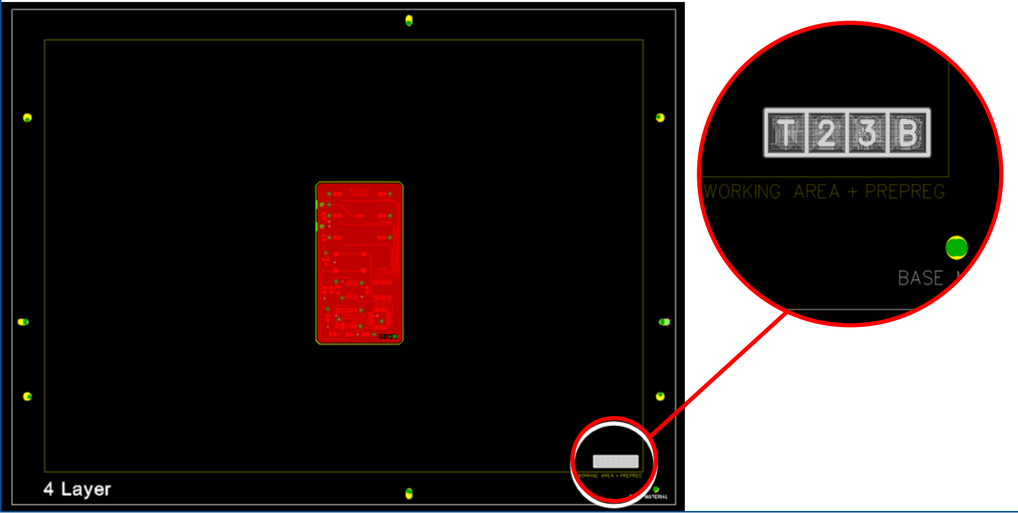


Fig. 84: *CAM view* after selecting index layers and toolpaths

17. Delete the selected layers and toolpaths by pressing [Del] or clicking on ❌.

☐ The selected layers and toolpaths have been deleted.

18. Click on *Toolpath* > *Technology Dialog* or on [icon].
19. Deactivate *Process* in the group *Insulate*.
20. Select *Edge gaps* in the group *Contour Routing*. This is usually the preferred method of contour routing.
21. Deactivate *Pockets* in the group *Convert to Toolpath*, since there are no “pockets” in the project for this tutorial.
22. Click on [Start].

☐ The *Computation results* are displayed in a dialog.

23. Click on [Close].

- The toolpaths have been calculated.

24. Save the file with the suffix _***PM***.

Delete all **toolpaths** and **auxiliary layers** before importing the data into CircuitPro PL 3.0. Only the initially imported layers should remain in your project.

25. Switch to the pane *Layers*.

- Select all **auxiliary layers** (including the *Fiducial* layer) by clicking on them while pressing the `Ctrl` key:

Fig. 85: Pane *Layers* and highlighted auxiliary layers

26. Delete the selected layers by pressing the key `Del` or clicking on ❌.

- The selected layers have been deleted.

27. Switch to the pane *Toolpath* and expand the *Toolpaths* folder.

- The view changes as follows:

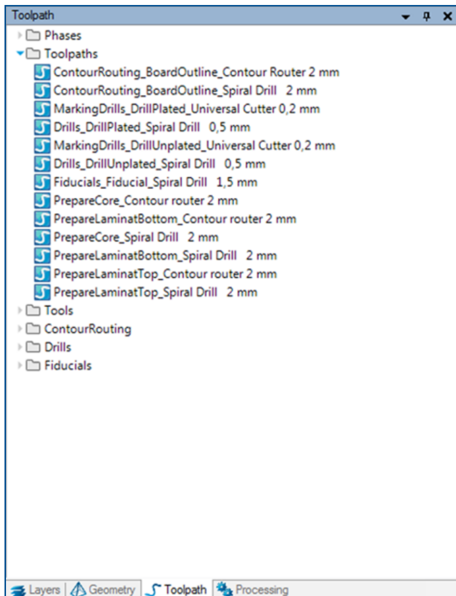


Fig. 86: List of toolpaths

- Select all toolpaths in this folder by clicking on them while pressing the `Ctrl` key:

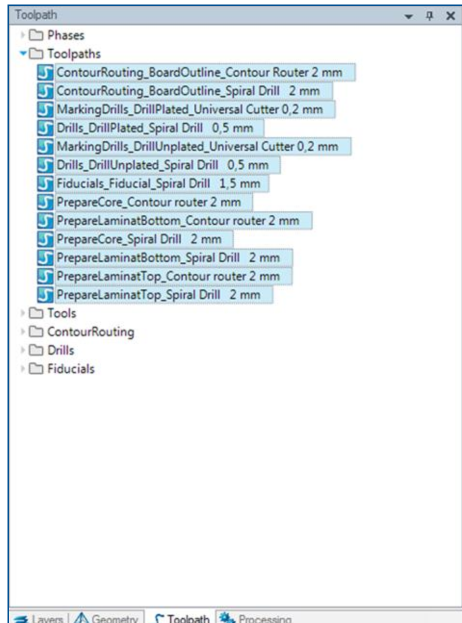


Fig. 87: Highlighted toolpaths

28. Delete them by pressing the key `Del` or clicking on ❌.

- The selected toolpaths have been deleted.

29. Save the file with the suffix ***_PL***.

☑ The data have been prepared in CircuitPro PM 2.3.

Preparing the core and laminate materials (with ProtoMat)

1. Open the ***_PM** file you generated in the previous procedure.
2. Load the tool magazine and assign the tools to positions.
3. Click on *Machining > Process all* or on ▶.
4. Perform the following phases (with the core material):
   - *MountMaterial*
   - *MaterialSettings*
   - *Placement*

☐ The phases *DrillFiducial* and *PrepareCore* are performed.

5. When the message *Processing Phase: FlipMaterial* is displayed, remove the core from the system and click on [Cancel] to temporarily stop the process.
6. Rinse the core with tap water and dry it with compressed air.
7. In the *Processing* pane, select *MountLaminate* from the drop-down list and click on .
8. Perform the following phases with the first laminate:
   - *MountLaminate*
   - *MaterialSettings_1*
   - *Placement_2*

☐ The *PrepareLaminateBottom* phase is performed.

9. When the message *Processing Phase: DismountMaterial_1* is displayed, remove the first laminate from the system and click on [OK].
10. Perform the following phases with the second laminate:
    - *MountLaminate_1*
    - *MaterialSettings_2*
    - *Placement_3*

☐ The *PrepareLaminateTop* phase is performed.

11. When the message *Processing Phase: DismountMaterial_2* is displayed, remove the second laminate from the system and click on [Cancel] to temporarily stop the process.

☑ The core and laminate materials have been prepared.

For detailed information on ProtoMat phases refer to the CircuitPro PM 2.1 How-to guides, Part II, chapter 3.8.

Preparing the data in CircuitPro PL 3.0

For detailed information on preparing data with CircuitPro PL 3.0 refer to chapter 2.1.

1. In the *Templates* tab of the *New document* dialog, select the template *PL-U4_PM_4Layer_GalvanicTHP_MultiPressS.cp2d*.
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type* and the *Material type* and enter the *Material thickness* (in this example the material type *FR4_1.00mm_Cu18 (laminated)* is used).

A multi-layer PCB consists of different materials. For **each material** used during production, a **corresponding *Material type*** has to be selected.

4. Click on [OK].
5. Click on *File* > *Import old version*.
6. Select the ***_PL*** file you previously generated with CircuitPro PM 2.3.
7. Click on [Open].

☐ The ***_PL*** file has been imported.

8. In the *Workflow setup* pane, right-click on the group *Toolpaths* and click on the context menu item *Compute all…*
9. Check the *Computation Results* message for possible errors and make corrections, if needed.
10. Click on [Close].
11. Save the file with the suffix ***_PL***.

☑ The data have been prepared in CircuitPro PL 3.0.

Structuring the core (with ProtoLaser)

1. Measure the thickness of the core with a caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing* view on .
3. After warm-up, the message *Processing phase: Process cutting on a ProtoMat* is displayed.
4. Click on [OK].
5. When the message *Processing phase: Mount core* is displayed, place the core with the **Bottom side** (*BottomLayer*) **facing upwards** (the positioning holes should be located in the right rear corner) in the center of the processing table.
6. Click on [OK].
7. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
8. Place the processing data **matching** the location of the core and fiducials on the processing table.

For detailed information on project placement according to fiducial positions refer to chapter 1.1, Project placement, Matching the fiducial position.

9. Click on [Continue].

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

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

- The Bottom side of the core (*Layer3*) is being structured.

10. When the message *Processing phase: Flip material* is displayed, turn the core over around the symmetry axis of the system (the positioning holes should be located in the right front corner) and click on [OK].
11. Place the processing data **matching** the location of the core and fiducials on the processing table.
12. When project placement is complete, click on [Continue].

- The laser system reads the fiducials on the Top side of the core (*Layer2*) and starts structuring.

13. When the structuring of the core's top layer is finished and the message *Processing phase: Dismount material* is displayed, click on [Cancel] to temporarily stop the process.
14. Dismount the core from the system.
15. Spray the core with LPKF Cleaner and use a brush to clean the core, rinse with tap water and dry it with compressed air.

☑ The core has been structured.

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

For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.3, Multi-layer process.

1. Start the pre-heating process of the MultiPress S.
2. Prepare the materials.
3. Assemble the multi-layer stack in the press mold.
4. Press the multi-layer stack.
5. Cure the multi-layer stack.

After pressing of the multi-layer stack, it must **cure** at ambient temperature for at least **12 to 18 hours**.

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

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

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

1. Open the ***_PM*** file you previously generated with CircuitPro PM 2.3.
2. Load the tool magazine and assign the tools to positions.
3. In the *Processing* pane, select *MountMaterialTop* from the drop-down list and click on .
4. Perform the following phases:
   - *MountMaterialTop*
   - *MaterialSettings_3*
   - *Placement_4*

☐ The phases *ReadFiducialsTop, MarkingDrills* and *DrillingPlated* are performed.

5. When the message *Processing Phase: DismountMaterial_3* is displayed, click on [Cancel] to temporarily stop the process.
6. Plate the Multi-layer PCB.
7. Dismount the multi-layer PCB from the system, rinse with tap water and dry it using compressed air.

☑ The plated through-holes have been drilled.

For detailed information on ProtoMat phases refer to the CircuitPro PM 2.1 How-to guides, Part II, chapter 3.8.

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

1. Switch on the system.
2. Select a profile.
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. Swipe the activator from the multi-layer PCB.
9. Copper-plate the multi-layer PCB.

   The multi-layer PCB must remain in **tank 5 for 160 minutes** in order to achieve the **final copper thickness** of approximately **30 µm** on **all layers**. This way, the surfaces of the holes are sufficiently plated and the laser rubout process functions properly.
10. Switch off the system.

☑ The multi-layer PCB has been galvanically through-hole plated.

For detailed information on galvanic through-hole plating process refer to the Contac S4 User manual, chapter 6.3.

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

1. Measure the thickness of the multi-layer PCB with a caliper gauge.
2. Click on *File > Open or* on .
3. Select the ***_PL** file you previously saved in **CircuitPro PL 3.0**.
4. Click on [Open].

- The processing data are displayed in the *Layout* view.

5. Click on *Processing > Material Settings* or on .
6. In the *Material settings* dialog select the *Machine type, Material type* and enter the *Material thickness* (in this example the material type *FR4_1.55mm_Cu35 (laminated)* is used).
7. Click on [OK].
8. Switch to the *Processing* view.
9. In the *Workflow setup* pane expand the group *Processing*.
10. Right-click on *Bottom Side (Standard)* and select *Process from here*.
11. When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*BottomLayer*) **facing upwards** (the positioning holes should be located in the right rear corner) onto the processing table and click on [OK].
12. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
13. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
14. When project placement is complete, click on [Continue].

- The laser system reads the fiducials on the Bottom side (*BottomLayer*) and starts structuring.

15. When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system (the positioning holes should be located in the right front corner) and click on [OK].
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 [Continue].

- The laser system reads the fiducials on the Top side (*TopLayer*) and starts structuring.

18. When the message *Processing phase: Process drilling on a ProtoMat* is displayed, click on [OK].
19. Dismount 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 ***_PM*** file you previously generated with CircuitPro PM 2.3.
2. Place the multi-layer PCB onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** (the positioning holes should be located in the front right corner) and fasten it using adhesive tape.
3. Switch to the pane *Processing*.
4. Select *ReadFiducialsTop_1* from the drop-down list and click on .

☐ The phases *ReadFiducialsTop_1*, *DrillingUnplated* and *ContourRouting* are performed.

5. Dismount the multi-layer PCB from the system and break or cut the breakout tabs.
6. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean the multi-layer PCB, rinse under tap water and dry it with compressed air.

☑ The unplated through-holes have been drilled and the multi-layer PCB has been cut out.

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** circuit board using a UV 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

- Base plate FR4 18/18 µm, 305 (k) mm x 229 mm x 1 mm (order no. 119574)
- Thin Laminate 104 ML 0/5 µm, 305 (k) x 229 x 0.2 mm with protection foil (order no. 119571)
- Prepreg Type 2125, 275 mm (k) x 200 mm x 0.1 mm (order no. 119572)

Auxiliaries

- 1 set of cutting tools (drills/contour routers for LPKF ProtoMat)
- LPKF Cleaner (order no. 115891)
- Hot air oven (order no. 115877)
- Oil-free compressed air
- Tap water

System

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

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PM 2.3
- Preparing the core and laminate materials (with ProtoMat)
- Preparing the data in CircuitPro PL 3.0
- Structuring the core (with ProtoLaser)
- Assembling and pressing the multi-layer stack (with the 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 of through-holes (with ProConduct)

Preparing the data in CircuitPro PM 2.3

For detailed information on preparing data in CircuitPro PM 2.3 refer to chapter 2.2.

1. In the *Templates* tab of the *New document* dialog, select the template: *ProtoLaser – ProtoMat - 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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PM 2.3\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 should automatically be assigned to the correct layers and the following dialog is displayed:

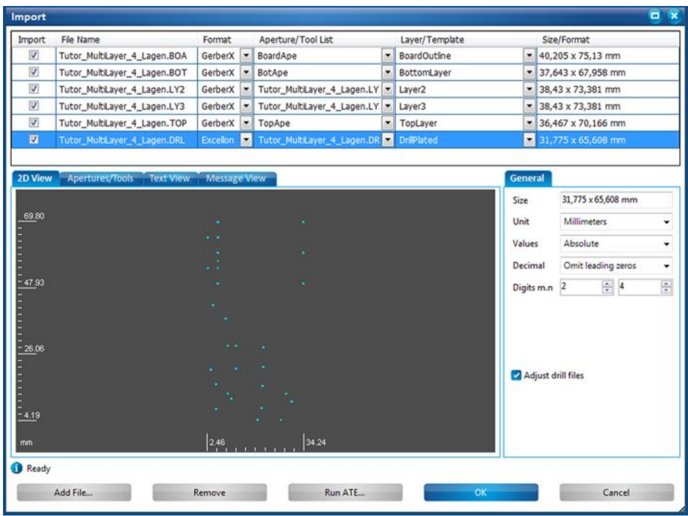


Fig. 88: Dialog *Import* and assigned layers

6. Assign the drill holes to the layer *DrillPlated* (in the *Layer/Template* column).
7. Click on [OK].

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

8. Select all holes that are not to be plated (i.e. that are to be assigned to the *DrillUnplated* layer) by clicking on them while pressing the `Ctrl` key.

☐ Selecting the holes changes their color.

9. Right-click on the layout and under *Assign objects to layer* select *DrillUnplated*.

☐ An additional layer is created automatically and the holes that are not to be plated are assigned to the *DrillUnplated* layer.

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

Usually it is necessary to move the layout within the multi-layer base material frame. It is preferable to move it **to the center** of the multi-layer base material frame.

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 .
15. Deactivate *Process* in the group *Insulate*.
16. Select *Edge gaps* in the group *Contour Routing*. This is usually the preferred method of contour routing.
17. Deactivate *Pockets* in the group *Convert to Toolpath*, since there are no “pockets” in the project for this tutorial.
18. Click on [Start].
- ☐ The *Computation results* are displayed in a dialog.
19. Click on [Close].
- ☐ The toolpaths have been calculated.
20. Save the file with the suffix _***PM***.

Delete all **toolpaths** and **auxiliary layers** before importing the data into CircuitPro PL 3.0. Only the initially imported layers should remain in your project.

21. Switch to the pane *Layers*.
- ☐ The view changes as follows:

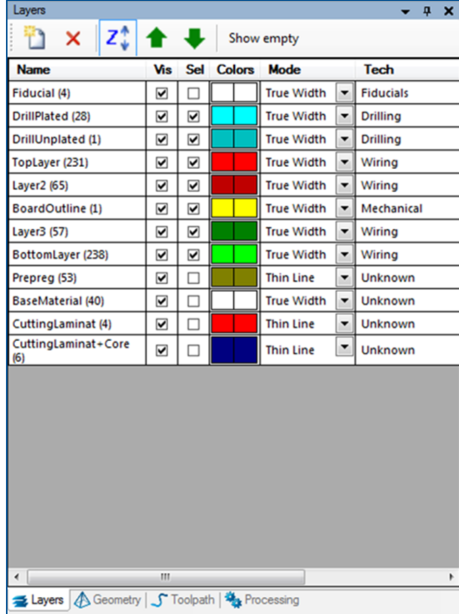


Fig. 89: Pane *Layers*

22. Activate all check boxes in the columns *Vis* and *Sel*.

☐ Select all **auxiliary layers** (including the *Fiducial* layer) by clicking on them while pressing the Ctrl key:

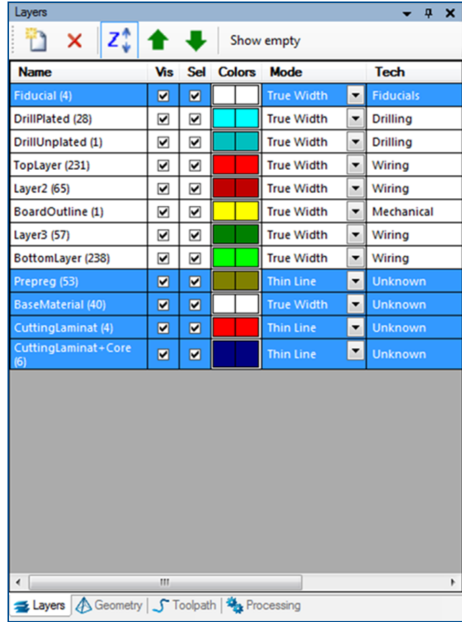


Fig. 90: Pane *Layers* and selected auxiliary layers

23. Delete the selected layers by pressing the key Del or clicking on .

☐ The selected layers have been deleted.

24. Switch to the pane *Toolpath* and expand the *Toolpaths* folder.

☐ A list of toolpaths is displayed.

25. Select all toolpaths in this folder by clicking on them while pressing the Ctrl key.

26. Delete them by pressing Del or clicking on .

☐ The selected toolpaths have been deleted.

For detailed information on deleting toolpaths in CircuitPro PM 2.3 refer to chapter 2.2.

27. Save the file with the suffix ***_PL***.

☑ The data have been prepared in CircuitPro PM 2.3.

Preparing the core and laminate materials (with ProtoMat)

1. Open the *_**PM** file you generated in the previous procedure.
2. Load the tool magazine and assign the tools to positions.
3. Click on *Machining* > *Process all* or on .
4. Perform the following phases (with the core material):
   - *MountMaterial*
   - *MaterialSettings*
   - *Placement*

☐ The phases *DrillFiducial* and *PrepareCore* are performed.

5. When the message *Processing Phase: FlipMaterial* is displayed, remove the core from the system and click on [Cancel] to temporarily stop the process.
6. Rinse the core with tap water and dry it with compressed air.
7. In the *Processing* pane, select *MountLaminate* from the drop-down list and click on .
8. Perform the following phases with the first laminate:
   - *MountLaminate*
   - *MaterialSettings_1*
   - *Placement_2*

☐ The *PrepareLaminateBottom* phase is performed.

9. When the message *Processing Phase: DismountMaterial_1* is displayed, remove the first laminate from the system and click on [OK].
10. Perform the following phases with the second laminate:
    - *MountLaminate_1*
    - *MaterialSettings_2*
    - *Placement_3*

☐ The *PrepareLaminateTop* phase is performed.

11. When the message *Processing Phase: DismountMaterial_2* is displayed, remove the second laminate from the system and click on [Cancel] to temporarily stop the process.

☑ The core and laminate materials have been prepared.

For detailed information on ProtoMat phases refer to the CircuitPro PM 2.1 How-to guides, Part II, chapter 3.8.

Preparing the data in CircuitPro PL 3.0

For detailed information on preparing data with CircuitPro PL 3.0 refer to chapter 2.1.

1. In the *Templates* tab of the *New document* dialog, select the template *PL-U4_PM_4Layer_ProConduct_MultiPressS.cp2d*.
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type, Material type* and enter the *Material thickness* (in this example, the material type *FR4_1.00mm_Cu18 (laminated)* is used).

A multi-layer PCB consists of different materials. For **each material** used during production, a **corresponding *Material type*** has to be selected.

4. Click on [OK].
5. Click on *File* > *Import old version*.
6. Select the ***_PL*** file you previously generated with CircuitPro PM 2.3.
7. Click on [Open].

☐ The ***_PL*** file has been imported.

8. In the *Workflow setup* pane, right-click on the group *Toolpaths* and click on the context menu item *Compute all…*
9. Check the *Computation Results* for possible warnings or errors and make corrections, if needed.
10. Click on [Close].
11. Save the file with the suffix ***_PL***.

☑ The data have been prepared in CircuitPro PL 3.0.

Structuring the core (with ProtoLaser)

1. Measure the thickness of the core with a caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing* view on .
3. After warm-up, the message *Processing phase: Process cutting on a ProtoMat* is displayed.
4. Click on [OK].
5. When the message *Processing phase: Mount core* is displayed, place the core with the **Bottom side** (*Layer3*) **facing upwards** (the positioning holes should be located in the right rear corner) in the center of the processing table.
6. Click on [OK].
7. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
8. Place the processing data **matching** the location of the core and fiducials on the processing table.

For detailed information on project placement according to fiducial positions refer to chapter 1.1, Project placement, Matching the fiducial position.

9. Click on [Continue].

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

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

☐ The Bottom side of the core (*Layer3*) is being structured.

10. When the message *Processing phase: Flip material* is displayed, turn the core over around the symmetry axis of the system (the positioning holes should be located in the front right corner) and click on [OK].
11. Place the processing data **matching** the location of the core and fiducials on the processing table.
12. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side of the core (*Layer2*) and starts structuring.

13. When the structuring of the core's Top side is finished and the message *Processing phase: Dismount material* is displayed, click on [Cancel] to temporarily stop the process.
14. Dismount the core from the system.
15. Spray the PCB with LPKF Cleaner and use a brush to clean it.
16. Rinse the PCB with tap water and dry it with compressed air.

☑ The core has been structured.

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

For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.3, Multi-layer process.

1. Start the pre-heating process of the MultiPress S.
2. Prepare the materials.
3. Assemble the multi-layer stack in the press mold.
4. Press the multi-layer stack.
5. Cure the multi-layer stack.

After pressing of the multi-layer stack, it must **cure** at ambient temperature for at least **12 to 18 hours**.

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

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

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

1. Measure the thickness of the multi-layer PCB with a caliper gauge.
2. Click on *File* > *Open* or on .
3. Select the ***_PL** file you previously saved in **CircuitPro PL 3.0.**
4. Click on [Open].

- Your layout is displayed in the *Layout* view.

5. Click on *Processing* > *Material Settings* or on .
6. In the *Material settings* dialog select the *Machine type, Material type* and enter the *Material thickness* (in this example, the material type *FR4_1.55mm_Cu18 (laminated)* is used).
7. Click on [OK].
8. Switch to the *Processing* view.
9. In the *Workflow setup* pane expand the group *Processing*.
10. Right-click on *Bottom Side (Standard)* and select *Process from here*.
11. When the message *Processing phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side** (*BottomLayer*) **facing upwards** (the positioning holes should be located in the right rear corner) onto the processing table and click on [OK].
12. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
13. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
14. When project placement is complete, click on [Continue].

- The laser system reads the fiducials on the Bottom side (*BottomLayer*) and starts structuring.

15. When the message *Processing phase: Flip material* is displayed, turn the multi-layer PCB over around the symmetry axis of the system (the positioning holes should be located in the right front corner) and click on [OK].
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 [Continue].

- The laser system reads the fiducials on the Top side (*TopLayer*) and starts structuring.

18. When the message *Processing phase: Process drilling on a ProtoMat* is displayed, click on [OK].
19. Dismount the multi-layer PCB from the system.
20. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean the multi-layer PCB, rinse 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 ***_PM*** file you previously generated with CircuitPro PM 2.3.
2. Place the multi-layer PCB onto the system's processing table with the **Top side** (*TopLayer*) **facing upwards** (the positioning holes should be located in the front right corner) and fasten it using adhesive tape.
3. Switch to the pane *Processing*.
4. Select *ReadFiducialsTop* from the drop-down list and click on .

- The phases *ReadFiducialsTop*, *MarkingDrills, DrillingUnplated* and *ContourRouting* are performed.

5. When the message *Processing Phase: DismountMaterial_3* is displayed, remove the multi-layer PCB from the system and click on [OK].
6. 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].
2. When the message *Processing Phase: MountMaterialTop* is displayed, place the multi-layer PCB with the **Top side** (*TopLayer*) **facing upwards** (the positioning holes should be located in the right front corner) onto the system's processing table and click on [OK].
3. Perform the *Placement_6* phase.

- The phases *ReadFiducialsTop_1* and *DrillingPlated* are performed.

4. When the message *Processing Phase: DismountMaterial_4* 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 film and expose them if necessary.

☑ The plated through-holes have been drilled into the multi-layer PCB.

Plating of 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 5-10 on the other side.
9. Slowly peel off the 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, rinse with tap water and dry it with compressed air.
13. 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 **ProConduct** Process description, chapter 3.

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** circuit board with blind vias and buried vias using a UV laser system, a circuit board plotter, a multi-layer press, and a galvanic through-hole plating system.

The production of a multi-layer PCB with blind vias and buried vias is supported only by the LPKF ProtoLaser U4 system.

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

Consumables

- Base plate 104 ML 18/18 µm, 305 (k) mm x 229 mm x 0.36 mm (order no. 119575)
- Thin Laminate 104 ML 0/5 µm, 305 (k) x 229 x 0.2 mm with protection foil (order no. 119571)
- Prepreg Type 2125, 275 mm (k) x 200 mm x 0.1 mm (order no. 119572)

Auxiliaries

- 1 set of cutting tools (drills/contour routers for LPKF ProtoMat)
- LPKF Cleaner (order no. 115891)
- Hot air oven (order no. 115877)
- Isopropyl alcohol
- Oil-free compressed air
- Tap water

System

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

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PM 2.3
- Drilling buried vias into the core (with ProtoMat)
- Preparing laminate materials (with ProtoMat)
- Galvanic through-hole plating of the core (with Contac S4)
- Preparing the data in CircuitPro PL 3.0
- Structuring the core (with ProtoLaser)
- Assembling and pressing the multi-layer stack (with the 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 of 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 1.4 before starting this tutorial.

Preparing the data in CircuitPro PM 2.3

For detailed information on preparing data with CircuitPro PM 2.3 refer to chapter 1.2.

1. In the *Templates* tab of the *New document* dialog, select the template: *PL_PM_4Layer_GalvanicTHP_MultiPressS_Blind-Buried vias.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 certain which files to select presently, then select all of them. You will see the preview of the files later and can deselect those you do not need.

- The following dialog is displayed:

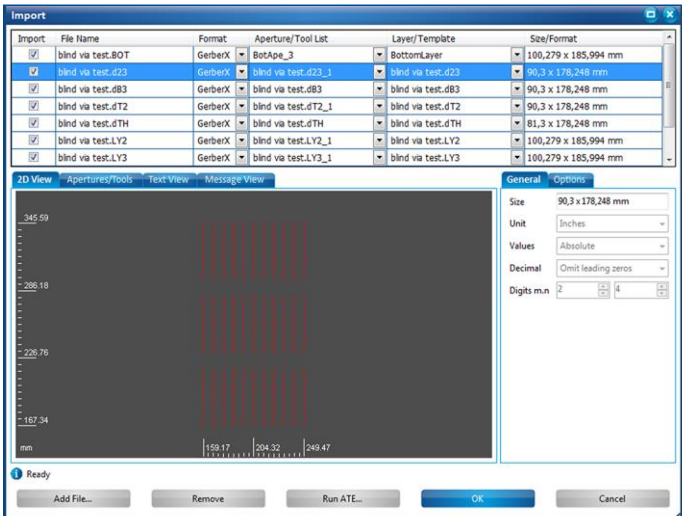


Fig. 91: Dialog *Import* and assigned layers

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 *Import* dialog).
- Deactivate *Use layer name*.
- Activate *Apply to all Gerber files*.

6. Click on [OK].

☐ The *CAM view* changes as follows:

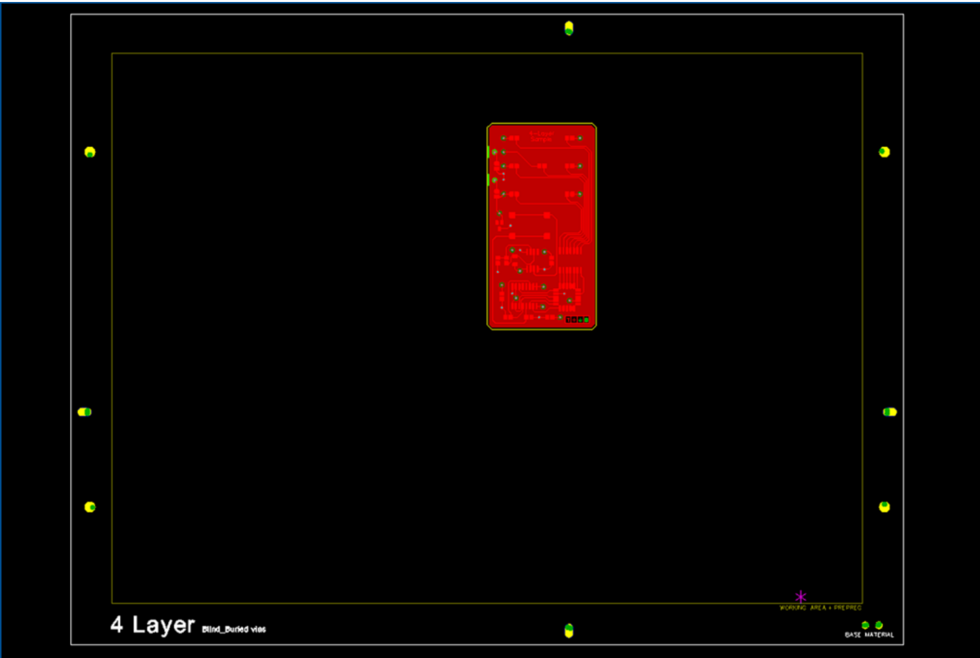


Fig. 92: Imported data in the *CAM view*

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. Using the left mouse button, drag and drop the layouts to the desired location.
11. In the *CAM view*, click anywhere on the black area in order to disable the *Move selected objects* function.

☐ The layout has been moved.

12. Click on *Toolpath* > *Technology Dialog* or on .
13. Click on [Start].

☐ The *Computation results* are displayed in a dialog.

14. Click on [Close].

☐ The toolpaths have been calculated.

Creating toolpaths for **buried vias** have to be done **separately**.

15. Click on *Toolpath* > *Technology Dialog* or on .
16. Deactivate every check box in the dialog except *Drills*.

- ☐ The *Technology dialog* changes as follows:

Fig. 93: ***Technology Dialog* in the process of creating toolpaths for buried vias**

17. Click on [Show Details].

- ☐ The details are displayed:

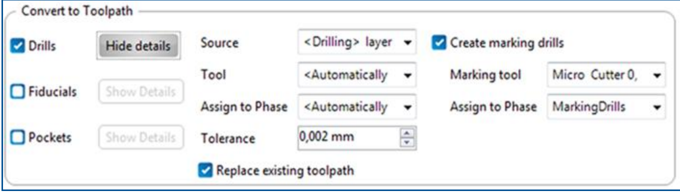


Fig. 94: Dialog *Convert to Toolpath*

18. Under *Source* select *Layer "Buried via (L2-L3)"* from the drop-down list.
19. Under *Assign to Phase* select *DrillBuriedVias* from the drop-down list.
20. Deactivate the option *Replace existing toolpath*.
21. Under the second *Assign to Phase* select *DrillBuriedVias* from the drop-down list.

- ☐ After your modifications, the following settings are displayed:

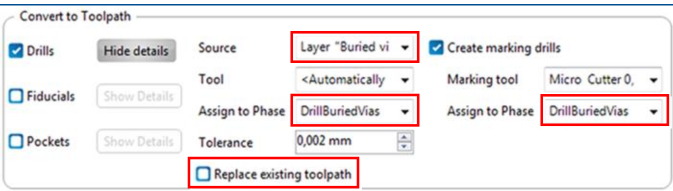


Fig. 95: Settings for buried vias

22. Click on [Start].

- ☐ The *Computation results* are displayed in a dialog.

23. Click on [Close].

- ☐ The toolpaths for buried vias have been calculated.

24. Switch to the pane *Toolpath*.

☐ The view changes as follows:

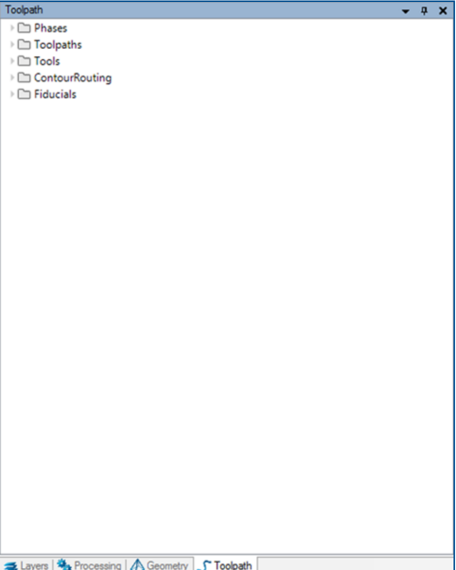


Fig. 96: Pane *Toolpath*

25. Double-click on the folder *Phases*.

☐ The view changes as follows:

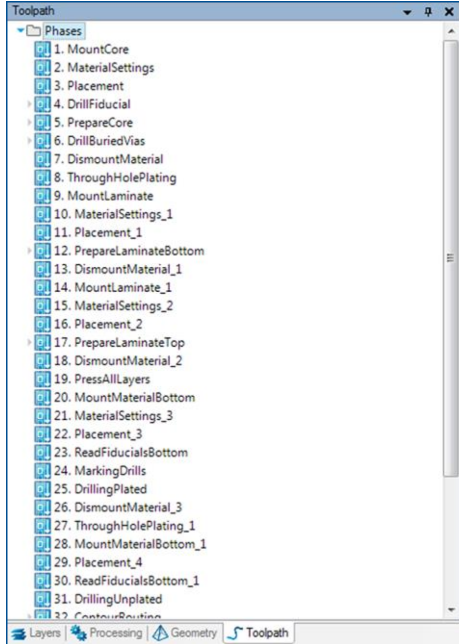


Fig. 97: Pane *Toolpath* and list of nodes

26. Expand the phase *6. DrillBuriedVias*.

- A list of toolpaths is displayed:

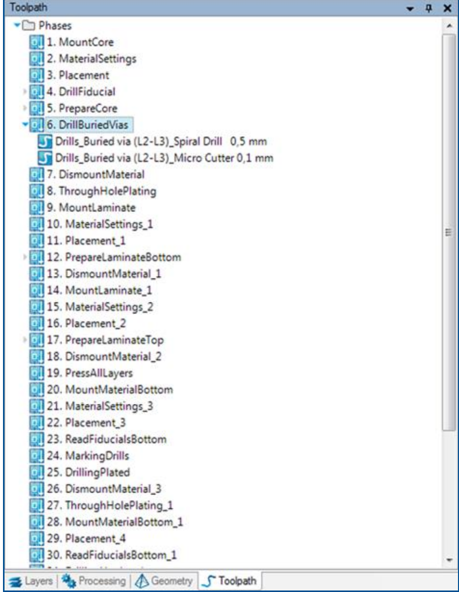


Fig. 98: List of toolpaths

The Micro Cutter toolpath must always be located in the first position. Should it not be (as shown in this example), move it to the correct position.

- The phase *6. DrillBuriedVias* is displayed as follows:

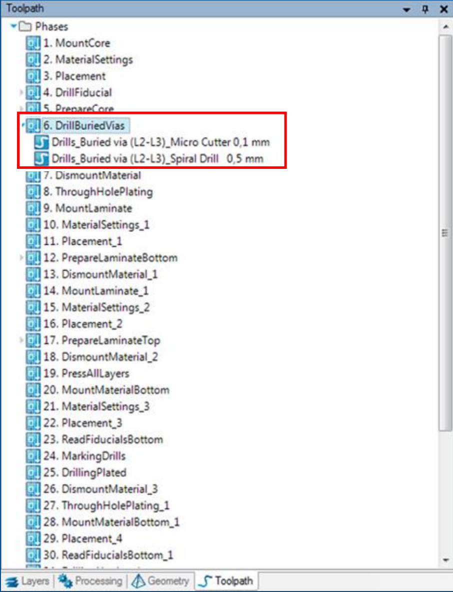


Fig. 99: New sequence of toolpaths

27. Save the file with the suffix _***PM***.

Delete all **toolpaths** and **auxiliary layers** before importing the data into CircuitPro PL 3.0. Only the initially imported layers should remain in your project.

28. Switch to the pane *Layers*.

☐ The view changes as follows:

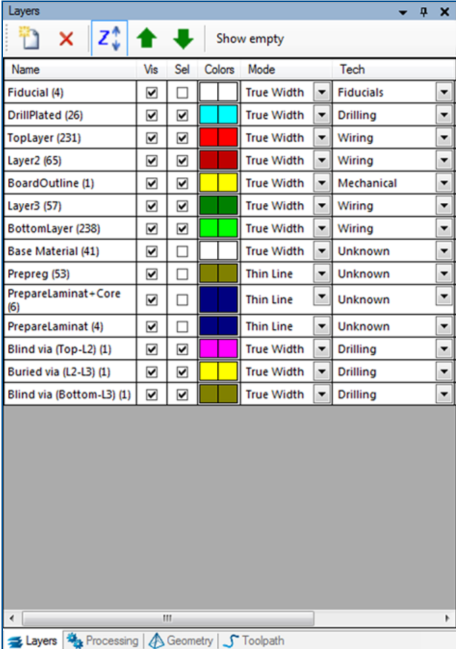


Fig. 100: Pane *Layers*

29. Activate all check boxes in the columns *Vis* and *Sel*.
30. Select all **auxiliary layers** (including *Fiducial* layer) by clicking on them while pressing the `Ctrl` key:

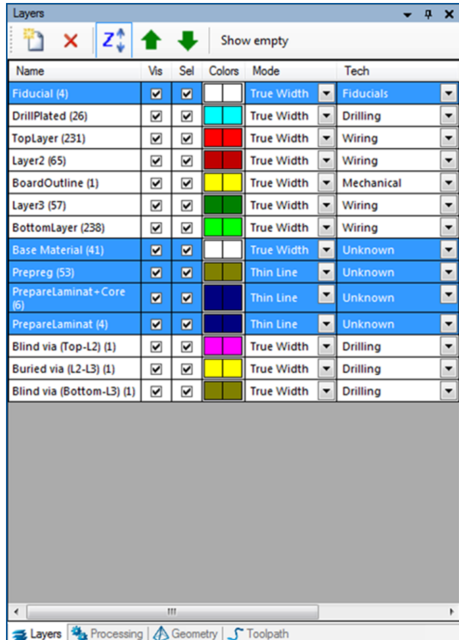


Fig. 101: Pane *Layers* and highlighted auxiliary layers.

31. Delete the selected layers by pressing the key `Del` or clicking on ❌.

☐ The selected layers have been deleted.

32. Switch to the pane *Toolpath* and expand the *Toolpaths* folder.

☐ The view changes as follows:

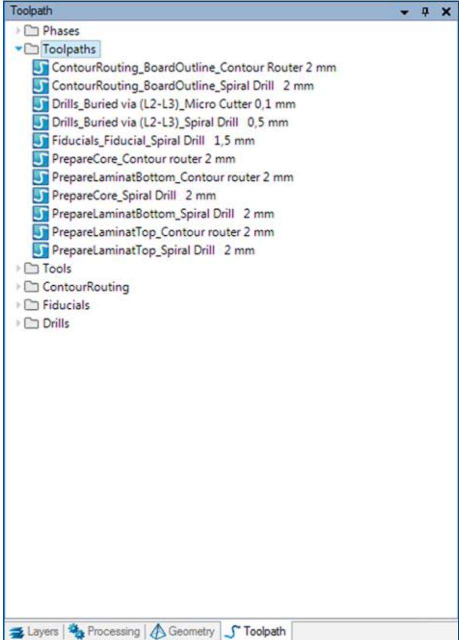


Fig. 102: List of toolpaths

33. Select all toolpaths in this folder by clicking on them while pressing the `Ctrl` key:

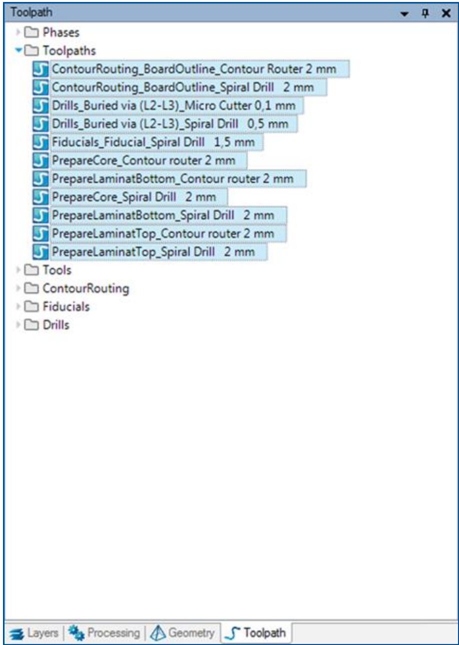


Fig. 103: Highlighted toolpaths

34. Delete them by pressing the key `Del` or clicking on ❌.

☐ The selected toolpaths have been deleted.

35. Save the file with the suffix _***PL***.

☑ The data have been prepared in CircuitPro PM 2.3.

Drilling buried vias into the core (with ProtoMat)

1. Open the *_**PM** file you generated in the previous procedure.
2. Load the tool magazine and assign the tools to positions.
3. Click on *Machining* > *Process all* or on ▶.
4. Perform the following phases:
   - *MountCore*
   - *MaterialSettings*
   - *Placement*

☐ The phases *DrillFiducial*, *PrepareCore* and *DrillBuriedVias* are performed.

5. When the message *Processing Phase: DismountMaterial* is displayed, remove the core from the system and click on [OK].
6. Rinse the core with tap water and dry it using compressed air.
7. When the message *Processing Phase: ThroughHolePlating* is displayed, **proceed** to **galvanic through-hole plating** of the core material (described in the following procedure).

While galvanic plating is in progress, **proceed** to **laminate materials preparation**. In the following steps positioning and pin holes are to be drilled.

☑ The buried vias have been drilled into the core.

For detailed information on drilling with the ProtoMat refer to the CircuitPro 2.1 How-to guides, Part II, chapter 3.8.

Preparing laminate materials (with ProtoMat)

1. Click on [OK] to continue the process.
2. Perform the following phases:
   - *MountLaminate*
   - *MaterialSettings_1*
   - *Placement_1*

☐ The *PrepareLaminateBottom* phase is performed.

3. When the message *Processing Phase: DismountMaterial_1* is displayed, remove the laminate from the system and click on [OK].
4. Perform the following phases:
   - *MountLaminate_1*
   - *MaterialSettings_2*
   - *Placement_2*

☐ The *PrepareLaminateTop* phase is performed.

5. When the message *Processing Phase: DismountMaterial_2* is displayed, remove the laminate from the system and click on [Cancel] to temporarily stop the process.

☑ The laminate materials have been prepared.

For detailed information on ProtoMat phases refer to the CircuitPro 2.1 How-to guides, Part II, chapter 3.8.

Galvanic through-hole plating of the core (with Contac S4)

During and after galvanic plating, **handle the core with care**. The core 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.
3. Start the process.
4. Prepare the core for through-hole plating.
5. Clean the core.
6. Condition the core.
7. Activate the core.
8. Swipe the activator from the core.
9. Copper-plate the core.

   The core must remain in **tank 5 for 100 minutes** in order to achieve the **final copper thickness** of approximately **30 µm** on **all layers**. This way, the surfaces of the holes are sufficiently plated and the laser rubout process functions properly.
10. Switch off the system.

☑ The core has been galvanically through-hole plated.

For detailed information on galvanic through-hole plating process refer to the Contac S4 User manual, chapter 6.3.

Tips for more efficient galvanic through-hole plating

- Remove the soot around the laser drilled holes using a cloth and isopropyl alcohol.
- In case of oxidation, clean it using a soft brush and LPKF Cleaner.
- Use compressed air to remove drilling debris from the holes.
- Use compressed air to remove the water from the 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 3.0

For detailed information on preparing data with CircuitPro PL 3.0 refer to chapter 2.1.

1. In the *Templates* tab of the *New document* dialog, select the template *PL-U4_PM_4Layer_GalvanicTHP_MultiPressS_Blind-Buried vias.cp2d*.
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type, Material type* and enter the *Material thickness* (in this example the material type *4Layer_Blind_buried vias (laminated)* is used).

Use only the *Material type 4Layer_Blind_buried vias (laminated)* (which selects the corresponding tool library*)* with the template selected. Selecting any other *Material type* may cause a wrong tool assignment.

4. Click on [OK].
5. Click on *File > Import old version*.
6. Select the ***_PL*** file you previously generated with CircuitPro PM 2.3.
7. Click on [Open].

☐ The ***_PL*** file has been imported.

8. In the *Workflow setup* pane, right-click on the group *Toolpaths* and click on the context menu item *Compute all…*
9. Check the *Computation Results* for any possible warnings or errors and make corrections, if needed.
10. Click on [Close].
11. Save the file with the suffix ***_PL***.

☑ The data have been prepared in CircuitPro PL 3.0.

Structuring the core (with ProtoLaser)

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

1. Measure the **thickness** of the core with a caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing* view on .

☐ After warm-up the following message is displayed:

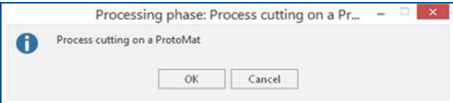


Fig. 104: Message *Processing phase: Process cutting on a ProtoMat*

3. Click on [OK].
4. When the message *Processing phase: Process drilling on a ProtoMat* is displayed, click on [OK].
5. When the message *Processing phase: Galvanic through-hole plating* is displayed, click on [OK].

☐ The following message is displayed:

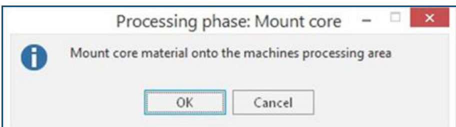


Fig. 105: Message *Processing phase: Mount core*

6. Place the galvanically plated core with the **Bottom side facing upwards** onto the processing table.

Two 3 mm positioning holes help you with the correct orientation of the core. When the **Bottom side** of the core is facing **upwards**, the positioning **holes** should be located in the **right rear corner** of the core.

☐ The following figure displays the positioning holes:

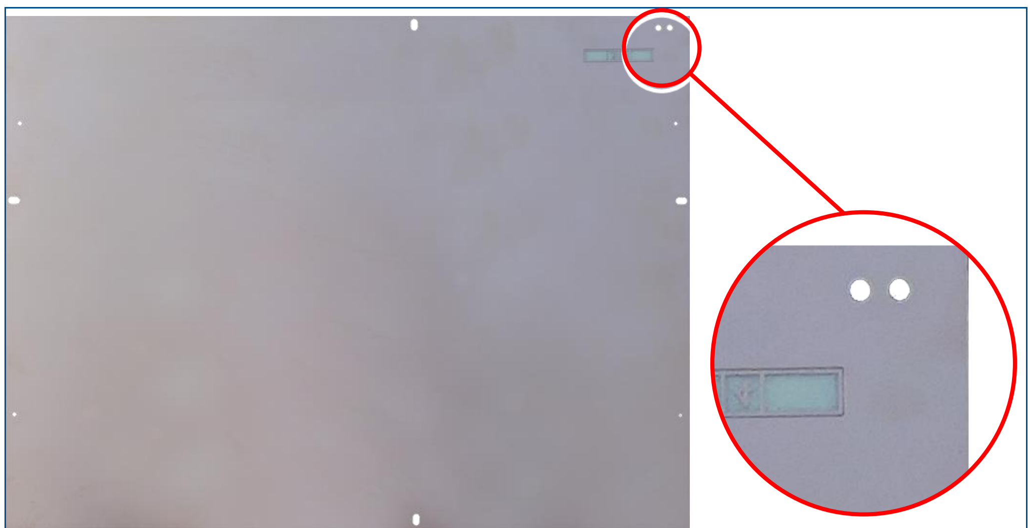


Fig. 106: Positioning holes in the right rear corner of the core

7. Click on [OK].
8. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
9. Place the processing data **matching** the location of the core and fiducials on the processing table.

For detailed information on project placement according to fiducial positions refer to chapter1.1, Project placement, Matching the fiducial position.

10. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the core (*Layer3*).

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

☐ The Bottom side of the core (*Layer3*) is being structured.

11. When the message *Processing Phase: Flip material* is displayed, turn the core over around the symmetry axis of the system and click on [OK].
12. Place the processing data **matching** the location of the core and fiducials on the processing table.
13. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side of the core (*Layer2*).

Structuring starts.

14. When the structuring of the top side is finished and the message *Processing phase: Dismount material* is displayed, click on [Cancel] to temporarily stop the process.
15. Dismount the core from the system.
16. Spray the PCB with LPKF Cleaner and use a brush to clean it.
17. Rinse the PCB with tap water and dry it with compressed air.

☑ The core has been structured.

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

For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.3, Multi-layer process.

1. Start the pre-heating process of the MultiPress S.
2. Prepare the materials.
3. Assemble the multi-layer stack in the press mold.
4. Press the multi-layer stack.
5. Cure the multi-layer stack.

Heat treatment of the multi-layer stack after the pressing cycle is **necessary** for **completing the curing process**. Heat the multi-layer stack in an oven for 50 minutes at 100 °C in order to fully cure the epoxy resin.

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

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

1. Open the ***_PM** file you previously generated with CircuitPro PM 2.3.
2. Load the tool magazine and assign the tools to positions.
3. In the pane *Processing*, select *MountMaterialBottom* from the drop-down list and click on .
4. Perform the following phases:
   - *MountMaterialBottom*
   - *MaterialSettings_2*
   - *Placement_3*

☐ The phases *ReadFiducialsBottom, MarkingDrills* and *DrillingPlated* are performed.

5. When the message *Processing Phase: DismountMaterial_3* is displayed, click on [Cancel] to temporarily stop the process.
6. Dismount the multi-layer PCB from the system.
7. Rinse the multi-layer PCB with tap water and dry it using compressed air.

☑ The plated through-holes have been drilled into the multi-layer PCB.

For detailed information on ProtoMat phases refer to the CircuitPro 2.1 How-to guides, Part II, chapter 3.8.

Drilling blind vias into the multi-layer PCB (with ProtoLaser)

1. Click on *File > Open or* on .
2. Select the ***_PL** file you previously saved in **CircuitPro PL 3.0**.
3. Click on [Open].

☐ The processing data are displayed in the *Layout* view.

4. Switch to the *Processing* view.

☐ The group *Processing* in the *Workflow setup* pane is expanded and the list of processing phases is displayed:

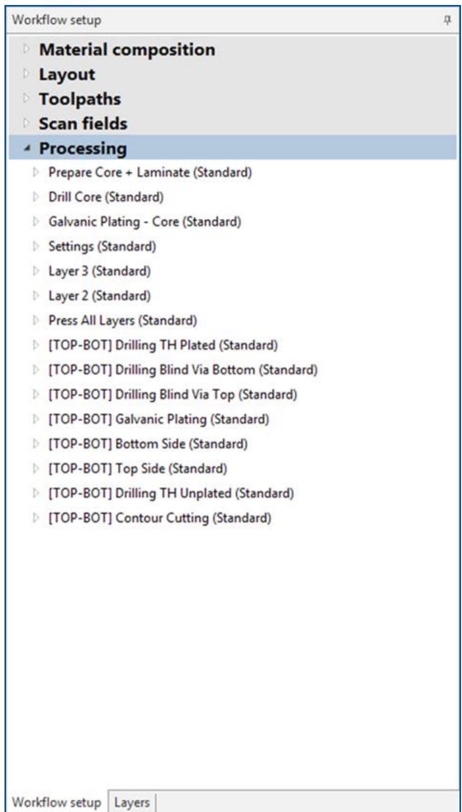


Fig. 107: List of processing phases

5. Right-click on *[TOP-BOT] Drilling Blind Via Bottom (Standard)*.

☐ The following context menu is displayed:

Fig. 108: Context menu *Processing*

6. Select *Process from here*.
7. When the message *Processing Phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side facing upwards** (the positioning holes should be located in the right rear corner) onto the processing table and click on [OK].
8. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
9. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
10. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Bottom side (*BottomLayer*) and the blind vias are being drilled.

11. 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].
12. Place the processing data matching the location of the multi-layer PCB and fiducials on the processing table.
13. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side (*TopLayer*) and the blind vias are being drilled.

14. When drilling the blind vias on the Top side (*TopLayer*) is finished and the message *Processing phase: Dismount material* is displayed, click on [Cancel] to temporarily stop the process.
15. Dismount the multi-layer PCB from the system.
16. Remove the soot around the laser drilled holes using a cloth and isopropyl alcohol.

☑ The blind vias have been drilled into the multi-layer PCB.

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

1. Switch on the system.
2. Select a profile.
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. Swipe the activator from the multi-layer PCB.
9. Copper-plate the multi-layer PCB.

   The multi-layer PCB must remain in **tank 5 for 160 minutes** in order to achieve the **final copper thickness** of approximately **30 µm** on **all layers**. This way, the surfaces of the holes are sufficiently plated and the laser rubout process functions properly.
10. Switch off the system.

☑ The multi-layer PCB has been galvanically through-hole plated.

For detailed information on galvanic through-hole plating process refer to the Contac S4 User manual, chapter 6.3.

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

1. Measure the thickness of the multi-layer PCB with a caliper gauge.
2. Open the ***_PL*** file you previously saved in **CircuitPro PL 3.0**.
3. Switch to the *Processing* view.
4. In the *Workflow setup* pane expand the group *Processing*.
5. Right-click on *[TOP-BOT] Bottom Side (Standard)* and select *Process from here*.
6. When the message *Processing Phase: Mount material bottom* is displayed, place the multi-layer PCB with the **Bottom side facing upwards** (the positioning holes should be located in the right rear corner) onto the processing table and click on [OK].
7. Place the processing data **matching** the location of the multi-layer PCB and fiducials on the processing table.
8. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Bottom side (*BottomLayer*) and starts structuring.

9. 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].
10. Place the processing data matching the location of the multi-layer PCB and fiducials on the processing table.
11. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side (*TopLayer*) and starts structuring.

12. After structuring of the Top side is finished, dismount the multi-layer PCB.

☑ 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 ***_PM** file you previously generated with CircuitPro PM 2.3.
2. Place the board onto the system's processing table with the **Bottom side facing upwards** (the positioning holes should be located in the right rear corner) and fasten it using adhesive tape.
3. Switch to the pane *Processing*.
4. Select *MountMaterialBottom_1* from the drop-down list and click on .
5. Perform the *Placement_4* phase.

☐ The phases *ReadFiducialsBottom_1*, *DrillingUnplated* and *ContourRouting* are performed.

6. Dismount the multi-layer PCB from the system and break or cut the breakout tabs.
7. Spray the multi-layer PCB with LPKF Cleaner and use a brush to clean the multi-layer PCB, rinse under 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 CircuitPro PM 2.1 How-to guides, Part II, chapter 3.8.

The multi-layer PCB production is finished.

Other processes

4 Other how-to examples

This chapter contains other how-to examples of different difficulty levels. To create a polyimid stencil as well as to structure a solder mask, basic knowledge of handling UV laser systems and the system software is sufficient. To create a flex-rigid PCB, advanced knowledge of multi-layer PCBs is required.

4.1 Producing a polyimide stencil

This chapter describes how to produce a polyimide stencil using a UV laser system.

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

Consumables	System
• Polyimide Foil A4, 125 µm, format for Stencils (order no. 108321)	• ProtoLaser U4/S4

The following steps are performed in this tutorial:

- Preparing the data in CircuitPro PL 3.0
- Cutting the stencil (with ProtoLaser)

Preparing the data in CircuitPro PL 3.0

For detailed information on preparing data with CircuitPro PL 3.0 refer to chapter 2.1.

1. In the *Templates* tab of the *New document* dialog, select the template *Stencil QR 266x380.cp2d*.
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type, Material type* and enter the *Material thickness* (For this procedure the material type *Polyimide_0.125mm (non-laminated)* was used).
4. Click on [OK].
5. Click on *File* > *Import* or on .
6. 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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PL 3.0\Example Data\UseCase_PolyimideStencil`.
7. Select the files you want to import (in this example *Tutor.SPT*).
8. Click on [Open].

☐ The data is automatically assigned to the correct layer.

Should the data not be assigned to the correct layer automatically, the dialog *Import* is displayed. In the column *Layer/Template* select the layer *SolderPasteTop* from the drop-down list and click on [OK].

☐ The *Layout* view changes as follows:

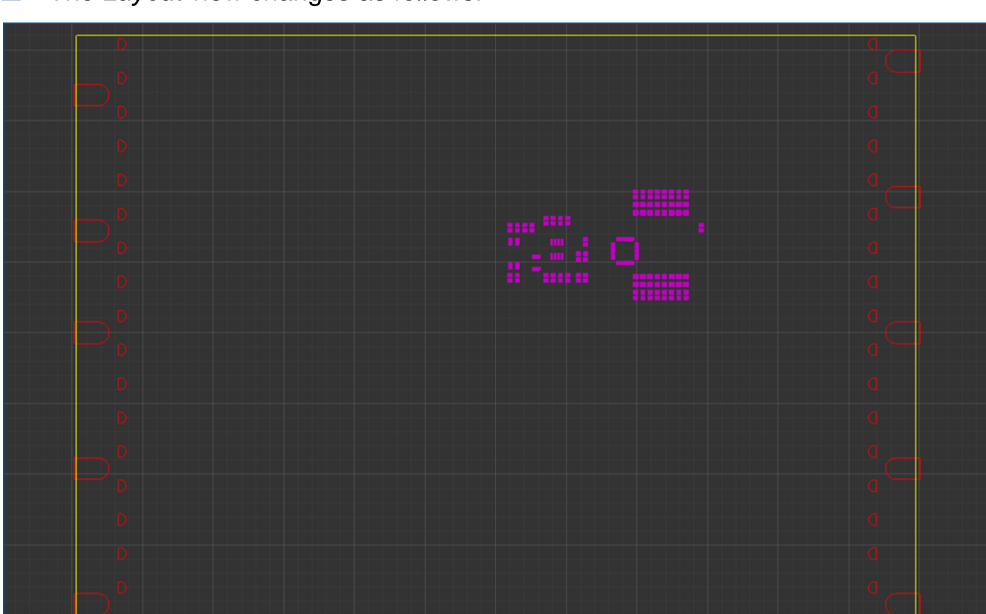


Fig. 109: *Layout* view of the imported data

9. Select the entire layout.

☐ The layout is highlighted and changes its color.

10. Perform one of the following steps:
    – Right-click on the highlighted layout and select *Move object* in the context menu;
    – Click on .

11. Click on a point **in the layout** to set the **reference point**.
12. Click on a point **in the stencil material** to set the **target point**.
13. In the *Layout* view, click anywhere on the black area to disable the *Move object* function.

☐ The layout has been moved.

14. In the *Workflow setup* pane, right-click on the group *Toolpaths* and click on the context menu item *Compute all…*
15. Check the *Computation Results* for any possible warnings or errors and make corrections, if needed.
16. Click on [Close].

☑ The data have been prepared in CircuitPro PL 3.0.

Cutting the stencil (with ProtoLaser)

1. Measure the thickness of the polyimide foil with caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing* view on ▶.

☐ After warm-up, the message *Processing phase: Mount material* is displayed.

3. Place the polyimide foil in the center of the processing table.
4. In the *Processing* view, click on .
5. Click on [OK].
6. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
7. Place the processing data **matching** the polyimide foil position and size to the processing area used by the CircuitPro PL software.

For detailed information on project placement according to polyimide foil position and size refer to chapter 1.1, Project Placement, Determining the processing area.

8. When project placement is complete, click on [Set center].

☐ The layout data are now precisely aligned and positioned in the center of the polyimide foil.

9. Click on [Continue].

☐ The laser system cuts the stencil.

10. When the message *Board production finished* appears, remove the polyimide stencil from the system.
11. Check whether the cut stencil can be fully removed from the residual polyimide foil.
12. Carefully detach the stencil from the residual polyimide foil.
13. 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 the production of a flex-rigid PCB. A flex-rigid PCB merges the properties of rigid and flexible circuit boards and consists of three different materials.

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

Consumables

- Base material FR4 18/18 µm, 305 (k) mm x 229 mm x 1 mm (order no. 119574)
- Prepreg Type 2125, 275 mm (k) x 200 mm x 0.1 mm (order no. 119572)
- Single-sided PyraluxTK, 0.12 mm, 12 Cu, 229 mm x 305 mm

Auxiliaries

- LPKF Cleaner (order no. 115891)
- Spray adhesive
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4
- MultiPress S

The following steps are performed in this tutorial:

- Preparing the data for the rigid material in CircuitPro PL 3.0
- Preparing the rigid material
- Preparing the data for the prepreg material in CircuitPro PL 3.0
- Preparing the prepreg material
- Preparing the data for the flexible material in CircuitPro PL 3.0
- Preparing the flexible material
- Assembling and pressing the flex-rigid stack (with the MultiPress S)
- Structuring the flexible part of the flex-rigid PCB
- Cutting out the flex-rigid PCB

In the CircuitPro PL 3.0 software, processing is divided into phases, which are further divided into work packages. Work packages can contain tasks.

The following figure displays the *Workflow setup* pane with the expanded group *Processing*:

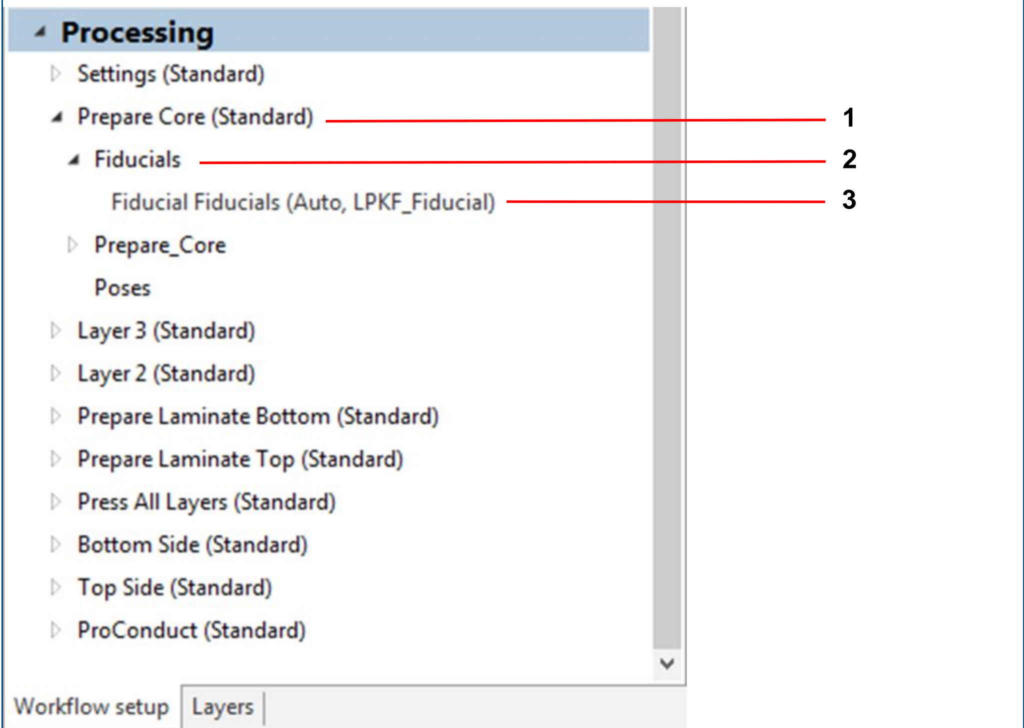


Fig. 110: Examples of a phase, work package and task

1 Phase

2 Work package

3 Task

A **phase** (e.g. *Prepare Core (Standard)*) consists of one or several **work packages** (e.g. *Fiducials*) that can contain one or several **tasks** (e.g. *Fiducial Fiducials (Auto, LPKF Fiducial)*).

Get familiar with these three terms, since they are used in the following example.

Preparing the data for the rigid material

For detailed information on preparing the data with CircuitPro PL 3.0 refer to chapter 2.1.

1. In the *Templates* tab of the *New document* dialog, select the template *PL-U4_4Layer_ProConduct_MultiPressS.cp2d*.

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 **pin holes** for the MultiPress S, since creating a flex-rigid PCB requires bonding of rigid and flexible materials. The pin holes ensure that the different physical layers can be aligned for bonding using the registration system of the MultiPress S.
The selected template contains some production phases and layers that are not necessary for this example and will therefore be ignored.

2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type* and *Material type* and enter the *Material thickness* (for this example the material type *FR4_1.00mm_Cu18 (laminated)* is used).

A flex-rigid PCB consists of different materials. For **each material** used during production, a **corresponding *Material type*** has to be selected.

4. Click on [OK].
5. Click on *File* > *Import* or on .
6. 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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PL 3.0\Example Data\UseCase_FlexRigidPCB`
7. Select the files you want to import (in this example *Tutor_Starrflex.BOA*, *Tutor_ Starrflex.BOT* and *Tutor_ Starrflex.CUI*).
8. Click on [Open].

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

Fig. 111: Dialog *Import* and assigned layers

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

9. Click on [OK].

- The following warning is displayed:

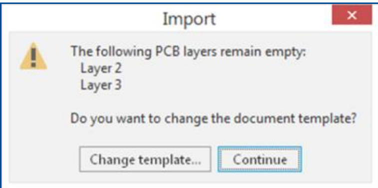


Fig. 112: Warning *Import*

10. Click on [Continue].

- The *Layout* view changes as follows:

Fig. 113: *Layout* view of the imported data

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

During the production of a flex-rigid PCB it is important to position the prepreg material accurately when assembling the materials in the press mold. Hence, note the exact positioning when stacking the materials in the press mold.

The pin holes in the selected template are located outside the prepreg's area. Therefore, **three prepreg positioning holes** with a **1 mm radius** have to be created in the layout. These positioning holes will help with positioning the prepreg material accurately in the press mold.

- Select the layer *DrillUnplated (0)* from the *Layers* drop-down list:

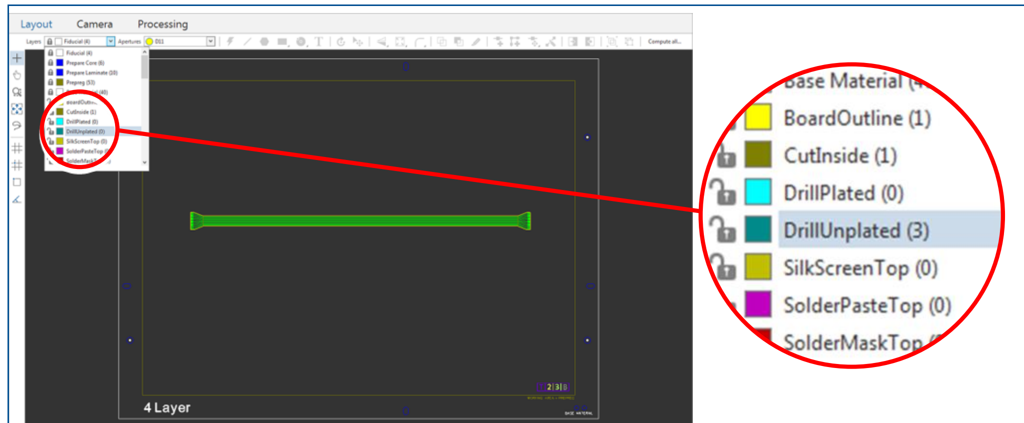


Fig. 114: Drop-down list layers

12. Perform one of the following steps:
    - Click on .
    - Click on *Insert* > *Circle or circle path…* > *Circle by radius*.
    - Right-click on the black area and select *Circle by radius* from the context menu.

- ☐ The input fields for specifying the center point are displayed:

Fig. 115: Specifying the center point

The three prepreg 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.

13. Click on a desired point in the black area to specify the center point of the first prepreg positioning hole.

- ☐ The center point of the prepreg positioning hole has been specified and an input field for specifying the radius is displayed:

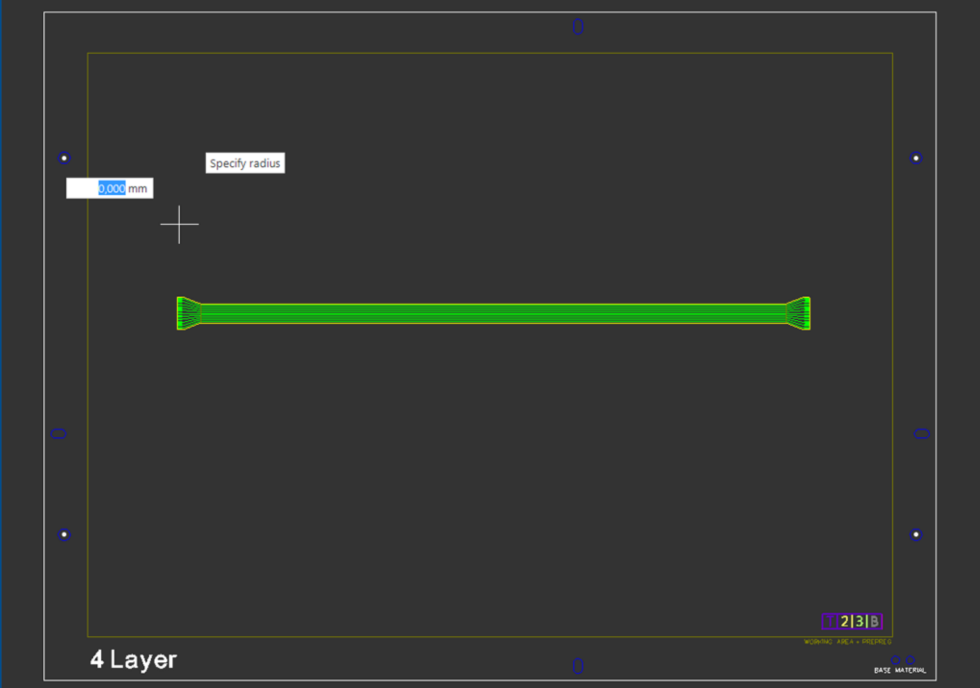


Fig. 116: Specifying the radius

14. Enter the radius in the input field (in this case **1 mm**) and press the `Enter` key.

- ☐ The first prepreg positioning hole is created in the layout. The input fields for specifying the center point of the next prepreg positioning hole are displayed:

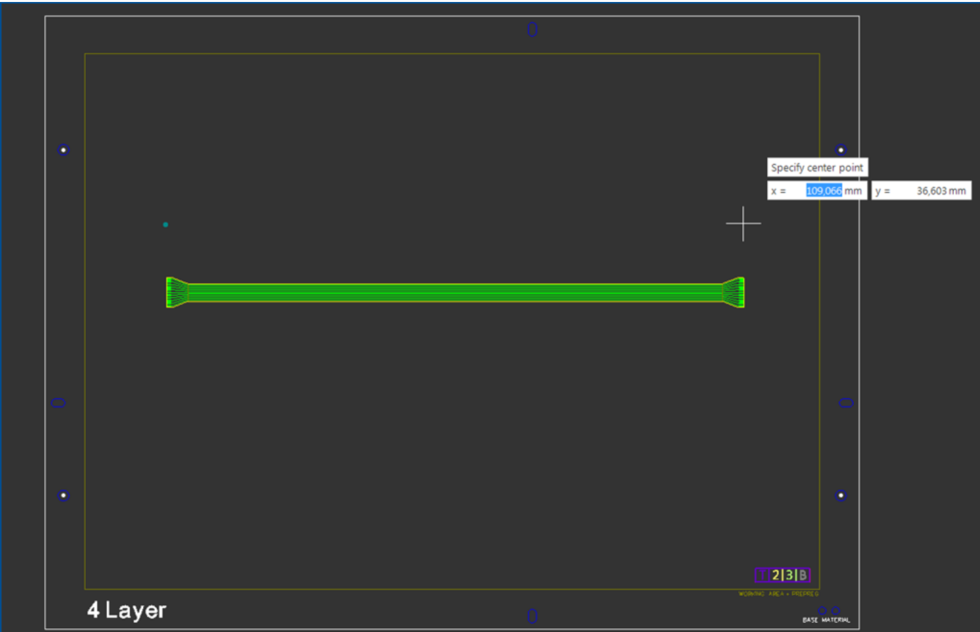


Fig. 117: Specifying the next center point

15. Repeat steps 13 and 14 for creating another two prepreg positioning holes.
16. Press the `Esc` key to exit the *Circle by radius* function.

- ☐ The three prepreg positioning holes have been created on the *DrillUnplated* layer:

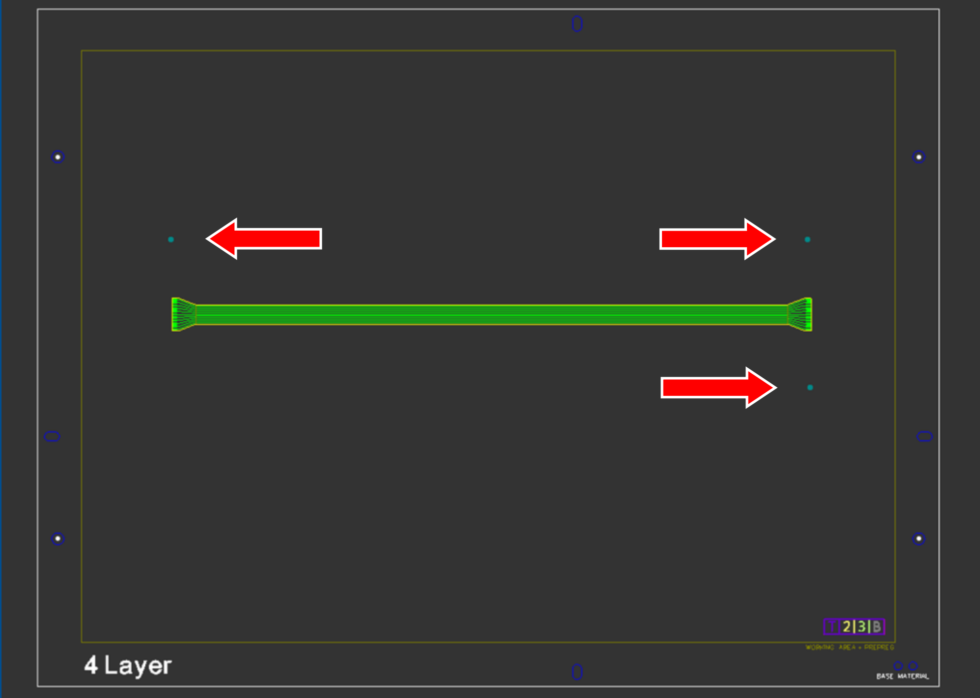


Fig. 118: Prepreg positioning holes

17. In the *Workflow setup* pane, expand the group *Processing* and the phase *Top Side (Standard)*.

☐ A list of work packages is displayed:

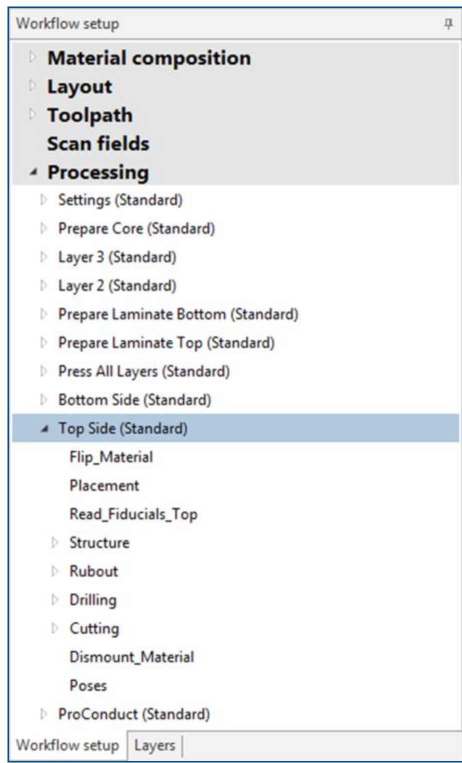


Fig. 119: Phase *Top Side (Standard)*

18. Expand the work packages *Drilling* and *Cutting*.

- The view changes as follows:

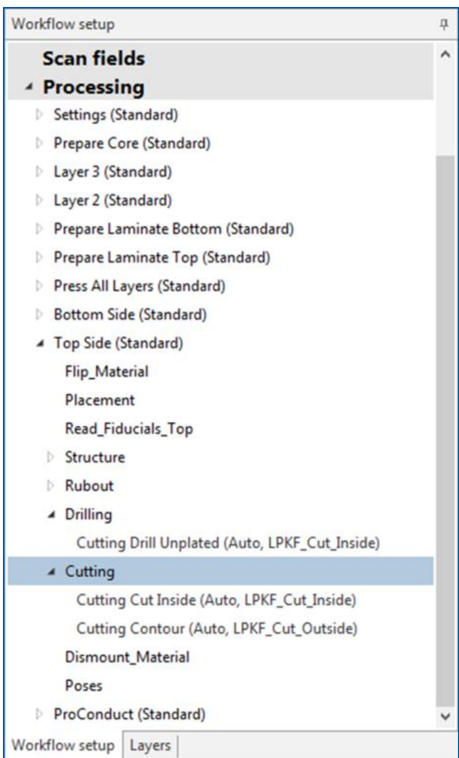


Fig. 120: Work packages *Drilling* and Cutting

19. Select the tasks *Cutting Drill Unplated (Auto, LPKF_Cut_Inside)* and *Cutting Cut Inside (Auto, LPKF_Cut_Inside)* by clicking on them while pressing the `Ctrl` key.

- The view changes as follows:

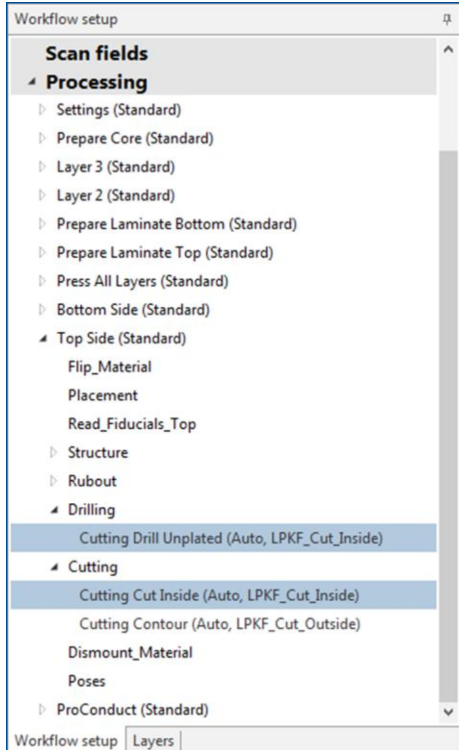


Fig. 121: Tasks selected

20. Delete the tasks by pressing the key Del or clicking on ✕.
- ☐ The selected tasks have been deleted.
21. Expand the group *Toolpath* and right-click on the node *Cut Inside*.
- ☐ The following context menu is displayed:

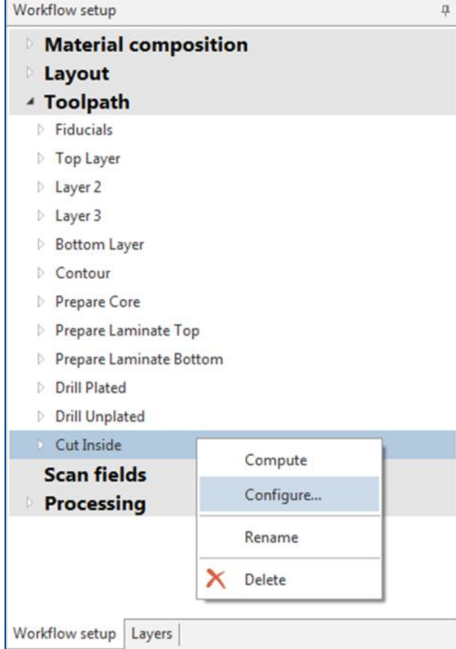


Fig. 122: Context menu *Configure…*

22. Click on the context menu item *Configure…*

☐ The following dialog is displayed:

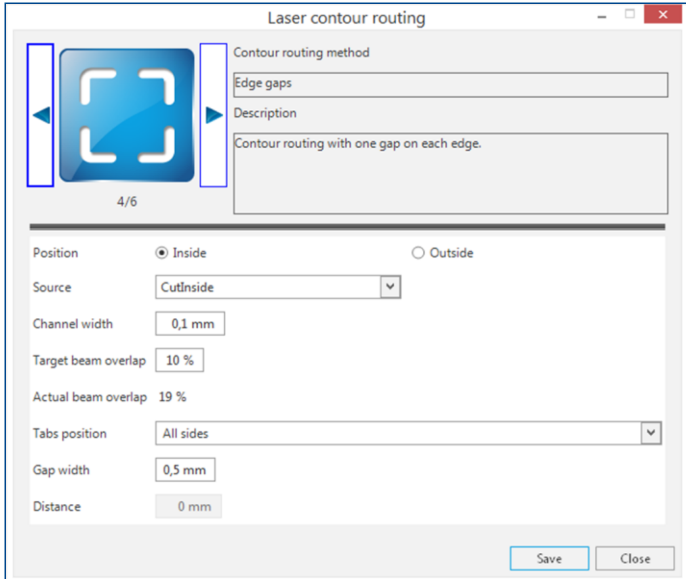


Fig. 123: Dialog *Laser contour routing*

☐ Select *Basic* in the *Contour routing method* group using the arrow buttons:

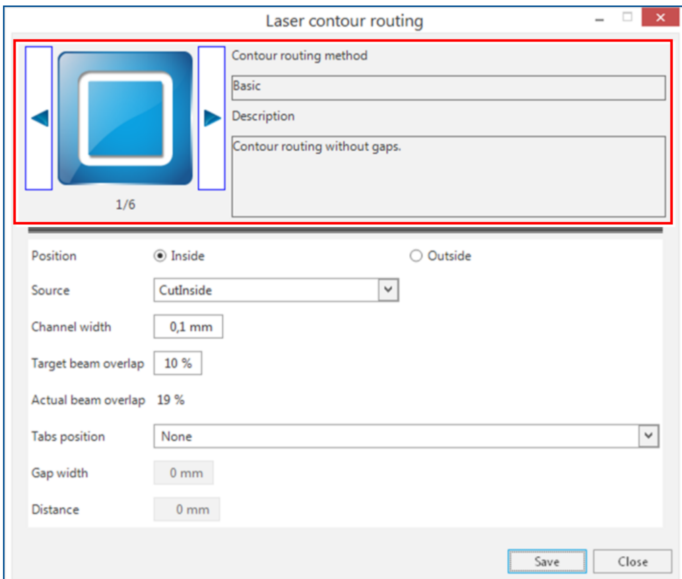


Fig. 124: Contour routing method *Basic*

23. Click on [Save].

☐ The contour routing method has been selected.

24. Expand the group *Processing* and the phase *Prepare Core (Standard)*.

☐ A list of work packages is displayed:

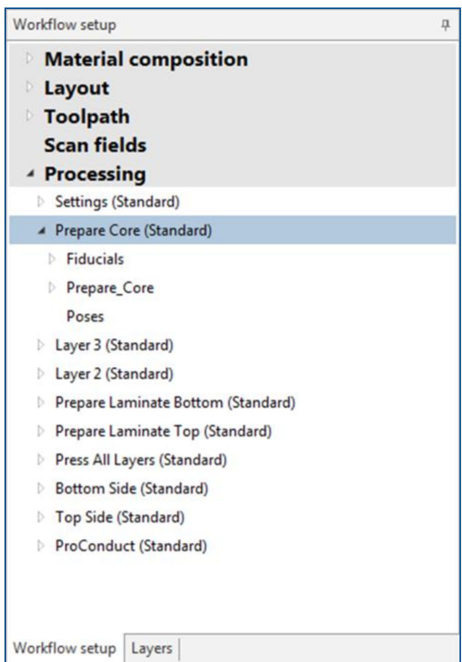


Fig. 125: Work packages of the phase *Prepare Core*

25. Right-click on the work package *Prepare_Core*.

☐ The following context menu is displayed:

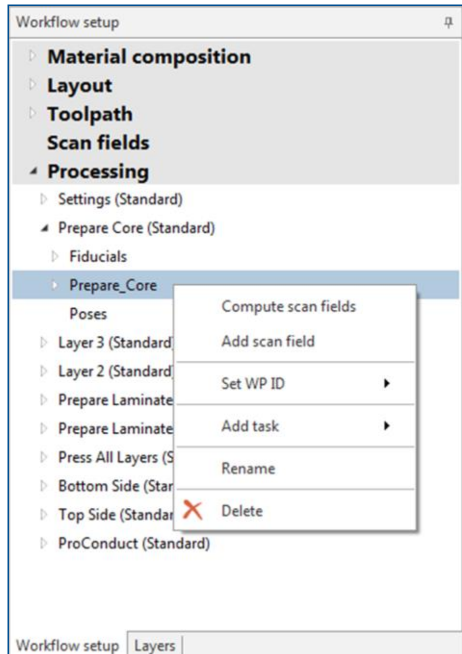


Fig. 126: Context menu *Add task*

☐ Click on *Add task* > *Drill Unplated* > *Cutting*:

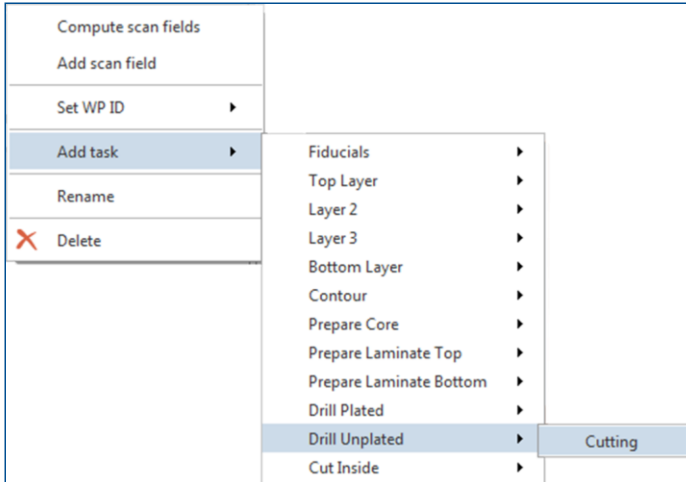


Fig. 127: Adding a task

☐ The task *Drill Unplated* has been added to the work package *Prepare_Core*.

26. Right-click on the work package *Prepare_Core*.
27. Click on *Add task* > *Cut Inside* > *Cutting*.

☐ The task *Cut Inside* has been added to the work package *Prepare_Core*.

28. Expand the work package *Prepare_Core*.

☐ The tasks have been added to the list:

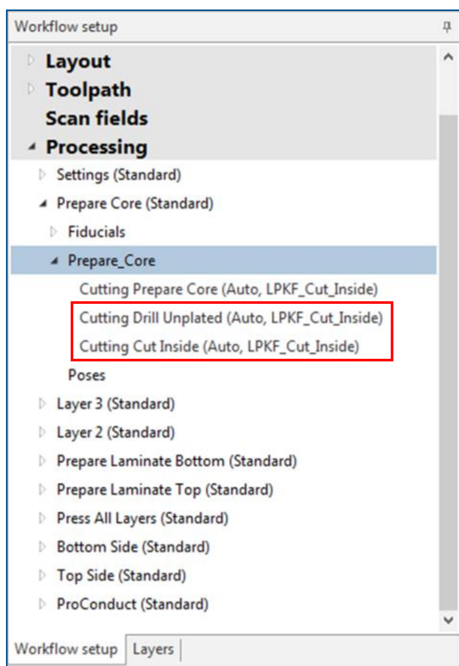


Fig. 128: Added tasks

29. Right-click on the phase *Prepare Core (Standard)*.

☐ The following context menu is displayed:

Fig. 129: Context menu *Compute toolpaths*

30. Click on the context menu item *Compute toolpaths*.
31. Check the *Computation Results* message for possible errors and make corrections, if needed.
32. Click on [Close].
33. Save the file with the suffix *_***Rigid***.

☑ The data for the rigid material have been prepared.

Preparing the rigid material

1. Measure the thickness of the rigid material with a caliper gauge.
2. Click on *Processing* > *Process all* or in the *Processing view* on ▶.
3. After warm-up, the message *Processing phase: Mount core* is displayed.
4. Place the rigid material in the center of the processing table.
5. Click on [OK].
6. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
7. Place the processing data **matching** the board position and size to the processing area used by the CircuitPro PL software.

For detailed information on project placement according to board position and size refer to chapter 1.1, Project Placement, Determining the processing area.

8. When project placement is complete, click on [Set center].

☐ The layout data are now precisely aligned and in the center of the rigid material.

9. Click on [Continue].

☐ The laser system drills the fiducials into the rigid material. Then the positioning holes and pin holes are being drilled in the rigid material and the cutout is being created.

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.

10. When the message *Processing phase: Flip material* is displayed, click on [Cancel] to temporarily stop the process.
11. Remove the rigid material from the system.
12. Spray the rigid material with LPKF Cleaner and use a brush to clean the rigid material, rinse it with tap water and dry it with compressed air.

☑ The rigid material has been prepared.

Preparing the data for the prepreg material

1. Click on *File* > *Open* or on .
2. Select the ***_Rigid** file you previously saved.
3. Click on [Open].

☐ The layout is displayed.

4. Click on *File* > *Save As* and save the file with the suffix ***_Prepreg**.
5. Switch to the *Processing* view.
6. Click on *Processing* > *Material Settings* or on .

☐ The *Material settings* dialog is displayed.

7. Select the *Machine type, Material type* and enter the *Material thickness* (for this example the material type *Polyimide_0.125mm (non-laminated)* is used).

In the CircuitPro PL 3.0. software, the *Material type* (tool library) for prepreg materials does not exist. Therefore, the tool library for polyimide is used.

8. Click on [OK].

☐ The following warning is displayed:

Fig. 130: Warning about deleting the toolpaths

9. Click on [Yes].
10. Switch to *Layout* view.

☐ The scan fields are displayed:

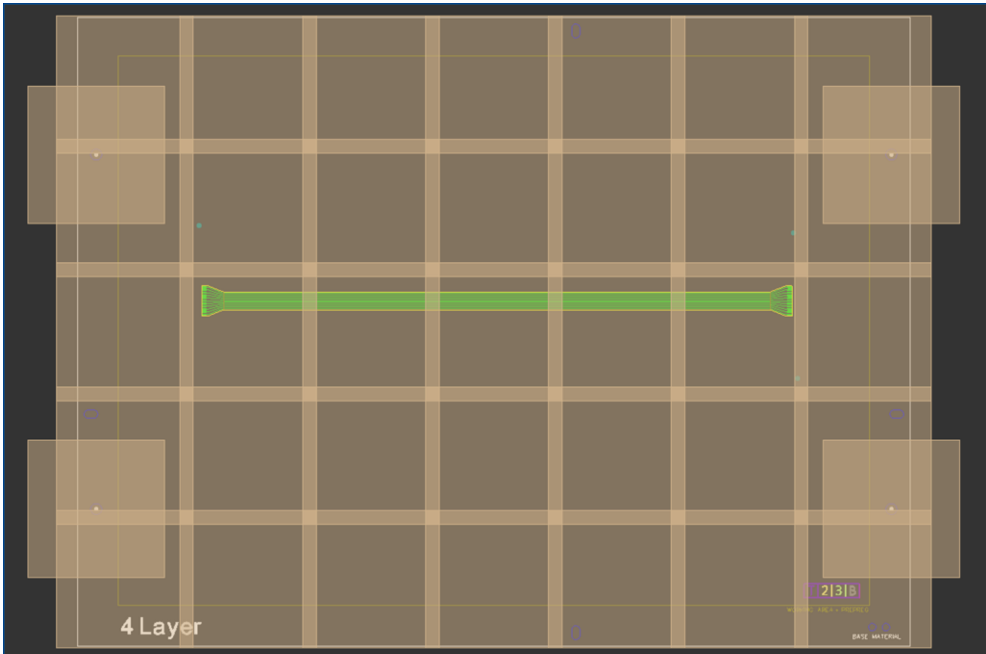


Fig. 131: Scan fields displayed in the *Layout* view

11. In the *Workflow setup* pane, right-click on the group *Scan fields*.

☐ The following context menu is displayed:

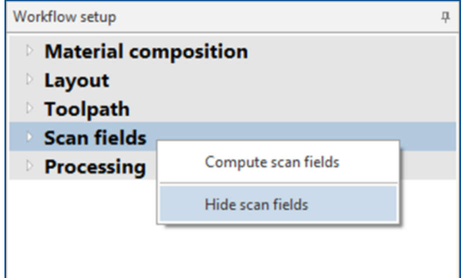


Fig. 132: Context menu | *Hide scan fields*

12. Click on *Hide scan fields*.

☐ The scan fields have been hidden.

The prepreg must **not be used for the flexible part** of the flex-rigid PCB. Therefore, a **cutout** into the **prepreg** material is needed. This cutout is shown as a rectangle on the layer *CutInside*, which is included in the imported example data.

The cutouts around the prepreg positioning holes and the cutout around the flexible part of the flex-rigid PCB have to be **enlarged in the prepreg material**. This prevents the prepreg material from spreading onto the flexible part during the bonding process.

To achieve this, the size of the prepreg positioning holes on the layer *DrillUnplated* and the size of the rectangle on the layer *CutInside* must be increased.

13. Right-click on one of the three prepreg positioning holes located on the layer *DrillUnplated* (the holes that were created in the first step of this tutorial).

☐ The following context menu is displayed:

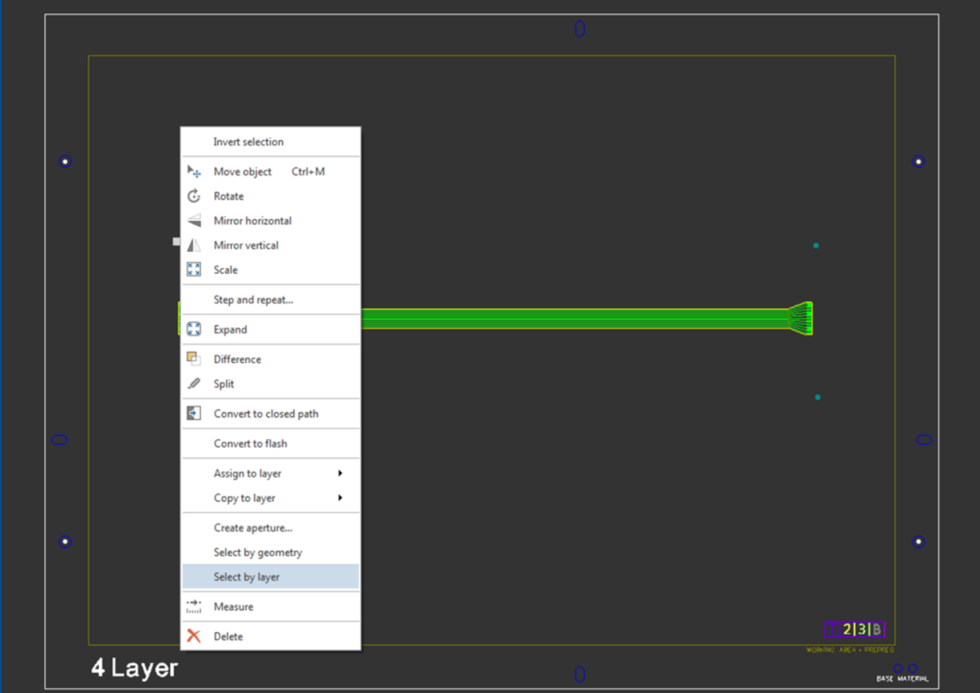


Fig. 133: Context menu *Select by layer*

14. Click on the context menu item *Select by layer*.

☐ All the objects on this layer have been selected and highlighted in gray (in this case the three prepreg positioning holes).

15. Perform one of the following steps:
   – Click on .
   – Clock on *Layout* > *Scale.*
   – Right-click on the black area and select *Scale* from the context menu.

☐ An input field for specifying the scaling coefficient and a drop-down list are displayed:

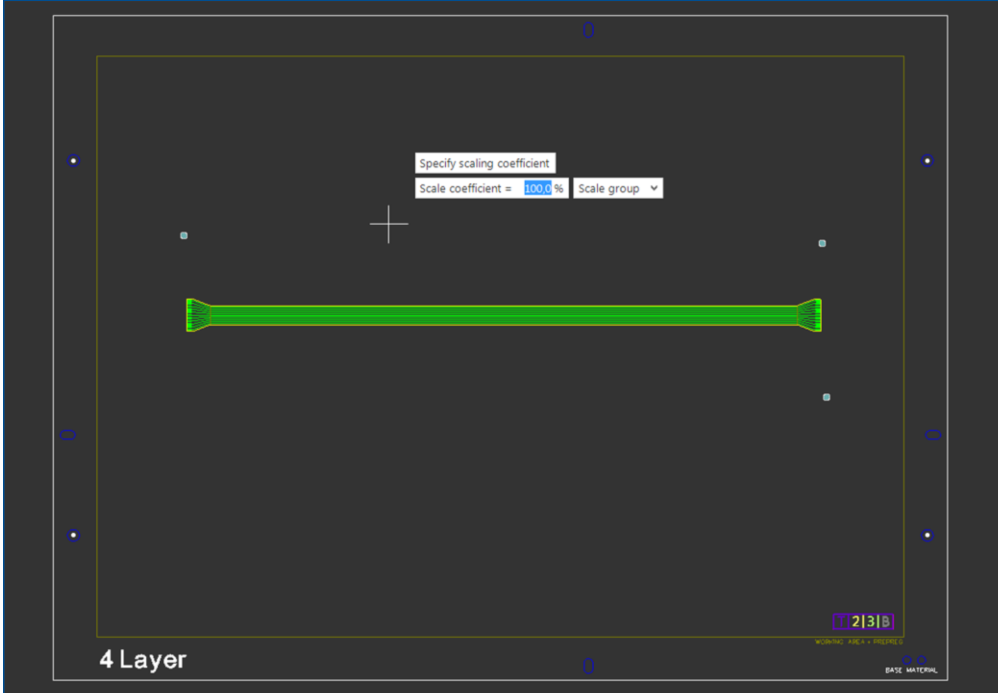


Fig. 134: ***Specify scaling coefficient***

16. Enter the scaling coefficient (in this case **200 %**) in the input field and press the Tab key.

☐ The drop-down list has been activated.

17. Press the ↓ key twice.

☐ The option *Individually* is selected in the drop-down list:

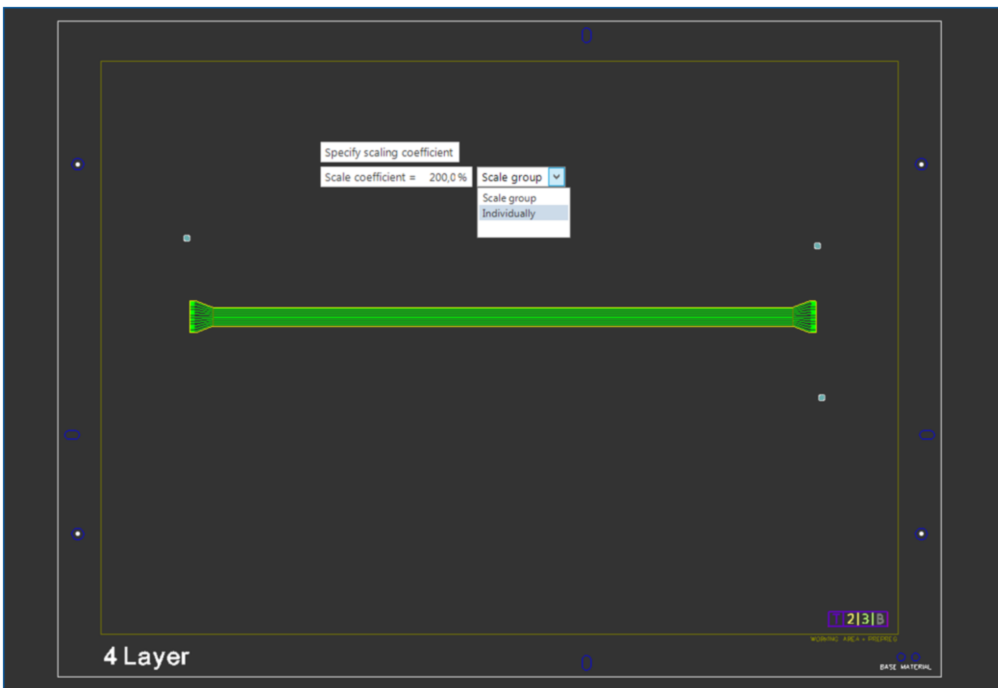


Fig. 135: Drop-down list *Individually*

18. Press the [Enter] key to confirm the selection.

☐ A preview of the scaled prepreg positioning holes is highlighted in orange:

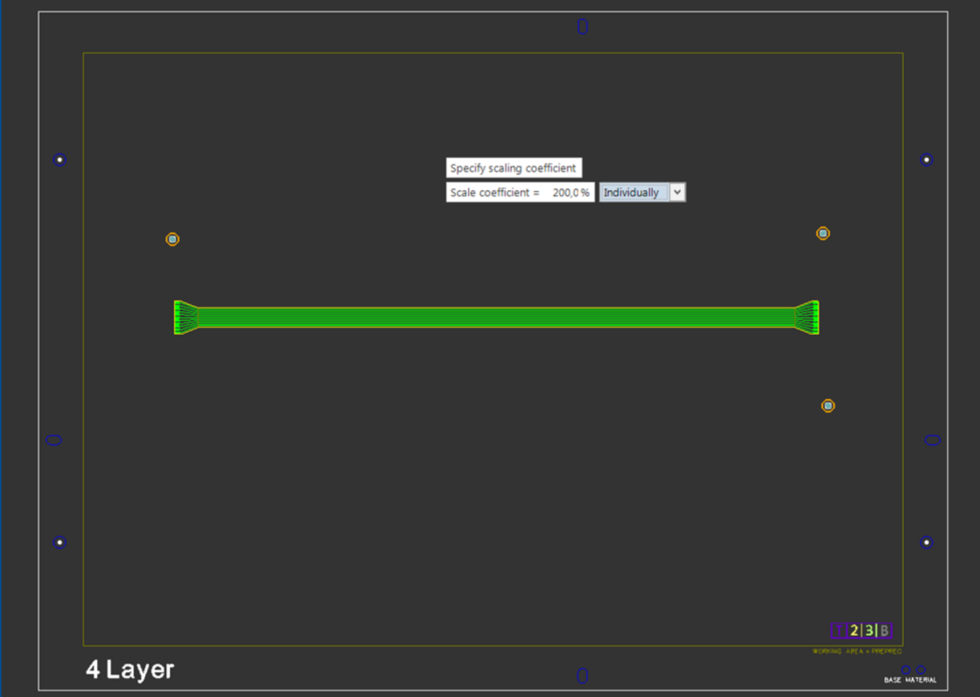


Fig. 136: Preview of the scaled prepreg positioning holes

19. Press the [Enter] key once again to stop the entering.
20. In the *Layout* view click anywhere on the black area or press the [Esc] key to stop the function *Scale*.

☐ The three prepreg positioning holes on the layer *DrillUnplated* have been increased in size.

21. In the *Workflow setup* pane, expand the group *Layout* and the node *Layers*.

☐ A list of layers is displayed:

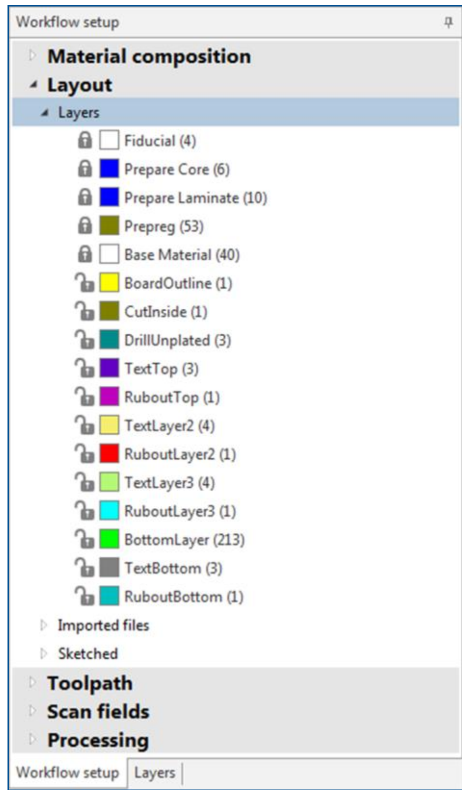


Fig. 137: List of layers

22. Right-click on the layer *CutInside (1)*.

- ☐ The following context menu is displayed:

Fig. 138: Context menu *Select by layer*

23. Click on the context menu item *Select by layer*.

- ☐ All the objects on this layer have been selected and highlighted in gray (in this case 1 rectangle).

The selected rectangle is a contour. To expand this rectangle in size, it has to be filled.

24. Perform one of the following steps:
    - Click on .
    - Click on *Layout* > *Convert to polygon.*
    - Right-click on the black area and select *Convert to polygon* from the context menu.

☐ The selected object has been filled and is highlighted:

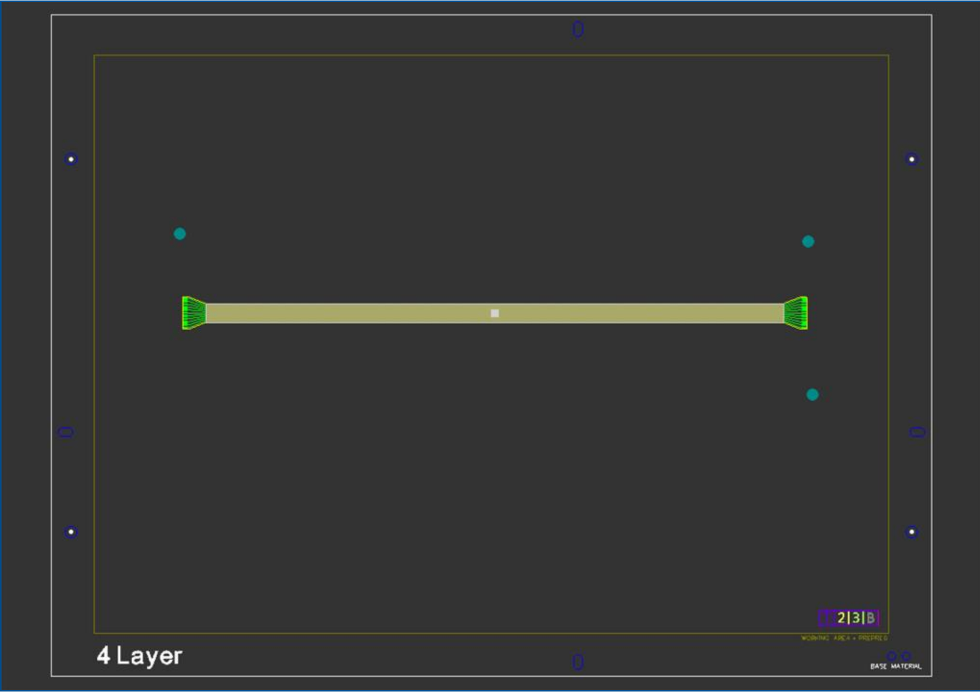


Fig. 139: Filled rectangle

25. Perform one of the following steps:
   – Click on *Layout* > *Expand*.
   – Right-click on the filled rectangle and select *Expand* from the context menu.

☐ An input field for the offset value is displayed:

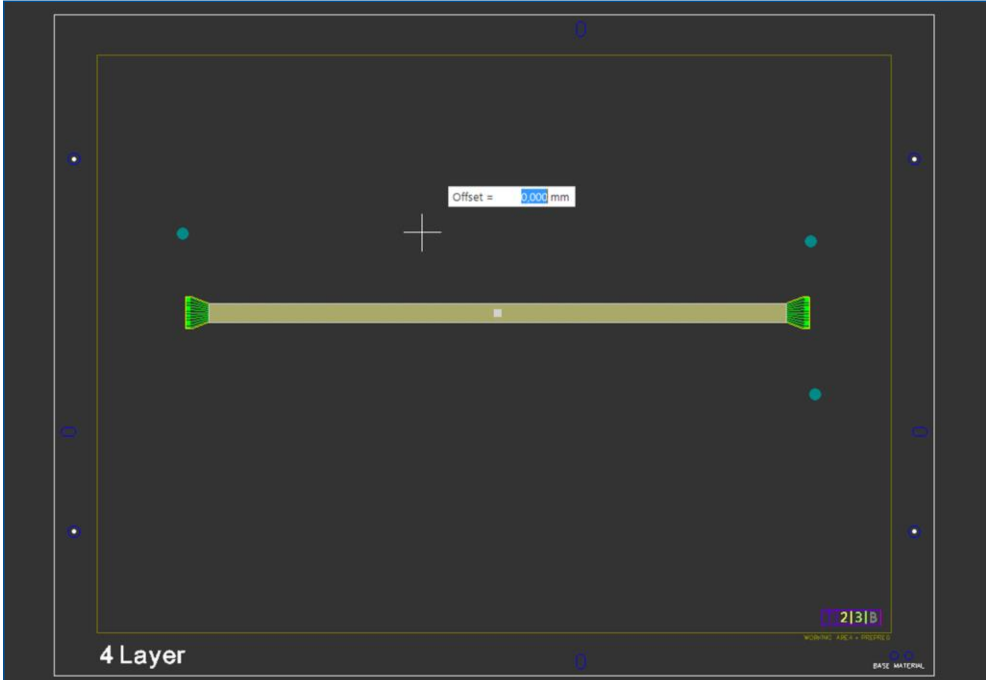


Fig. 140: Input field *Offset* value

26. Enter the value in the input field (in this case **4 mm**).

- A preview of the expanded rectangle is highlighted in orange:

Fig. 141: Preview of the expanded rectangle

27. Press the Enter key to confirm the entry.

- The filled rectangle has been expanded.

28. In the *Layout* view click anywhere on the black area or press the Esc key to deselect the object.

- The layout changes as follows:

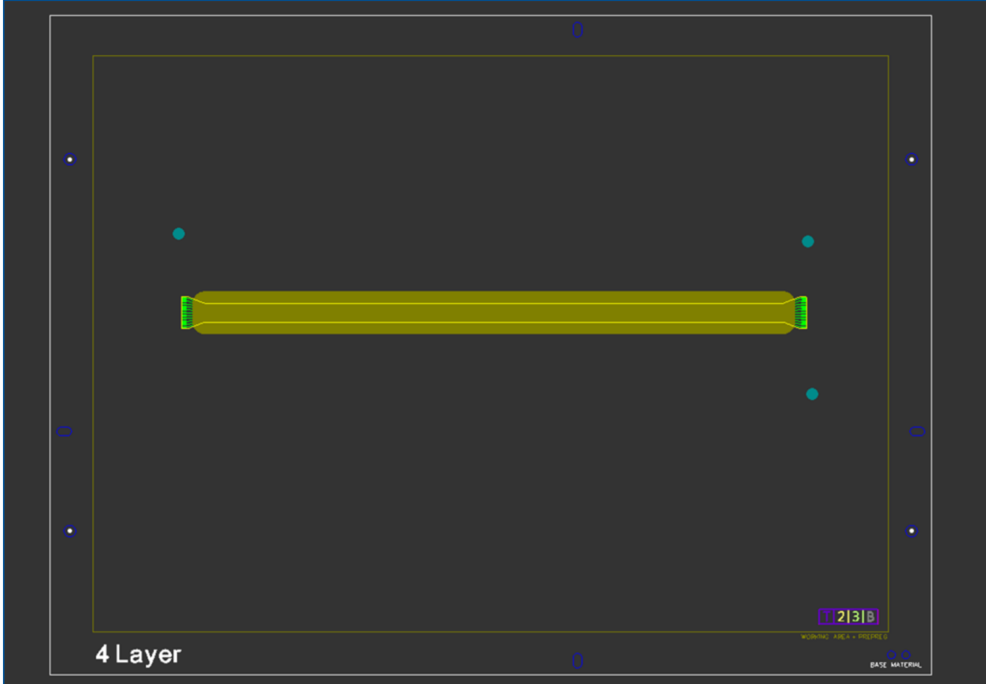


Fig. 142: *Layout* view after expanding the rectangle

29. In the *Workflow setup* pane, expand the group *Toolpath* and right-click on the node *Drill Unplated*.
30. Click on the context menu item *Configure...*

☐ A dialog *Laser contour routing* is displayed.

31. In the field *Channel width* enter **0,01 mm**.

☐ The dialog changes as follows:

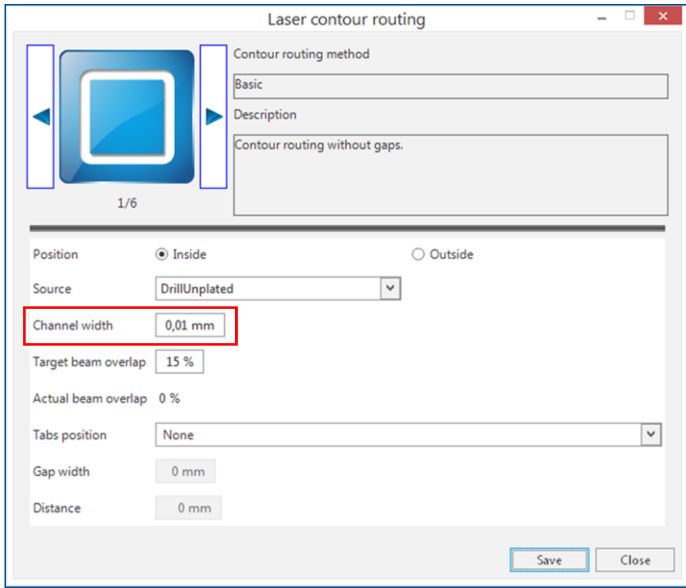


Fig. 143: Dialog *Laser contour routing*

32. Click on [Save].

☐ The *Channel width* has been defined.

33. Right-click on the node *Cut Inside*.
34. Repeat steps 30 to 32.
35. Expand the group *Processing* and the phase *Prepare Core (Standard)*.
36. Expand the work package *Prepare_Core*.
37. Select the task *Cutting Prepare Core (Auto, LPKF_Cut_Inside)* and press the [Del] key or click on ✕.

☐ The task *Cutting Prepare Core (Auto, LPKF_Cut_Inside)* has been deleted.

38. Right-click on the phase *Prepare Core (Standard)* and click on the context menu item *Compute toolpaths*.
39. Check the *Computation Results* message for possible errors and make corrections, if needed.
40. Click on [Close].
41. Click on *File* > *Save* or on 💾.

☑ The data for the prepreg material have been prepared.

Selecting objects on a layer

There are three ways how to select objects on a layer in CircuitPro PL 3.0. Two of them are described in the step above. Another possibility is:

1. Switch to *Layer* pane.

☐ A list of layers is displayed:

Fig. 144: Pane *Layers*

2. Click on the desired layer (in this case *CutInside*).

☑ All the objects on the layer have been selected.

Preparing the prepreg material

1. Switch to the *Processing* view.
2. Place the prepreg material in the center of the processing table and click on to turn on the vacuum.
3. Click on *Processing* > *Placement…* or right-click in the processing area and select *Placement…* from the context menu.

☐ The *Placement* dialog is displayed.

4. Place the processing data **matching** the prepreg position and size to the processing area used by the CircuitPro PL software.

For detailed information on project placement according to board position and size refer to chapter 1.1, Project Placement, Determining the processing area.

5. When project placement is complete, click on [Set center].

☐ The layout data are now precisely aligned and in the center of the prepreg.

6. Click on [Continue].
7. In the *Workflow setup* pane, expand the group *Processing* and the phase *Prepare Core (Standard)*.
8. Right-click on the work package *Prepare_Core* and click on the context menu item *Process*.

☐ The following warning is displayed:

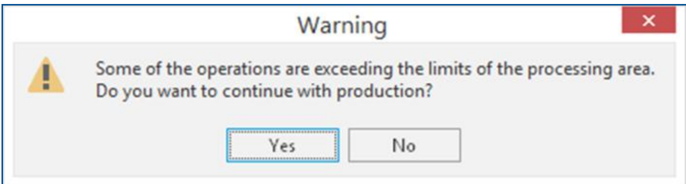


Fig. 145: Warning exceeding the limits

9. Click on [Yes].

☐ The laser system drills the prepreg positioning holes and the cutout is being created in the prepreg material.

10. When the message *Board production finished* is displayed, click on [OK].
11. Remove the prepreg from the system.

☑ The prepreg material has been prepared.

Preparing the data for the flexible material

1. Measure the thickness of the flexible material with a caliper gauge.
2. Click on *File* > *Open or* on .
3. Select the ***_Rigid** file you previously saved.
4. Click on [Open].

☐ The layout is displayed.

5. Click on *File* > *Save As* and save the file with the suffix ***_Flex**.
6. Switch to the *Processing* view.
7. Click on *Processing* > *Material Settings* or on .
8. In the *Material settings* dialog, select the *Machine type* and the *Material type* and enter the *Material thickness* (for this example the material type *PyraluxTK_0.12mm_Cu12 (laminated)* is used).
9. Click on [OK].
10. In the *Workflow setup* pane, expand the group *Processing* and the phase *Prepare Core (Standard)*.
11. Select the work package *Prepare_Core*.
12. Perform one of the following steps:
    - Right-click on the work package *Prepare_Core* and click on the context menu item *Delete*.
    - Press the `Del` key.
    - Click on .

☐ The following dialog is displayed:

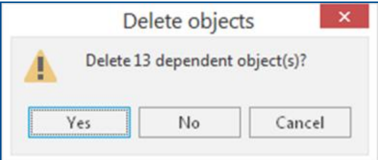


Fig. 146: Dialog *Delete objects*

13. Click on [Yes].

☐ The work package *Prepare_Core* has been deleted.

14. Expand the phase *Prepare Laminate Bottom (Standard)*.
15. Right-click on the work package *Prepare_Laminate* and in the context menu click on *Add task* > *Drill Unplated* > *Cutting*.

☐ The task *Drill Unplated* has been added to the work package *Prepare_Laminate*.

16. Expand the work package *Prepare_Laminate*.

☐ The following tasks are included in this work package:

Workflow setup
Scan fields
Processing
Settings (Standard)
Prepare Core (Standard)
Layer 3 (Standard)
Layer 2 (Standard)
Prepare Laminate Bottom (Standard)
Mount_Laminate
Material_Settings
Placement
Prepare_Laminate
Cutting Prepare Laminate Bottom (Auto, LPKF_Cut_Inside)
Cutting Drill Unplated (Auto, LPKF_Cut_Inside)
Dismount_Material
Poses
Prepare Laminate Top (Standard)
Press All Layers (Standard)
Bottom Side (Standard)
Top Side (Standard)
ProConduct (Standard)
Workflow setup | Layers

Fig. 147: Tasks in the work package *Prepare_Laminate*

17. Right-click on the phase *Prepare Laminate Bottom (Standard)* and click on the context menu item *Compute toolpaths*.
18. Check the *Computation Results* message for possible errors and make corrections, if needed.
19. Click on [Close].
20. Click on *File* > *Save* or on .

☑ The data for the flexible material have been prepared in.

Preparing the flexible material

1. Measure the thickness of the flexible material (pyralux) with a caliper gauge.
2. Right-click on the phase *Prepare Laminate Bottom (Standard)* and click on the context menu item *Process*.
3. When the message *Processing phase: Mount laminate* is displayed, place the flexible material with the **copper side facing upwards** on the center of the processing table.
4. Click on [OK].
5. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
6. Place the processing data **matching** the flexible material position and size to the processing area used by CircuitPro PL.

For detailed information on project placement according to board position and size refer to chapter 1.1, Project Placement, Determining the processing area.

7. When project placement is complete, click on [Set center].

☐ The layout data are aligned and in the center of the flexible material.

8. Click on [Continue].

☐ The laser system drills the positioning holes and pin holes into the flexible material.

9. When the message *Processing phase: Dismount material* is displayed, remove the flexible material from the system and click on [OK].

☑ The flexible material has been prepared.

Assembling and pressing the flex-rigid stack (with the MultiPress S)

For detailed information on assembling and pressing the multi-layer stack refer to chapter 1.3, Multi-layer process.

1. Start the pre-heating process of the MultiPress S.
2. Prepare the materials.
3. Assemble the materials in the press mold using the following order:
   - **Flexible material** (pyralux)

     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. Use the three prepreg positioning holes created in the previous steps for help with alignment. Apply some spray adhesive on the corners of the prepreg material to ensure that its position will remain correct during assembling.
   - **Rigid material** (FR4)

     Align the rigid material to the flexible 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 step **“Preparing the rigid material”**. This strip supports the flexible material during pressing. In this way, better pressing results are achieved.

Apply self-adhesive sealing rings on positioning holes, pin holes and fiducials. This protects the holes from spreading prepreg materials.

4. Press the flex-rigid stack.
5. Cure the flex-rigid stack.

After pressing of the flex-rigid stack, it must **cure** at ambient temperature for at least **12 to 18 hours**.

Alternatively, you can **accelerate the curing** cycle. Heat the flex-rigid stack in an oven for **50 minutes at 100 °C** for the epoxy resin to fully cure. This way the long curing at ambient temperature can be skipped. After heating, cool the flex-rigid PCB at ambient temperature for 5 minutes before proceeding with production.

☑ The flex-rigid stack has been assembled and pressed.

Structuring the flexible part of the flex-rigid PCB

1. Measure the thickness of the flex-rigid PCB with a caliper gauge.
2. Click on *File* > *Open or* on .
3. Select the ***_Flex*** file you previously saved.
4. Click on [Open].

- The layout is displayed.

5. Switch to the *Processing* view.
6. Click on *Processing* > *Material Settings* or on .
7. In the *Material settings* dialog select the *Machine type* and the *Material type* and enter the *Material thickness* (for this example the material type *PyraluxTK_0.12mm_Cu12 (laminated)* is used).
8. Click on [OK].
9. In the *Workflow setup* pane expand the group *Processing* and right-click on the phase *Bottom Side (Standard)*.
10. Click on the context menu item *Compute toolpaths*.
11. Check the *Computation Results* message for possible errors and make corrections, if needed.
12. Click on [Close].
13. Right-click on the phase *Bottom Side (Standard)* and select the context menu item *Process*.
14. When the message *Processing phase: Mount material bottom* is displayed, place the flex-rigid PCB with the **flexible side (pyralux) facing upwards** on the center of the processing table (the positioning holes should be located in the right rear corner).
15. Click on [OK].
16. In the *Material settings* dialog, enter the *Material thickness* and click on [Continue].
17. Place the processing data **matching** 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, Project placement, Matching the fiducial position.

18. When project placement is complete, click on [Continue].

- The laser system reads the fiducials on the Bottom side of the flex-rigid PCB (*BottomLayer*).

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

- The flexible part of the flex-rigid PCB (*BottomLayer*) is being structured.

19. When the message *Board production finished* is displayed, click on [OK].
20. Click on *File* > *Save* or on .
21. Leave the flex-rigid PCB mounted on the system's processing table.

☑ The flexible part of the flex-rigid PCB has been structured.

Cutting out the flex-rigid PCB

1. Continue working with the ***_Flex*** file.
2. Click on *File* > *Save As* and save the file with the suffix ***_Cutout***.
3. Switch to the *Processing* view.
4. Click on *Processing* > *Material Settings* or on .
5. In the *Material settings* dialog, select the *Machine type* and the *Material type* and enter the *Material thickness* you measured in the previous procedure (for this example the material type *FR4_1.55mm_Cu18 (laminated)* is used).
6. Click on [OK].
7. In the *Workflow setup* pane, expand the group *Toolpath* and right-click on the node *Contour*.
8. Click on the context menu item *Configure...*

☐ The dialog *Laser contour routing* is displayed.

9. Select *Basic* in the *Contour routing method* group using the arrow buttons.
10. Click on [Save].

☐ The contour routing method has been selected.

To **avoid flipping** the flex-rigid PCB during structuring, the phase *Top Side (Standard)* must be set to *Mirrored*.

11. Expand the group *Processing* and right-click on the phase *Top Side (Standard)*.

☐ Click on the context menu item *Set mirrored* > *Mirrored*.

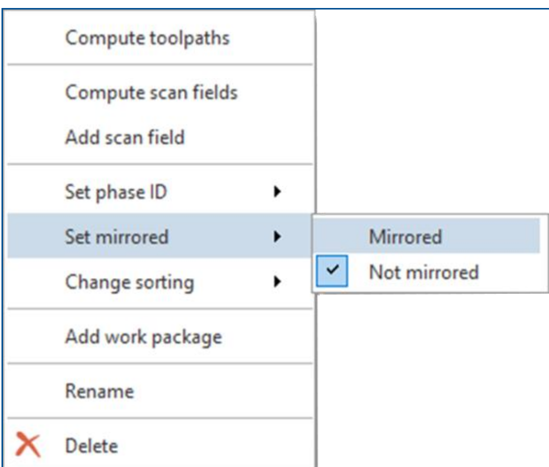


Fig. 148: Setting the phase *Top Side (Standard)* to *Mirrored*

☐ The phase *Top Side (Standard)* has been set to *Mirrored*.

12. Expand the group *Toolpath*, right-click on the node *Contour* and click on the context menu item *Compute*.
13. Check the *Computation Results* message for possible errors and make corrections, if needed.
14. Click on [Close].
15. Expand the group *Processing* and the phase *Top Side (Standard)*.
16. Right-click on the work package *Cutting* and click on the context menu item *Process*.

☐ The laser system reads the fiducials on the Bottom side (*BottomLayer*) and cuts out the flex-rigid PCB.

17. When the message *Board production finished* is displayed, click on [OK].
18. Click on *File* > *Save* or on .
19. Remove the flex-rigid PCB from the system.

20. Spray the flex-rigid PCB with LPKF Cleaner and use a brush to clean the flex-rigid PCB, rinse under tap water and dry it with compressed air.

☑ The flex-rigid PCB has been cut out.

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.

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

Prerequisites

- Fully structured and through-plated PCB

Auxiliaries

- LPKF Cleaner (order no. 115891)
- Hot air oven (order no. 115877)
- Oil-free compressed air
- Tap water

System

- ProtoLaser U4/S4
- 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 3.0
- Structuring the solder mask

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 the resist and hardener.
2. Coat the PCB.

Ensure that the fiducials are not coated!

3. Predry the PCB in a hot air oven.
4. Expose the PCB **without the artwork** to the UV light.
5. Postcure the PCB in a hot air oven.

☑ The solder resist has been applied on the PCB.

For detailed information on applying the solder mask on the PCB refer to ProMask/ProLegend Process description, chapters 4.2, 4.3 and 4.5.

Preparing the data for solder mask structuring in CircuitPro PL 3.0

For detailed information on preparing the data with CircuitPro PL 3.0 refer to chapter 2.1.

1. Measure the thickness of the coated PCB with a caliper gauge.
2. 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 on *File* > *Import* or on . Make sure you assign them to layers *SolderMaskBottom* and *SolderMaskTop*.

- ☐ Your layout is displayed in the *Layout* view; the scan fields have been computed and are displayed:

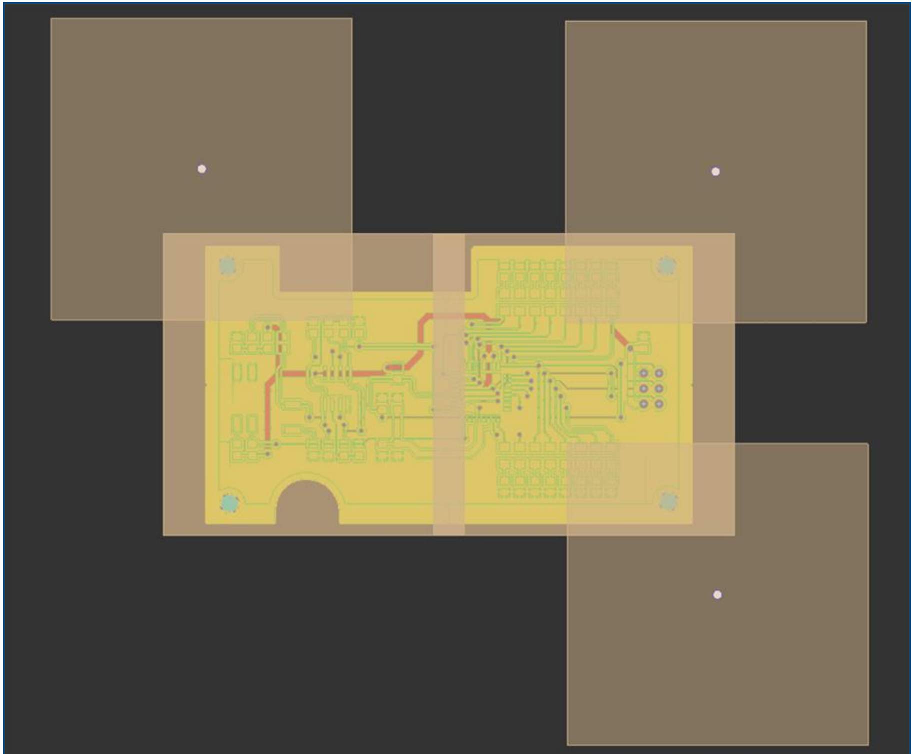


Fig. 149: Layout with computed and displayed scan fields

3. In the *Workflow setup* pane, expand the group *Layout* and expand the node *Layers*.

- ☐ A list of layers is displayed.

4. Select the following layers by clicking on them while pressing the `Ctrl` key:
   - *Fiducial*
   - *SolderMaskTop*
   - *SolderMaskBottom*

- The view changes as follows:

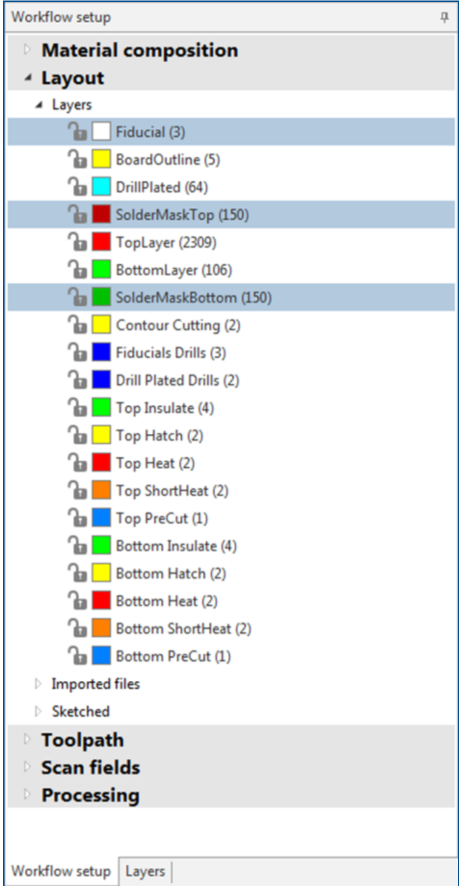


Fig. 150: Selected layers

5. Click on *Edit* > *Copy* or press the key combination `Ctrl` + `C`.

- The selected layers have been copied to the clipboard.

6. Click on *File* > *New* or click on .

- The dialog *New document* is displayed.

7. Select the template *ProMask.cp2d*.
8. Click on [Load template].
9. In the *Material settings* dialog, select the *Machine type and the Material type* and enter the *Material thickness* (in this example the material type *ProMask_removal (non-laminated)* is used).
10. Click on [OK].
11. Click on *Edit* > *Paste* or press the key combination `Ctrl` + `V`.

- The copied layers (and the objects they contain) have been pasted and are displayed in the *Layout* view:

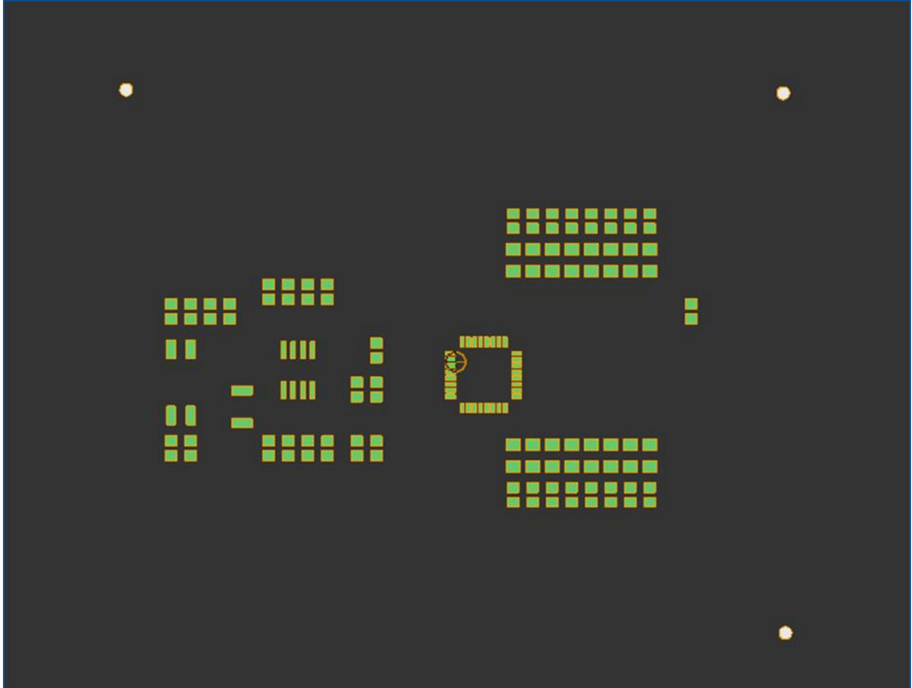


Fig. 151: **Preview of the pasted layers**

12. Click anywhere in the black area or press the `Enter` key to position the pasted layers and objects.
13. In the *Layout* view, click anywhere on the black area or press the `Esc` key to deselect the objects.

- The *Layout* view changes as follows:

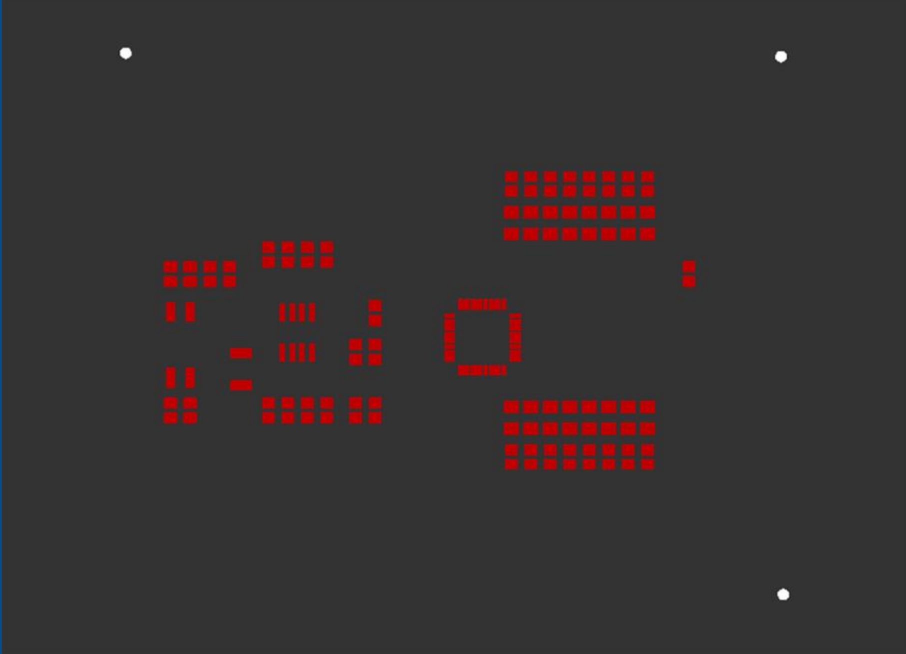


Fig. 152: ***Layout* view of pasted objects**

14. In the *Workflow setup* pane, right-click on the group *Toolpath* and click on the context menu item *Compute all*.
15. Check the *Computation Results* message for possible warnings or errors and make corrections, if needed.
16. Click on [Close].

☑ The data for solder mask structuring have been prepared in CircuitPro PL 3.0.

Structuring the solder mask

1. Click on *Processing* > *Process all* or in the *Processing* view on ▶.
2. After warm-up, the message *Processing phase: Mount material* is displayed.
3. Place the coated PCB with the **Bottom side** (*BottomLayer*) **facing upwards** onto the processing table.
4. Click on [OK].
5. In the *Material settings* dialog, enter the *Material thickness* you measured in the previous procedure and click on [Continue].
6. Place the processing data **matching** the location of the coated PCB and fiducials on the processing table.

For detailed information on project placement according to fiducial positions refer to chapter 1.1, Project placement, Matching the fiducial position.

7. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Bottom side of the coated PCB (*BottomLayer*).

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

☐ The laser system removes the solder resist from the solder pads on the Bottom side of the PCB (*BottomLayer*).

8. 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].
9. Place the processing data **matching** the location of the PCB and fiducials on the processing table.
10. When project placement is complete, click on [Continue].

☐ The laser system reads the fiducials on the Top side (*TopLayer*) and removes the solder resist from the solder pads on the Top side (*TopLayer*).

11. After processing of the Top side (*TopLayer*) is finished, remove the PCB from the system.
12. Spray the PCB with LPKF Cleaner and use a brush to clean the it.
13. 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.

Processing files

5 Processing files

This chapter describes how to import and process different file formats (CAD files). It also shows typical errors during the import and suitable measures for troubleshooting.

5.1 Processing DXF files in CircuitPro PL 3.0

This chapter describes how to process DXF files in CircuitPro PL 3.0. It describes how to import and to convert a DXF file.

The following steps are performed in this tutorial:

- Importing the DXF file
- Assigning the objects to corresponding layers
- Converting objects to polygons
- Converting drill holes to flash objects

Importing a DXF file

1. In the *Templates* tab of the *New document* dialog, select the template that suits the type of project you have (in this example the template *PL-U4_SingleSided_Top.cp2d*. is used).
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type* as well as the *Material type* and enter the *Material thickness* (in this example the material type *FR4_1.00mm_Cu18 (laminated)* is used).
4. Click on [OK].
5. Click on *File* > *Import* or on .
6. 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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PL 3.0\Example Data\UseCase_DXFFiles`.
7. Select the files you wish to import (in this example *Tutor_dxf.dxf*).
8. Click on [Open].

☐ The dialog *Import* is displayed.

9. In the column *Layer/Template*, select the layer *TopLayer* from the drop-down list.

☐ The dialog changes as follows:

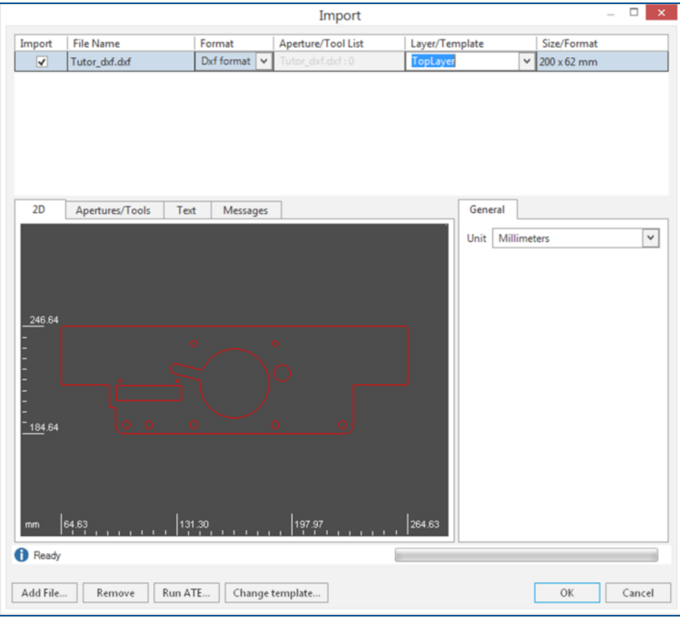


Fig. 153: Dialog *Import* and assigned layer

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 *General* under *Unit*. Should this not be the case, select the correct unit from the drop-down list.

10. Click on [OK].

☐ The *Layout* view changes as follows:

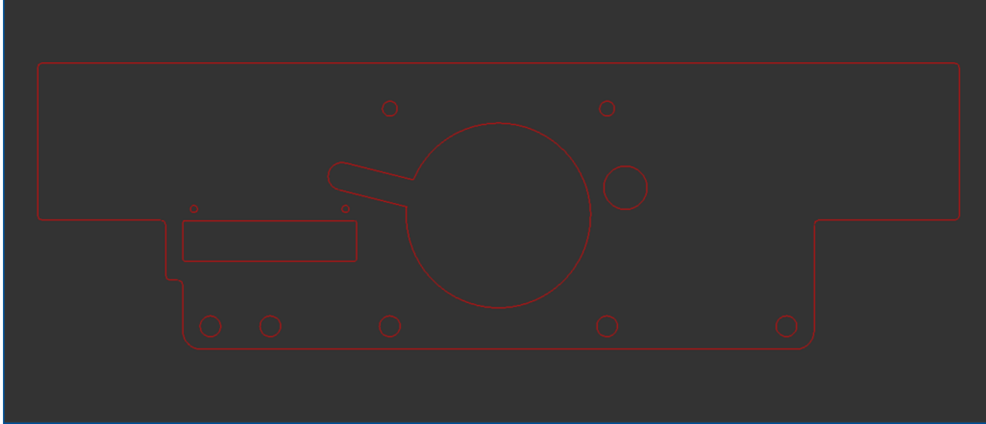


Fig. 154: *Layout* view of the imported data

☑ The DXF file has been imported.

Data on non-visible layers

Certain data of DXF files could be on non-visible layers. When importing such files, the corresponding layer is also non-visible in CircuitPro PL 3.0. You can make these layers visible by switching to the *Layers* pane and activating the check boxes in the column *Vis.*

Assigning the objects to the corresponding layers

Data imported from a DXF file can consist of **open paths**. Should this be the case, open paths need to be **converted** to **closed paths** before starting the processing.

1. Select the board outline. To do so, click on all its elements while pressing the `Ctrl` key.

- The board outline elements are highlighted in gray:

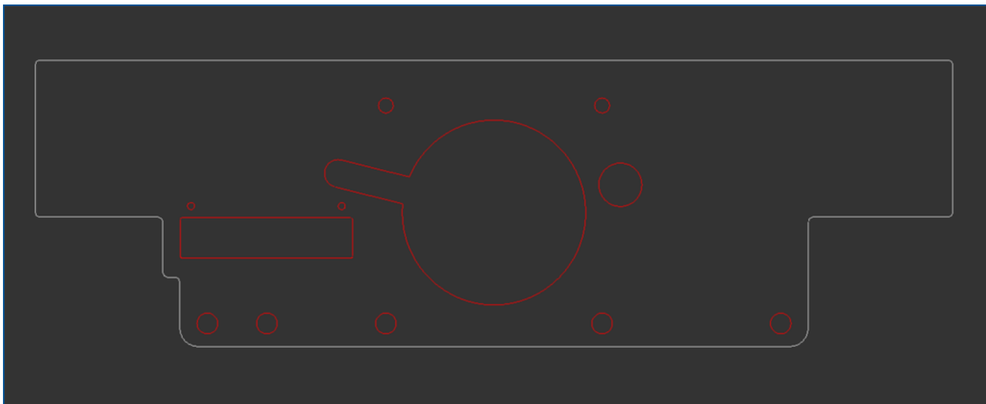


Fig. 155: Elements selected

2. Right-click on the black area and click in the context menu on *Combine open paths*.

- The *Layout* view changes as follows:

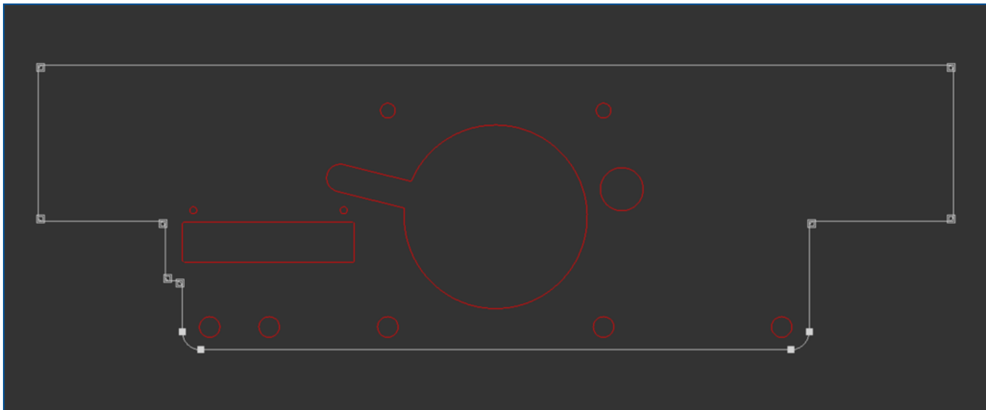


Fig. 156: Combined open path

- The open paths of the board outline have been combined.

3. Right-click on the black area.

- ☐ In the context menu, click on *Assign to layer* > *BoardOutline*:

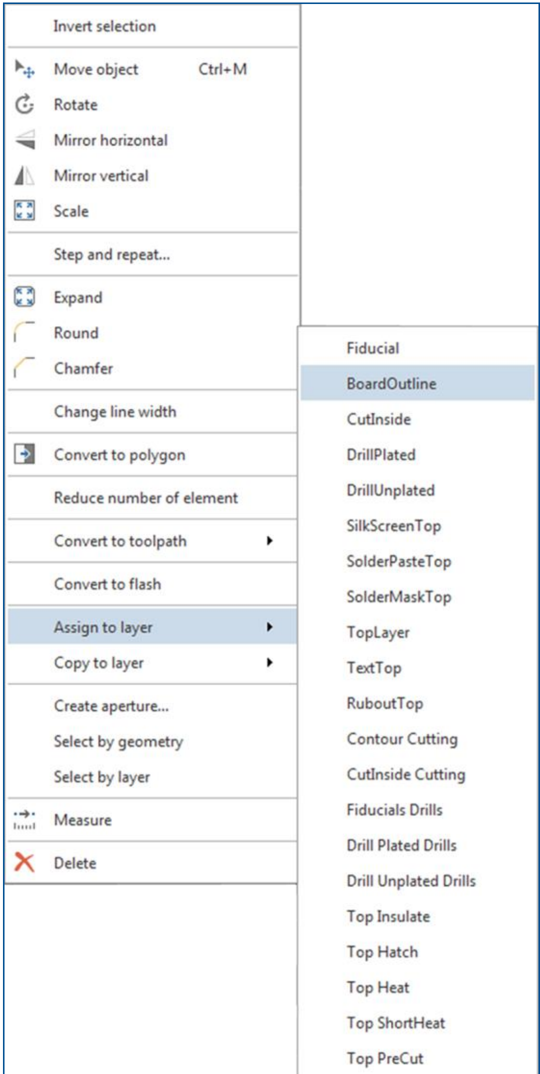


Fig. 157: Assigning combined path to the layer *BoardOutline*

- ☐ The combined path has been assigned to the layer *BoardOutline*.

4. In the *Layout* view, click anywhere on the black area or press the Esc key to deselect the combined path.

- ☐ The board outline is highlighted in yellow:

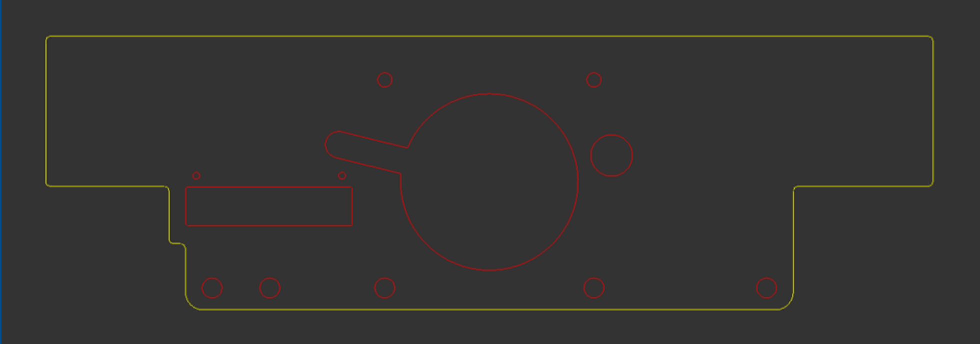


Fig. 158: Board outline assigned to layer

5. Select all paths that are to be assigned to the *TopLayer* by pressing the Ctrl key.

☐ The paths are highlighted in gray:

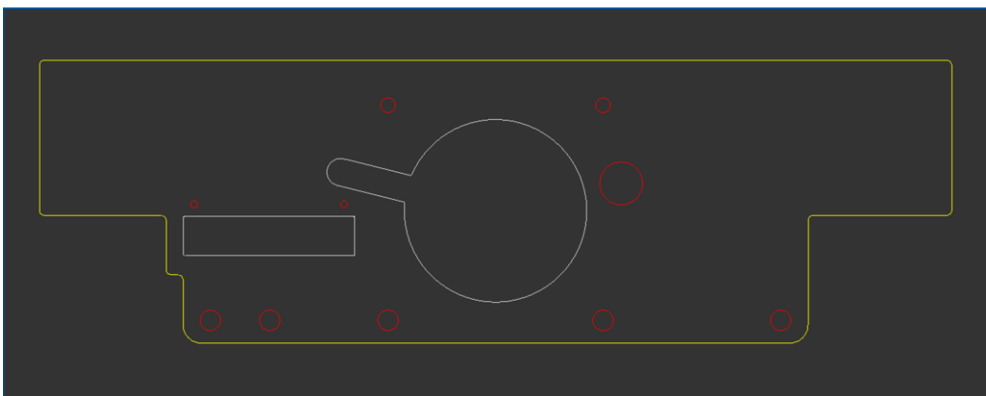


Fig. 159: Top layer paths selected

6. Right-click on the black area and select *Combine open paths* from the context menu.

☐ The paths have been combined.

7. Select the combined paths that are to be assigned to the *TopLayer* by clicking on them while pressing the Ctrl key.

☐ The combined paths are highlighted in gray.

8. Right-click on the black area and in the context menu click on *Assign to layer > TopLayer*.

☐ The selected combined paths have been assigned to the *TopLayer*.

9. In the *Layout* view, click anywhere on the black area or press the Esc key to deselect the combined paths.

10. Select all the holes that are to be assigned to the layer *DrillUnplated* by clicking on them while pressing the Ctrl key.

☐ The holes are highlighted in gray:

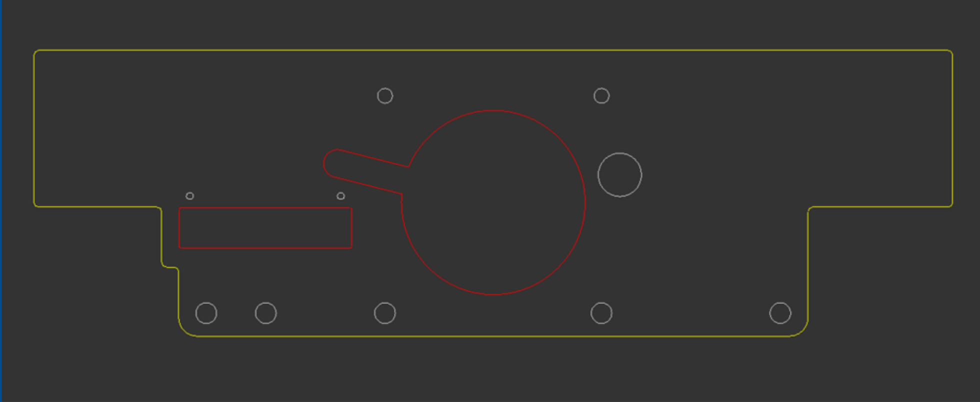


Fig. 160: Holes selected

11. Right-click on the black area and in the context menu click on *Assign to layer > DrillUnplated*.

☐ The selected holes have been assigned to the layer *DrillUnplated*.

12. In the *Layout* view, click anywhere on the black area or press the Esc key to deselect the holes.

☐ The color of the holes changes to dark green:

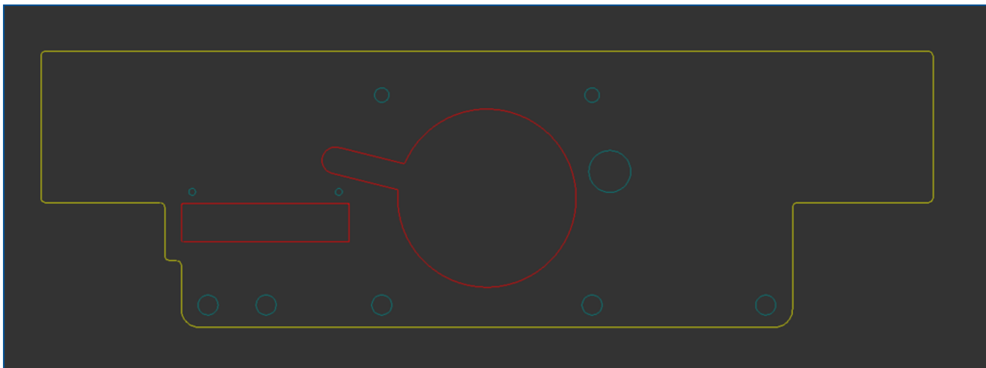


Fig. 161: Holes assigned to layer

☑ The objects have been assigned to the corresponding layers.

Problems with combining open paths

After selecting multiple elements and applying the *Combine open paths* function, it can happen that the **elements** are **still not connected**.

There are two possible causes of this problem:

- Not all the elements of an object are selected. Select the multiple elements once again and pay special attention not to miss the elements on the corners of the object (they are usually quite small).
- Click on *File* > *Options*. In the *Options* dialog click on *Miscellaneous* and adjust the value *Curve combine*. This setting determines how far the paths to be combined can be apart for the *Combine open paths* function to work.
- There could be a gap between two elements that is too large and results from the DXF file. Connect the separated elements by using the drawing tools. If the design problem cannot be solved in CircuitPro PL 3.0., return to the AutoCAD software and redesign your layout.

Converting objects to polygons

Closed path objects which are not be rubbed-out during processing need to be converted to polygons before starting the processing.

1. In the *Workflow setup* pane, expand the group *Layout* and expand the node *Layers*.

☐ A list of layers is displayed:

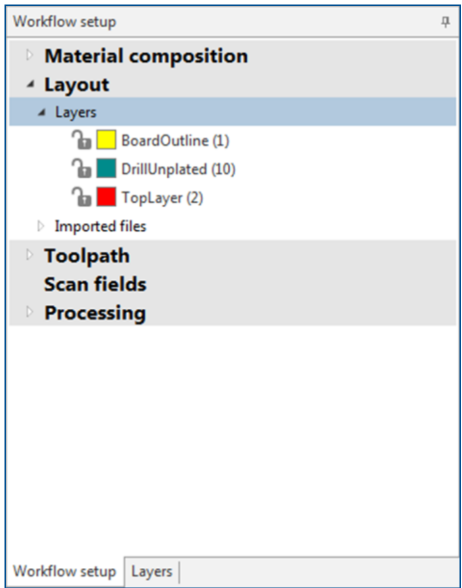


Fig. 162: List of layers

2. Right-click on the *TopLayer (2)* and select the context menu item *Select by layer*.

☐ All objects on this layer have been selected.

3. Perform one of the following steps:
   - Click on .
   - Click on *Layout > Convert to polygon*.
   - Right-click on the black area and select *Convert to polygon* from the context menu.

☐ The selected objects have been filled.

4. In the *Layout* view, click anywhere on the black area or press the Esc key to deselect the objects.

☐ The objects are highlighted in red:

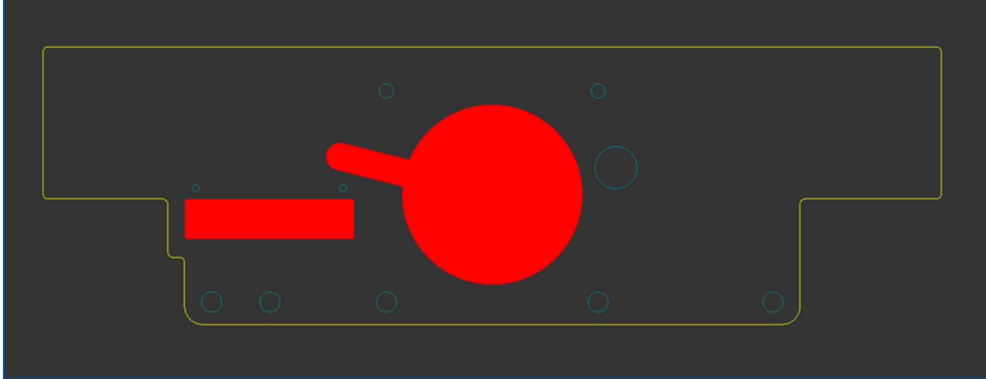


Fig. 163: Filled objects

☑ The objects have been converted to polygons.

Converting drill holes to flash objects

Holes need to be converted to flash objects before starting the processing.

1. Right-click on one of the holes located on the layer *DrillUnplated* and click in the context menu on *Select by layer*.

☐ All the holes on this layer have been selected and are highlighted in gray.

2. Right-click on the black area and click in the context menu on *Convert to flash*.

☐ The selected drill holes have been converted to flash.

3. In the *Layout* view click anywhere on the black area or press the Esc key to deselect the drill holes.

☑ The drill holes have been converted to flash objects.

The DXF file has been successfully imported and converted in CircuitPro PL 3.0.

5.2 Processing Gerber and Excellon files in CircuitPro PL 3.0

This chapter describes how to process Gerber and Excellon files in CircuitPro PL 3.0. It explains the most common difficulties when importing these types of files and offers tips for troubleshooting.

The following steps are performed in this tutorial:

- Selecting Gerber and Excellon 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
- Viewing/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 *Templates* tab of the *New document* dialog, select the template that suits the type of PCB you wish to process (in this example the template *PL-U4_SingleSided_Bottom.cp2d*. is used).
2. Click on [Load template].
3. In the *Material settings* dialog, select the *Machine type* as well as the *Material type* and enter the *Material thickness* (in this example the material type *FR4_1.00mm_Cu18 (laminated)* is used).
4. Click on [OK].
5. Click on *File* > *Import* or on .
6. 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:\Users\User name\Documents\LPKF Laser & Electronics\LPKF CircuitPro PL 3.0\Example Data\UseCase_BasicCAMOperations.`

Instead of selecting individual Gerber or Excellon files, you can select a whole folder. In this case, all files in the selected folder are displayed in the dialog *Import*.

7. Select the folder *UseCase_BasicCAMOperations*.
8. Click on [Open].

- The dialog *Import* is displayed:

Fig. 164: Dialog *Import*

The columns of the table displayed contain the following information/settings:

Column	Description
Import	Activate the check boxes of the files that you wish to import.
File Name	The name of the selected file is displayed.
Format	The format of the selected file is displayed. If CircuitPro PL 3.0 has not recognized the file format correctly, you can assign the correct format in the corresponding drop-down list.
Aperture/Tool List	The Aperture list is usually a part of the Gerber or Excellon file. The apertures contained are displayed in the tab *Apertures/Tools*.
Layer/Template	The layer that is to contain the imported data is displayed or can be assigned.
Size/Format	The size of the imported layout is displayed in this column.

Table 7: Table columns

In the column *Import* **deactivate** the check boxes of the files you **do not wish to import** (in this example: *Tutor.SMB*, *Tutor.SMT*, *Tutor.SPT* and *Tutor.TOP*).

☐ The dialog *Import* changes as follows:

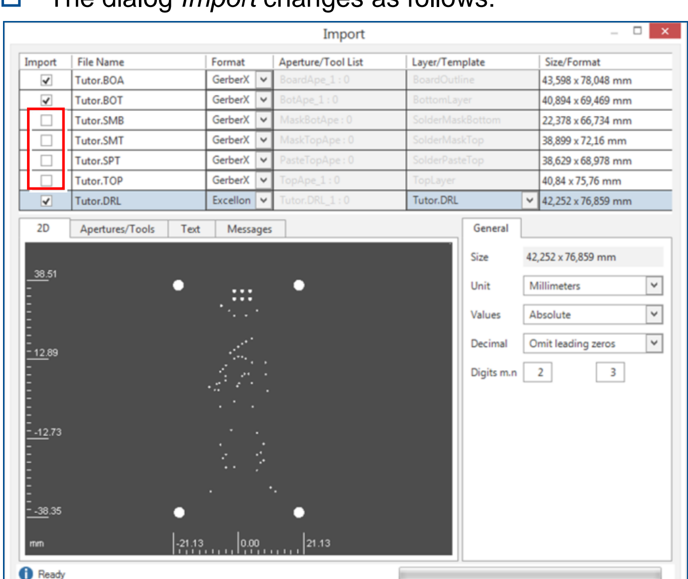


Fig. 165: **Deselected files**

The files have been deselected.

☑ The Gerber and Excellon files have been selected.

Selecting the file format

Sometimes CircuitPro PL 3.0 does not recognize the format of the file selected.

1. Select the files you wish to import (in this example *Tutor.BOA*, *Tutor.BOT* and *Tutor.DRL*).
2. Click on [Open].

Note that the file problem described in this procedure was created intentionally. The UseCase files do not contain these problems.

- In the dialog *Import*, the file format as well as other file-specific information is missing (= *Undefined*).

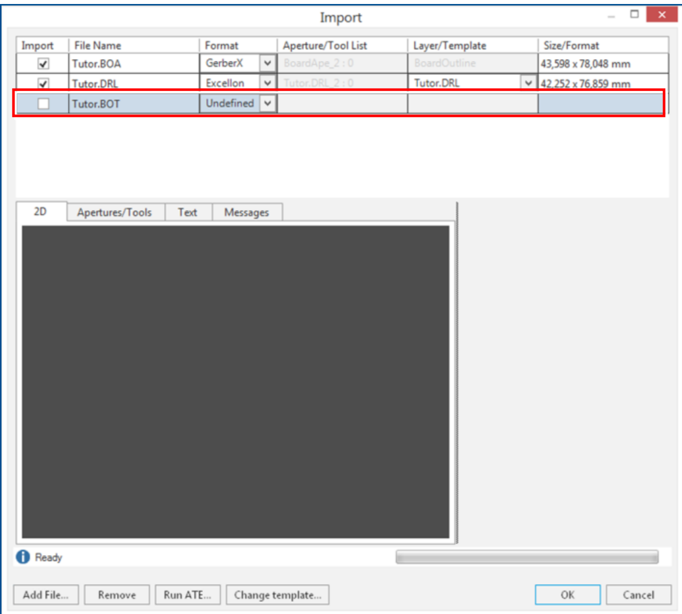


Fig. 166: **Dialog *Import* and *Format* undefined**

- In the column *Format* select the appropriate file format from the drop-down list (in this case *GerberX*):

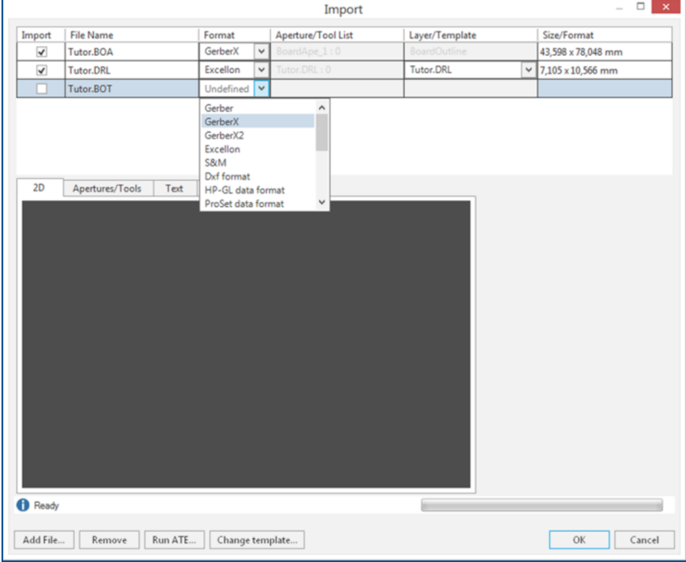


Fig. 167: **Selecting the appropriate file format**

☐ The missing file-specific information is displayed automatically and the check box in the column *Import* is activated:

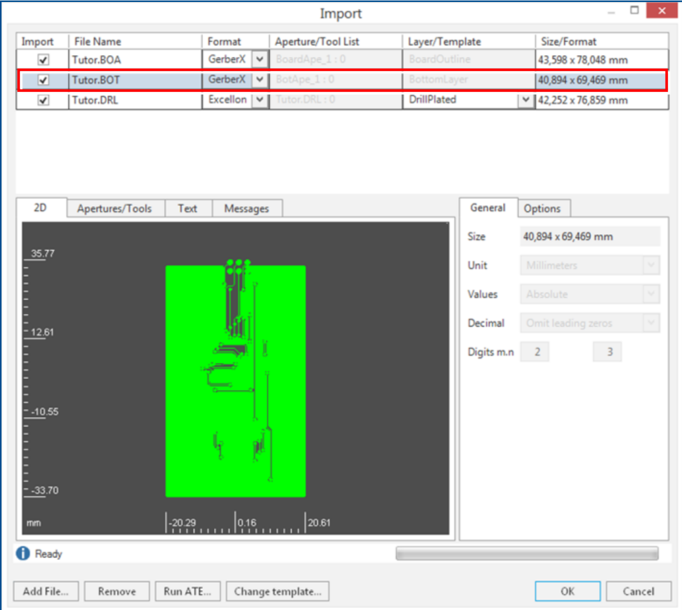


Fig. 168: **Appropriate file format selected**

☑ The file format has been selected.

Sometimes CircuitPro PL 3.0 does not assign the file to the desired layer automatically.

There are two ways to assign layers. This takes place either during or after the files import. Both possibilities are described as follows:

Assigning the desired target layer during import

1. Select the files you wish to import (in this example *Tutor.BOA*, *Tutor.BOT* and *Tutor.DRL*).
2. Click on [Open].

- In the dialog *Import*, the file *Tutor.DRL* has not been automatically assigned to an existing layer (the cell in the *Layer/Template* column is active and contains the name of the file):

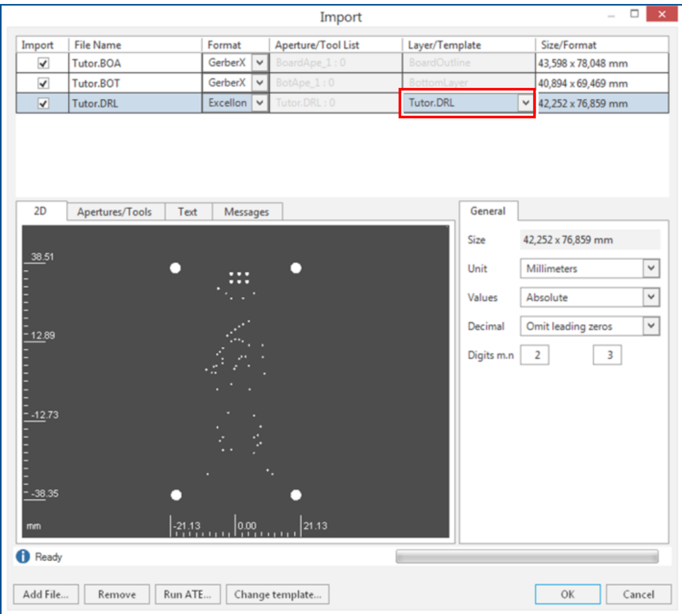


Fig. 169: **Unassigned layer**

3. Click on the arrow button in the *Layer/Template* column.

☐ A drop-down list is displayed:

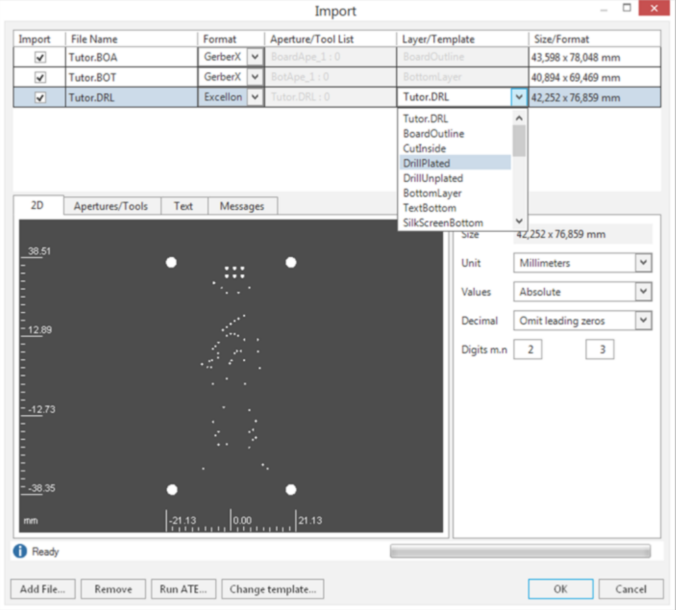


Fig. 170: **Selecting the target layer**

4. Select the desired target layer (in this example *DrillPlated*).

☐ The dialog *Import* changes as follows:

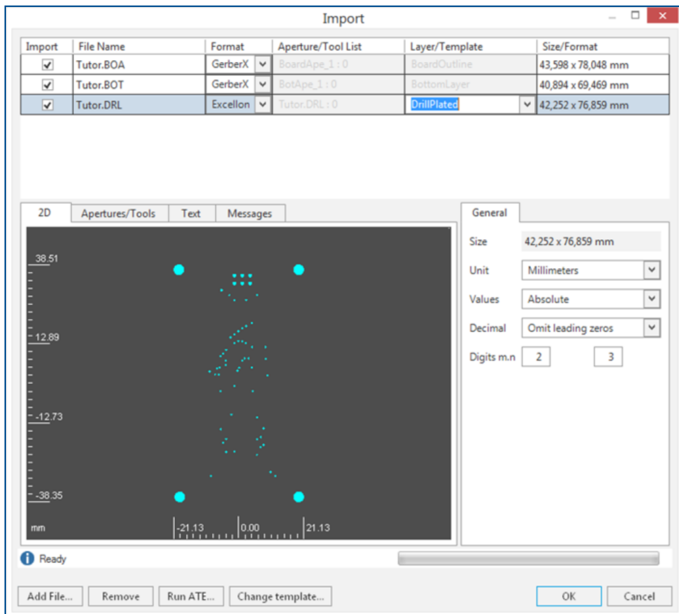


Fig. 171: **Selected target layer**

5. Click on [OK].

☐ The processing data are displayed in the *Layout* view:

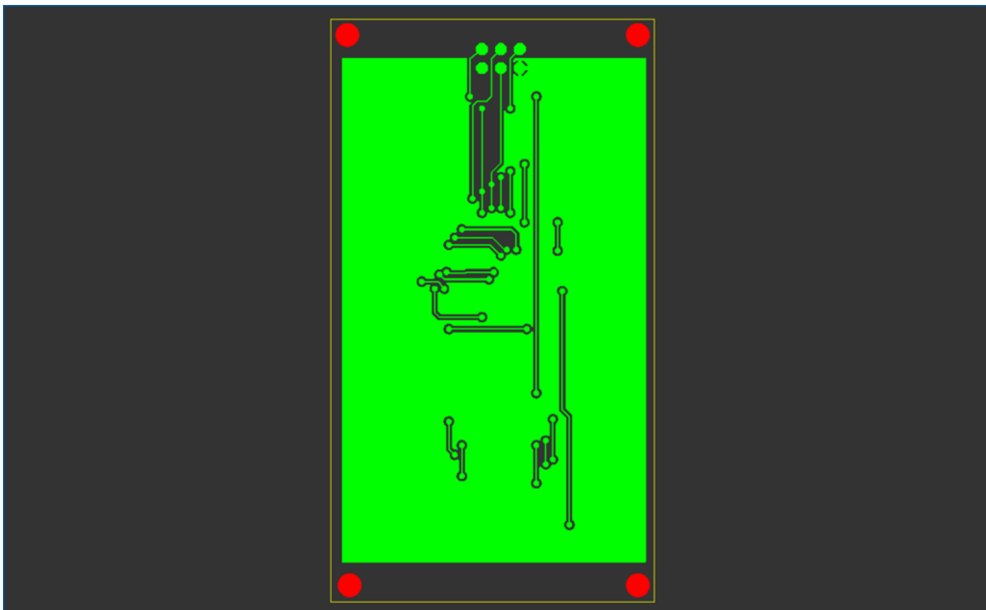


Fig. 172: **Layout view of the imported files**

☑ The desired target layer has been assigned during import.

Assigning the desired target layer after import

1. Select the files you wish to import (in this example *Tutor.BOA*, *Tutor.BOT* and *Tutor.DRL*).
2. Click on [Open].

☐ In the dialog *Import* the file *Tutor.DRL* has not been automatically assigned to an existing layer:

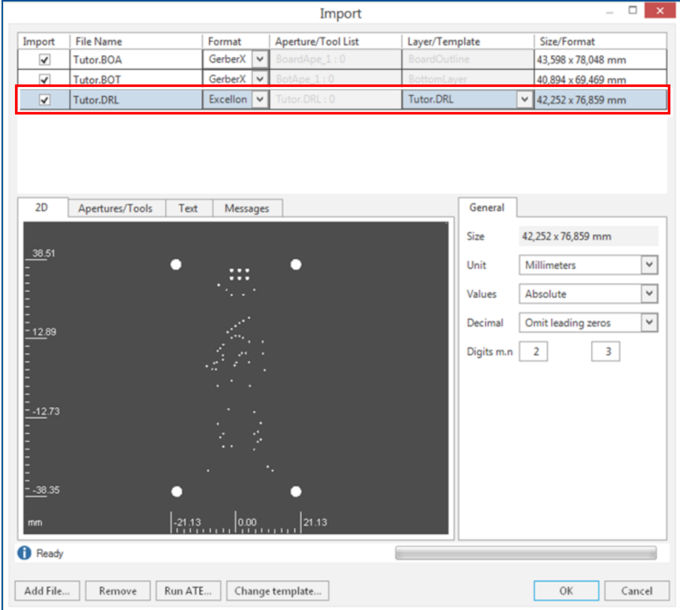


Fig. 173: **Unassigned layer**

3. Click on [OK].

☐ The following warning is displayed:

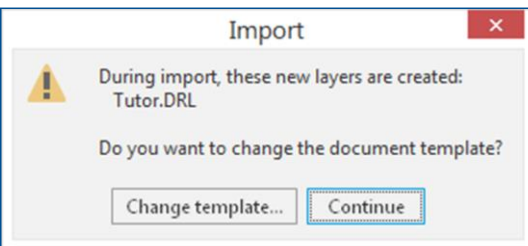


Fig. 174: **Warning *Import***

4. Click on [Continue].

☐ The following warning is displayed:

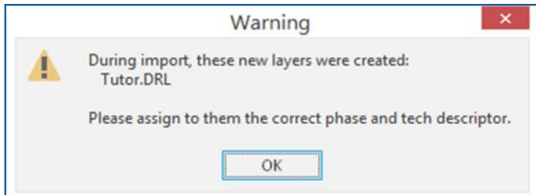


Fig. 175: **Warning about new layer created**

5. Click on [OK].

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

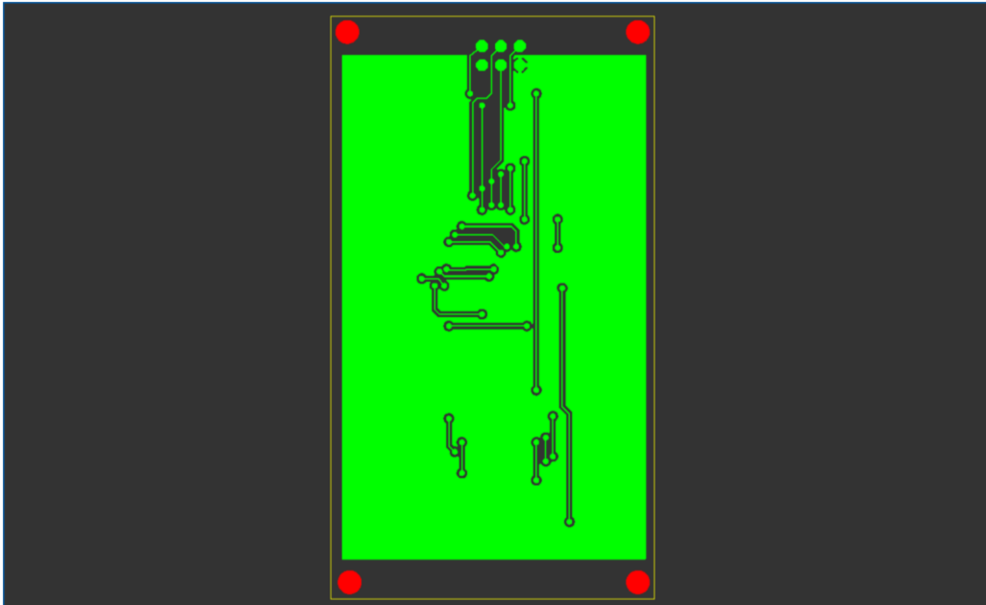


Fig. 176: **Layout view of the imported files**

6. In the *Workflow setup* pane, expand the group *Layout* and expand the node *Layers*.

☐ A list of layers is displayed:

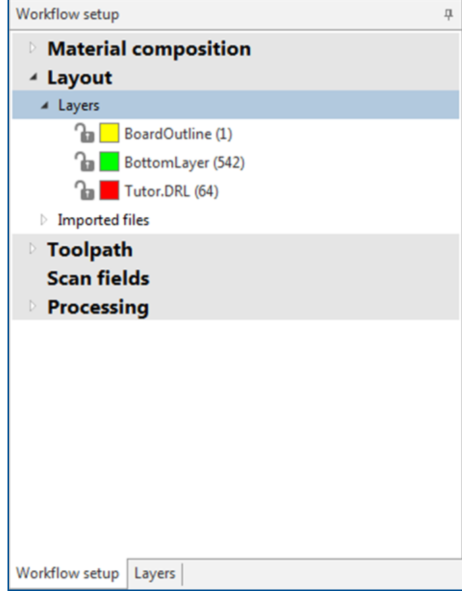


Fig. 177: **List of layers**

7. Right-click on the layer *Tutor.DRL (64)* and click on the context menu item *Select by layer*.

☐ All the objects on this layer are selected and highlighted.

8. Right-click on the black area.

- ☐ In the context menu, click on *Assign to layer* > *DrillPlated*:

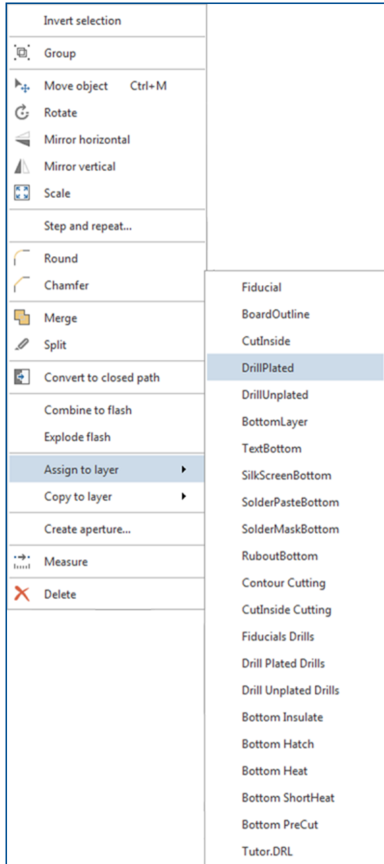


Fig. 178: **Assigning objects**

9. In the *Layout* view, click anywhere on the black area or press the Esc key to deselect the highlighted objects.

- ☐ The color of the drill holes changes to turquoise:

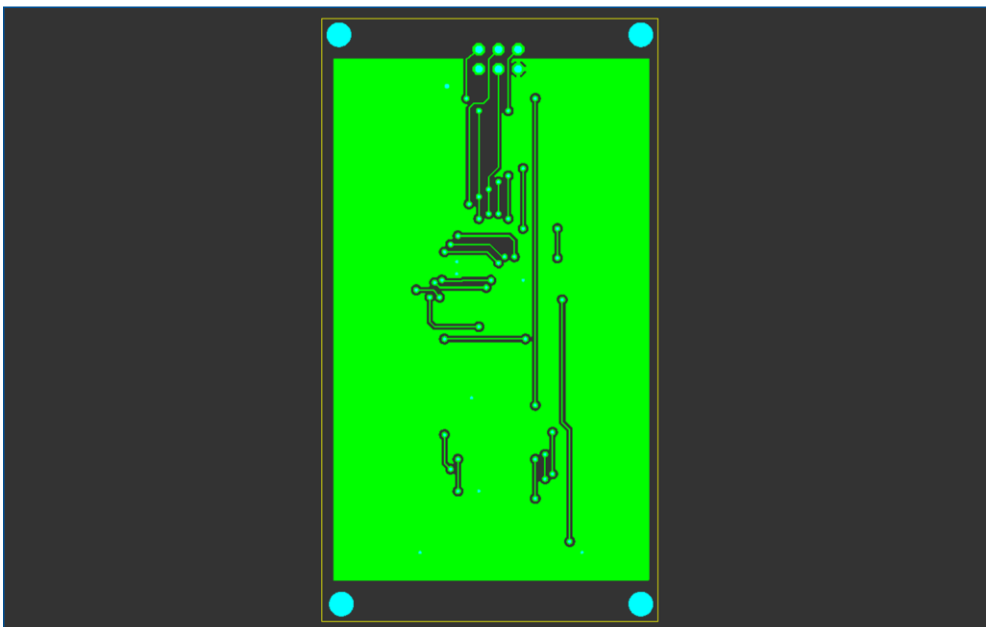


Fig. 179: **Assigned drill holes**

- ☐ The objects from the layer *Tutor.DRL* have been assigned to the layer *DrillPlated*.
- ☑ 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 *Layer/Template* 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.

To do so, perform the following:

1. Click on *File > Options…*

❑ The dialog *Options* is displayed.

2. Click on *Import assignments*.

❑ The dialog changes as follows:

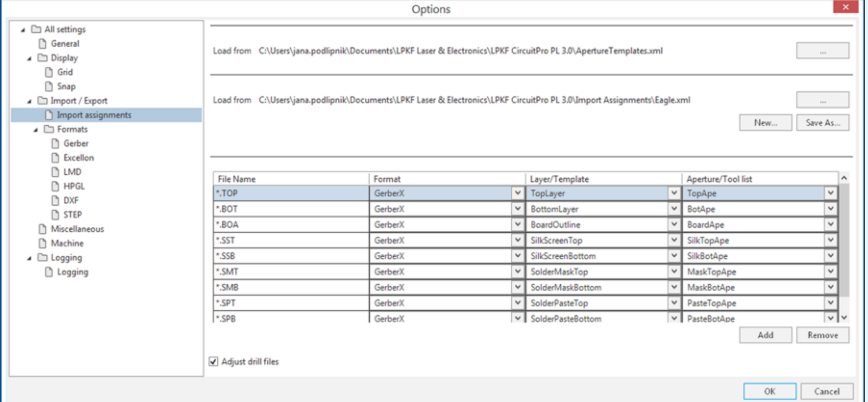


Fig. 180: ***Import assignments***

3. Click on [New].
4. Enter the new .xml file name and click on [Save].

❑ The dialog changes as follows:

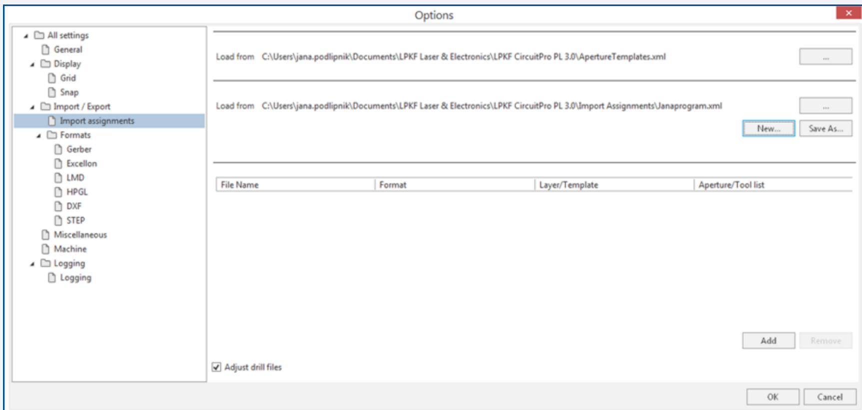


Fig. 181: **New import assignments**

5. Click on [Add].
6. Enter the file name suffix 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 steps 6 and 7 for creating further *Import assignments*.
9. When you have finished creating the desired *Import assignments*, click on [OK].

❑ The following figure displays an example of an import assignment setting for the file names with the suffix **.CUT*:

File Name	Format	Layer/Template	Aperture/Tool list
*.CUT	GerberX	BoardOutline	BoardApe

Fig. 182: **Example of an import assignment**

Setting/correcting the size and format

Sometimes the layout of the file is not displayed correctly in CircuitPro PL 3.0. There are four possible causes for this:

- Wrong measurement unit: A wrong measurement unit was selected when importing the file (inch instead of mm).
- Wrong number of decimal digits: The number of decimal digits entered does not match the file's contents.
- Wrong declaration of the value (relative/absolute).
- Wrong zero suppression (decimal).

1. Select the files you wish to import (in this example *Tutor.BOA*, *Tutor.BOT* and *Tutor.DRL*).
2. Click on [Open].

Note that the file problem described in this procedure was created intentionally. The UseCase files do not contain these problems.

- ☐ In the dialog *Import*, the file *Tutor.DRL* displays a peculiarly large size of the layout (see column *Size/Format*):

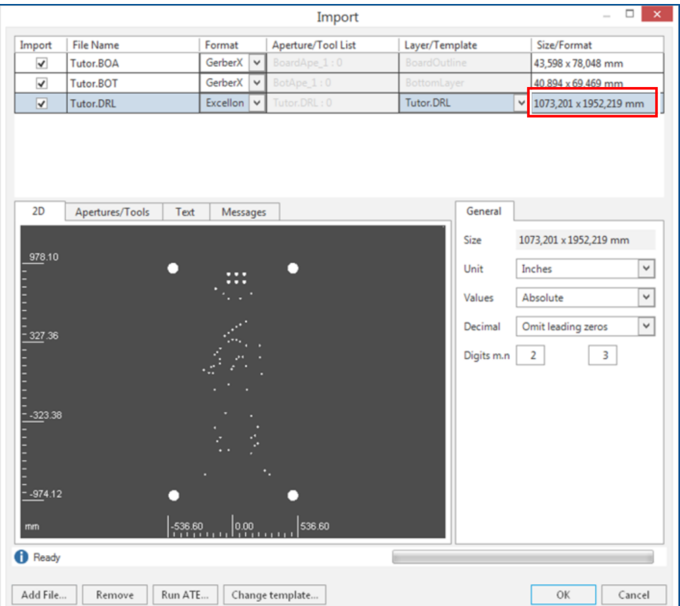


Fig. 183: **Large layout size**

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 *General* of the *Import* dialog (in this example *Inches* are selected).

☐ Click on the arrow button and select *Millimeters* from the drop-down list:

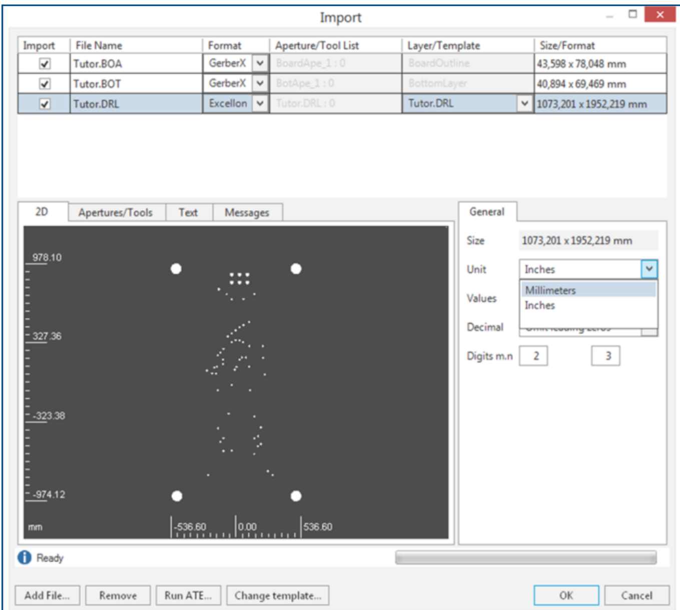


Fig. 184: **Selecting the appropriate unit**

☐ The dimensions of the layout change automatically:

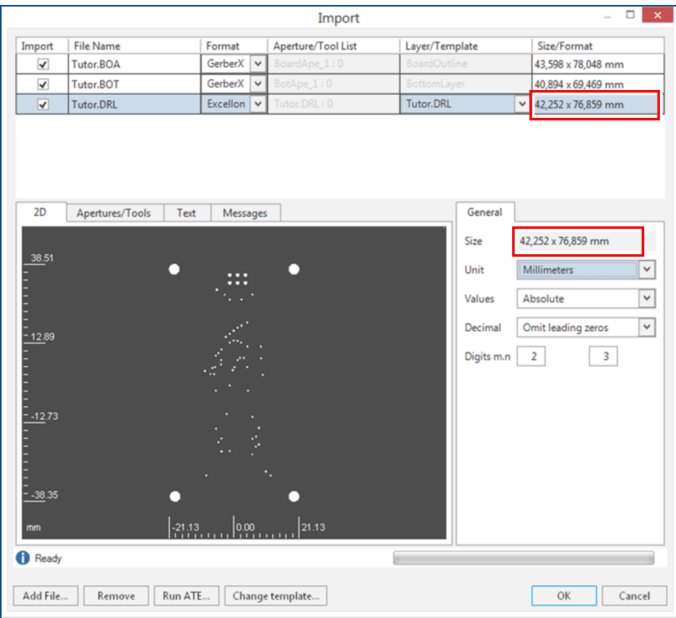


Fig. 185: **Changed layout dimensions**

☐ 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. 186: **Incorrect preview of the layout data**

4. Check the *Digits m.n* in the sub-tab *General* of the *Import* dialog.
5. Enter *3* in the *n* digit count.

☐ The dialog *Import* changes as follows:

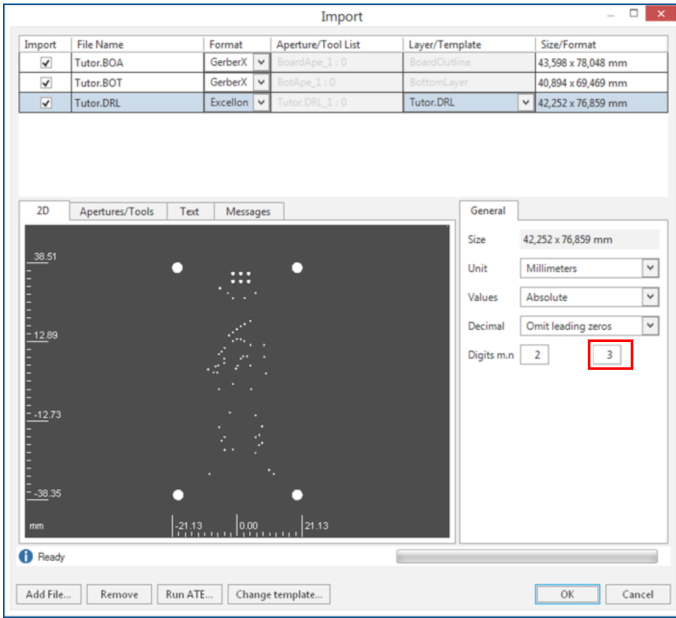


Fig. 187: **Correct preview of the layout data**

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

☑ The size and format have been set/corrected.

Viewing/modifying aperture properties

You can view the properties of the apertures contained in the files and modify them according to your needs.

1. In the dialog *Import*, select the file whose apertures you want to view (in this example the file *Tutor.BOA* is selected).

☐ The dialog *Import* changes as follows:

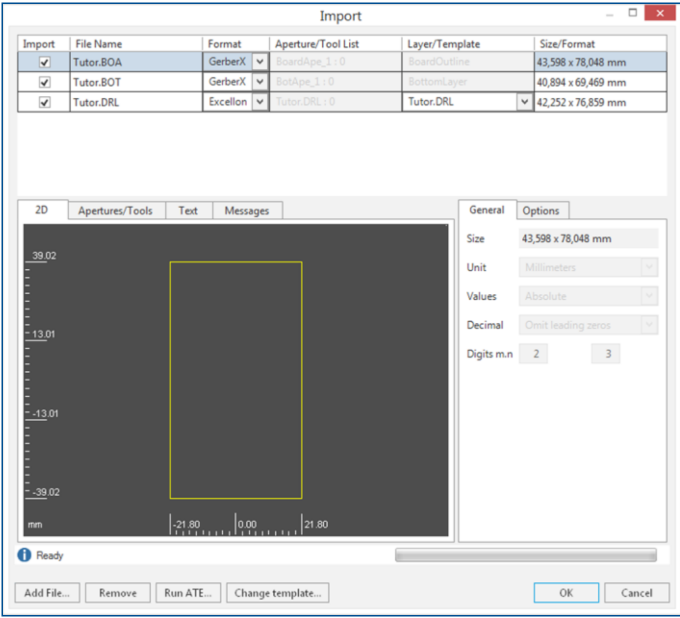


Fig. 188: **File *Tutor.BOA* selected**

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

☐ The dialog *Import* changes as follows:

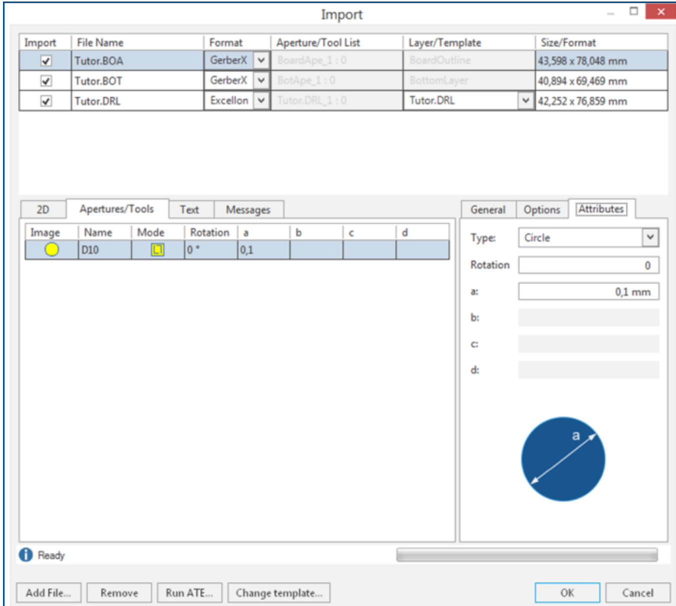


Fig. 189: **Tab *Apertures/Tools* and sub-tab *Attributes***

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:

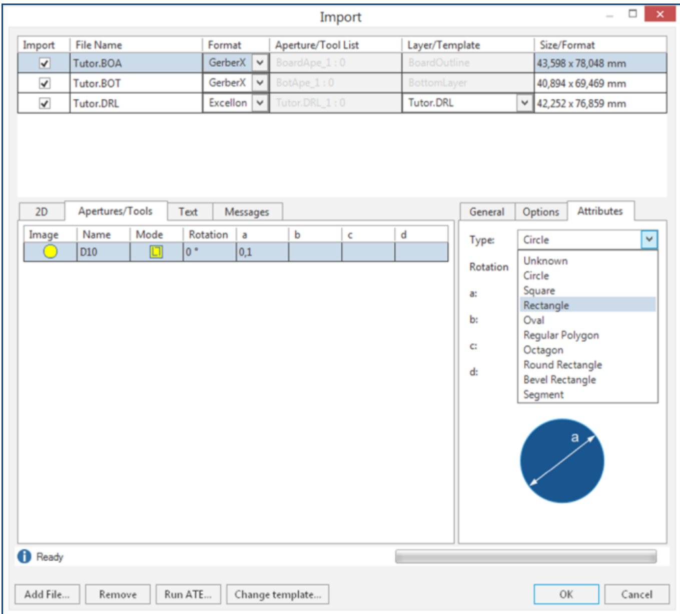


Fig. 190: **Selecting the type of aperture**

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

☐ In the sub-tab *Attributes*, a preview of the modified aperture is displayed:

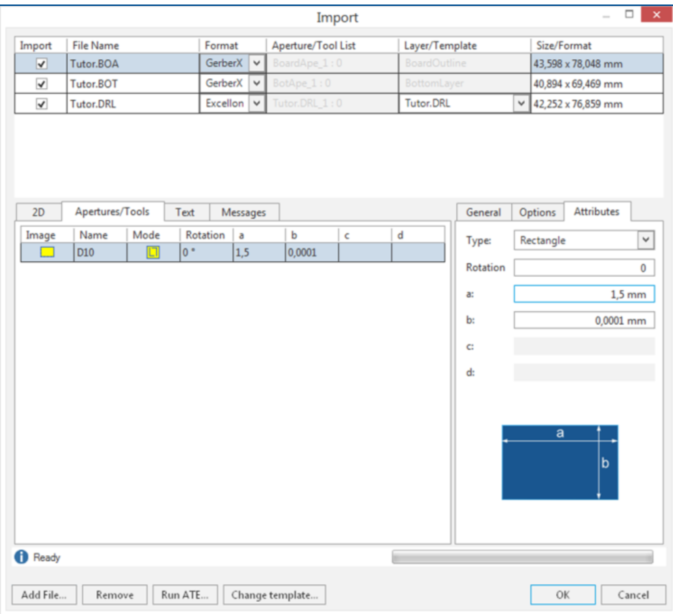


Fig. 191: **Preview of the modified aperture**

☑ The aperture properties have been viewed/modified.

The Gerber file format has an input field for the layer name. There are two ways to use the layer name for import. Either you define in 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:

You can view the layer name defined in the Gerber file in the tab *Text* of the dialog *Import* (in this example the file *Tutor.BOA* which contains the layer name *BoardOutline* is used):

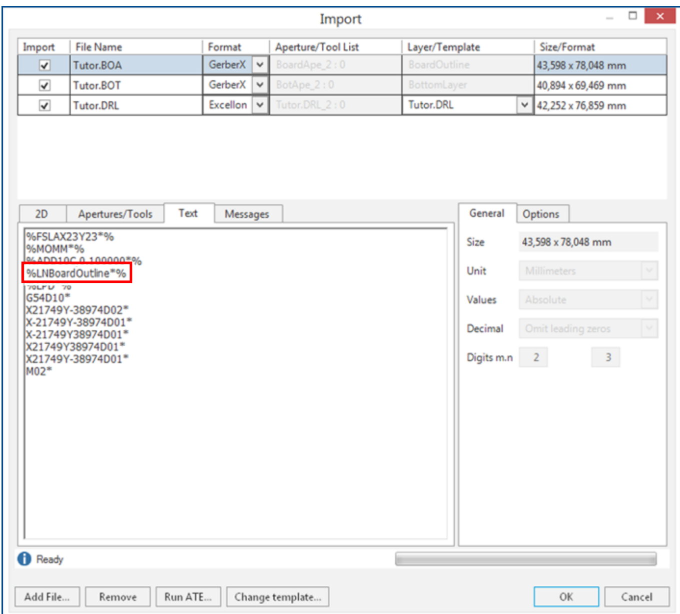


Fig. 192: Layer name in the Tutor.BOA file

Using layer names from the Gerber file for import

1. Select the files to be imported (in this example *Tutor.BOA*, *Tutor.BOT* and *Tutor.DRL*).
2. Click on [Open].

☐ The following dialog is displayed:

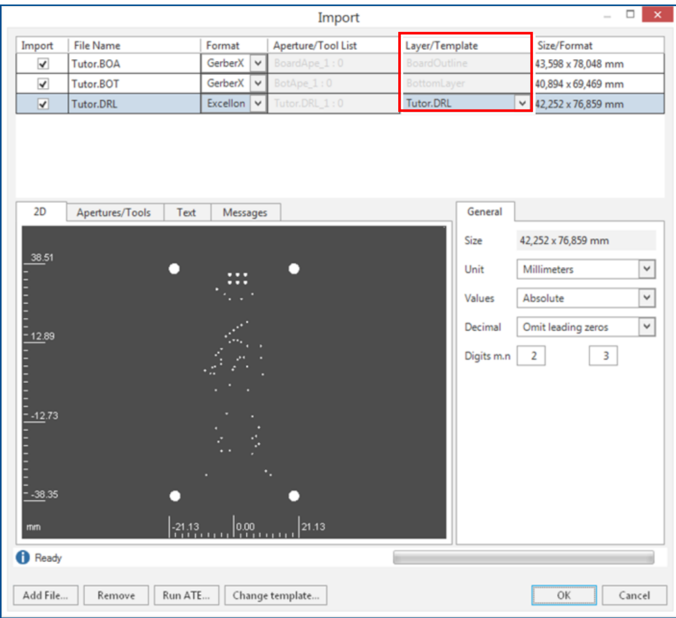


Fig. 193: Layer names not used in the Gerber file

The layer names defined in the Gerber file **are not used** and **not displayed** in the column *Layer/Template*:

3. Select one of the GerberX files (in this example the file *Tutor.BOA* is selected) and switch to the sub-tab *Options*.
4. Activate the following check boxes:
   - *Use layer name*
   - *Apply to all Gerber files*

- The dialog changes as follows:

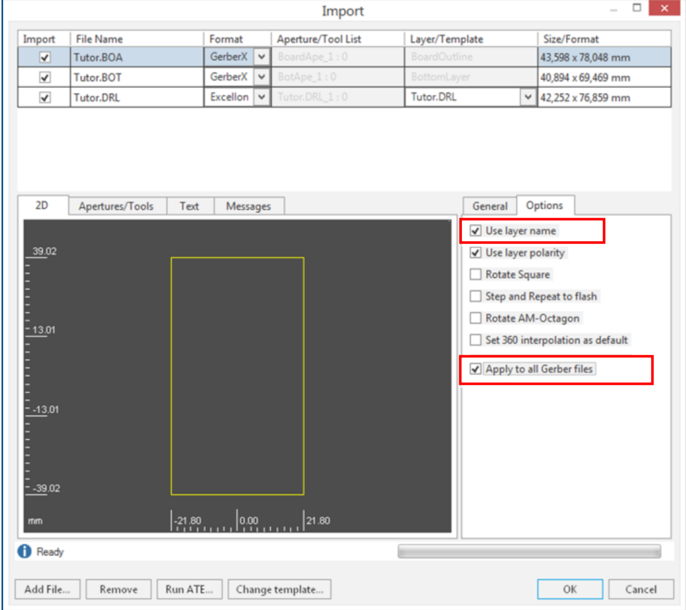


Fig. 194: Activated check boxes

- The layer names defined in the Gerber files are now displayed in the column *Layer/Template*:

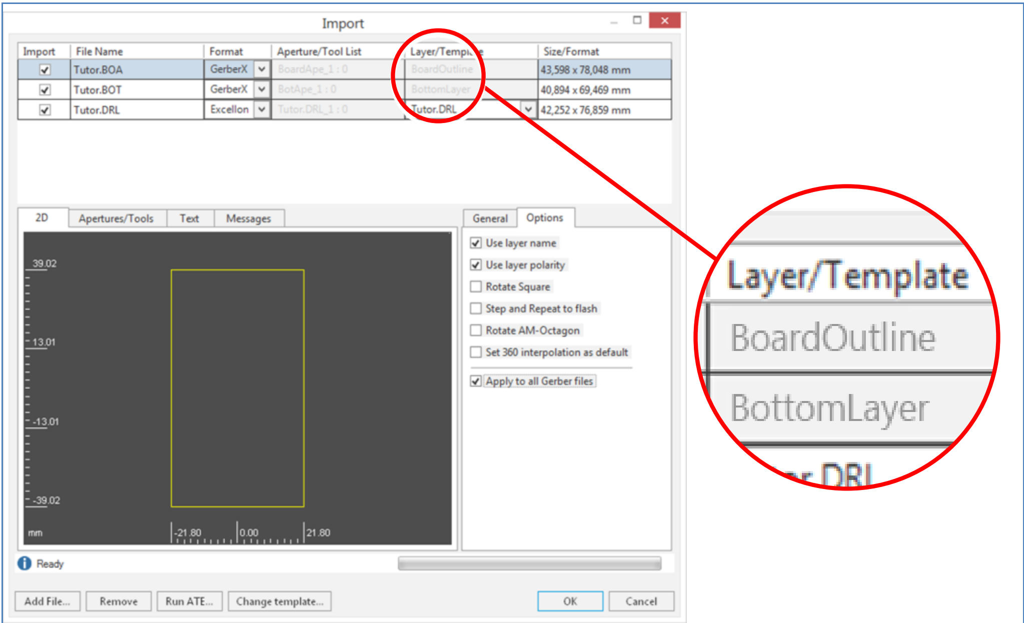


Fig. 195: Layer names from the Gerber file used

☑ 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 the option *Gerber Import*.

☐ The dialog changes as follows:

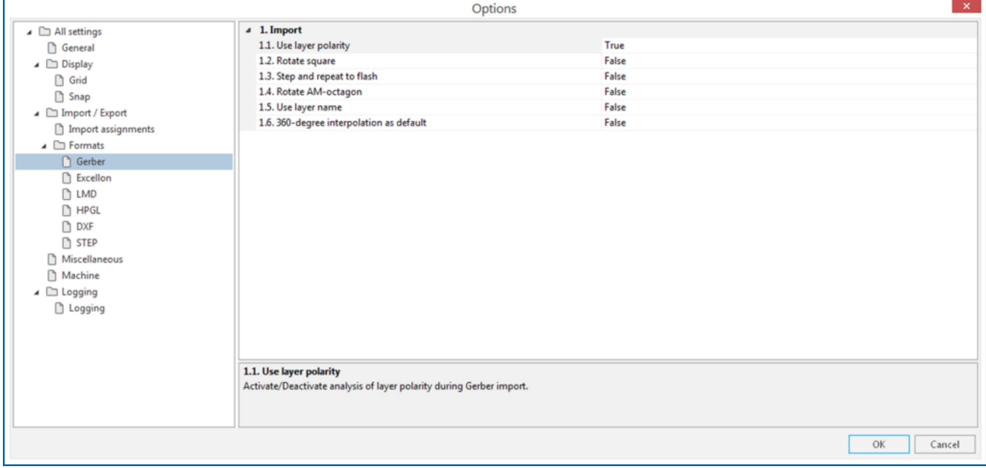


Fig. 196: **Dialog *Options* with *Gerber* import options**

3. Under *Use layer name* select the value *True*:

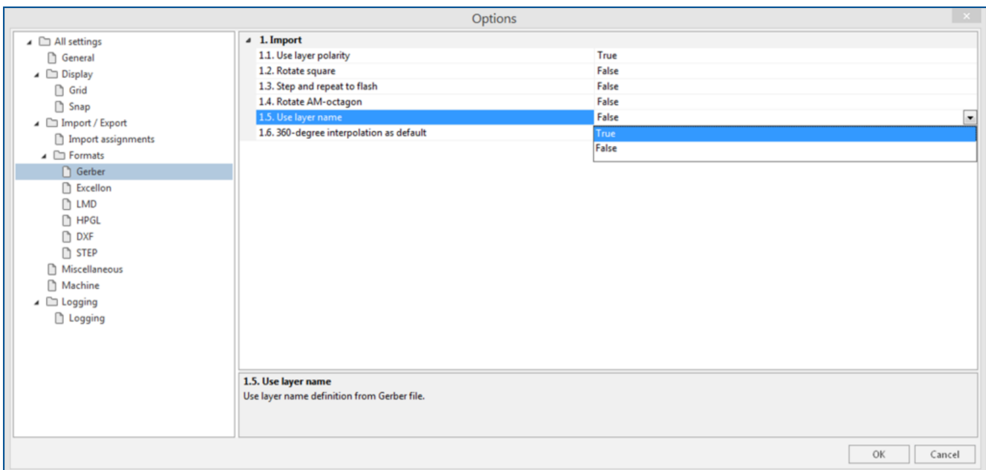


Fig. 197: **Selecting the value *True* for *Use layer name***

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 wish to use the layer names contained in the files.

4. Click on [OK].

☐ The option for the default use of the contained layer name has been activated.

5. Select the files you wish to import (in this example *Tutor.BOA*, *Tutor.BOT* and *Tutor.DRL*).
6. Click on [Open].

☐ The layer names defined in the Gerber files are displayed in the column *Layer/Template*:

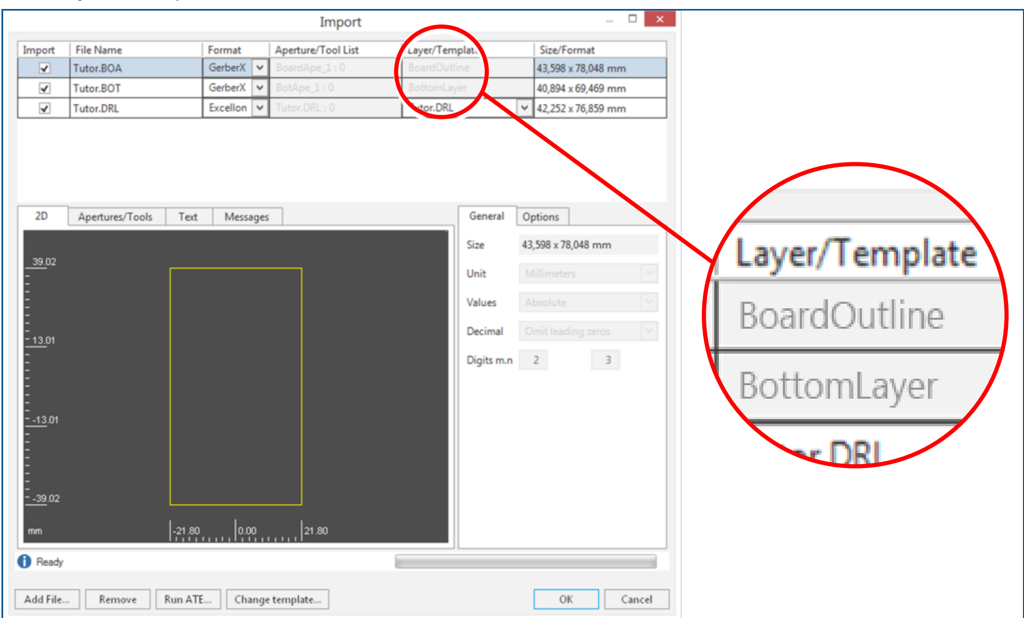


Fig. 198: **Layer names defined in the Gerber files displayed**

☑ The layer names from the Gerber file have been used as default.

The Gerber and Excellon files have been processed in CircuitPro PL 3.0.

6 Appendix

This chapter contains navigation elements of the document.

6.1 List of figures

Fig. 1: Fiducial in the *Camera* view 13
Fig. 2: Inaccurate position of the layout 13
Fig. 3: Matching laser head and fiducial position in the *Processing* view 14
Fig. 4: Matching position of crosshair to board edge 15
Fig. 5: *Camera* view of a recognized fiducial 16
Fig. 6: Message *Spiral search active* 16
Fig. 7: *Manual fiducial definition* 17
Fig. 8: Visible fiducial 17
Fig. 9: *Camera* view of a poor quality fiducial 18
Fig. 10: *Camera* view of the matched position of poor quality fiducial 18
Fig. 11: Assembly of the press mold and the materials 20
Fig. 12: A 4-layer PCB structure with all hole and via types 22
Fig. 13: Annular ring 23
Fig. 14: A through hole with external and internal annular rings 24
Fig. 15: Staggered vias and minimum offset 24
Fig. 16: Stacked vias 25
Fig. 17: Incorrect inner layer fill in the *Layout* view 25
Fig. 18: Correct inner layer fill in the *Layout* view 26
Fig. 19: Desktop icon CircuitPro PL 3.0 29
Fig. 20: Message *System connection* 29
Fig. 21: Dialog *Signal light check* 29
Fig. 22: Dialog *New document* 29
Fig. 23: Dialog *New Document* 30
Fig. 24: Dialog *Material settings* 30
Fig. 25: Dialog *Open* 31
Fig. 26: Dialog *Import* and assigned layers 32
Fig. 27: Imported data in the *CAM* view 32
Fig. 28: Context menu *Step and repeat* 33
Fig. 29: Dialog *Step and repeat* 33
Fig. 30: Dialog *Step and repeat* after entering values 33
Fig. 31: *Layout* view of the multiplied layout 34
Fig. 32: Inserted fiducials 35
Fig. 33: Context menu *Compute all* 36
Fig. 34: Message *Computation results* 36
Fig. 35: Message *Laser warm up phase in progress…* 37
Fig. 36: Message *Processing phase: Mount material* 37
Fig. 37: Dialog *Material settings* 37
Fig. 38: Dialog *Placement* 38
Fig. 39: Message *Processing phase: Flip material* 38
Fig. 40: Message *Manual fiducial definition* 38
Fig. 41: Message *Board production finished* 39
Fig. 42: Desktop icon CircuitPro PM 2.3 41
Fig. 43: Dialog *Connection steps* 41
Fig. 44: Dialog *New document* 41
Fig. 45: Dialog *Open* 42
Fig. 46: Dialog *Import* and assigned layers 42
Fig. 47: Imported data in the *CAM view* 43
Fig. 48: *CAM view* after selecting holes 44
Fig. 49: Context menu *Assign objects* to *DrillUnplated* layer 44
Fig. 50: Context menu *Step & Repeat* 45
Fig. 51: Dialog *Step & Repeat* 45

Fig. 52: Dialog *Step & Repeat* after entering values 46
Fig. 53: *CAM view* of the multiplied layout 46
Fig. 54: Dialog *Create fiducial* 47
Fig. 55: *CAM view* after inserting fiducials 47
Fig. 56: Dialog *Technology* 49
Fig. 57: Message *Computation Results* 49
Fig. 58: *Layout* view of the imported layout 52
Fig. 59: Message *Processing phase: Process drilling* 53
Fig. 60: Message *Processing phase: Galvanic through-hole plating* 53
Fig. 61: Message *Processing phase: Mount material bottom* 53
Fig. 62: Dialog *Open* 55
Fig. 63: *CAM view* of the opened project 55
Fig. 64: *Machining view* of the opened project 56
Fig. 65: Drop-down list of the processing phases 56
Fig. 66: *Machining view* of drilling and contour-routing data 57
Fig. 67: Dialog *Import* and assigned layers 65
Fig. 68: *Layout* view of the imported data 66
Fig. 69: Context menu *Move object* 66
Fig. 70: Copy of the layout 67
Fig. 71: Layout after specifying the reference point 67
Fig. 72: Desired position of the copied layout 68
Fig. 73: Layout moved to the desired location 68
Fig. 74: Message *Processing phase: Mount core* 69
Fig. 75: Positioning holes in the right front corner of the core 70
Fig. 76: Message *Processing phase: Dismount material* 70
Fig. 77: Group *Processing* and list of processing phases 71
Fig. 78: Context menu *Processing* 72
Fig. 79: Message *Processing phase: Mount laminate* 72
Fig. 80: Message *Processing phase: Mount material top* 74
Fig. 81: Dialog *Import* and assigned layers 77
Fig. 82: Imported data in the *CAM view* 78
Fig. 83: Pane *Layers* 79
Fig. 84: *CAM view* after selecting index layers and toolpaths 79
Fig. 85: Pane *Layers* and highlighted auxiliary layers 80
Fig. 86: List of toolpaths 81
Fig. 87: Highlighted toolpaths 81
Fig. 88: Dialog *Import* and assigned layers 90
Fig. 89: Pane *Layers* 91
Fig. 90: Pane *Layers* and selected auxiliary layers 92
Fig. 91: Dialog *Import* and assigned layers 101
Fig. 92: Imported data in the *CAM view* 102
Fig. 93: *Technology Dialog* in the process of creating toolpaths for buried vias ... 103
Fig. 94: Dialog *Convert to Toolpath* 103
Fig. 95: Settings for buried vias 103
Fig. 96: Pane *Toolpath* 104
Fig. 97: Pane *Toolpath* and list of nodes 104
Fig. 98: List of toolpaths 105
Fig. 99: New sequence of toolpaths 105
Fig. 100: Pane *Layers* 106
Fig. 101: Pane *Layers* and highlighted auxiliary layers. 106
Fig. 102: List of toolpaths 107
Fig. 103: Highlighted toolpaths 107
Fig. 104: Message *Processing phase: Process cutting on a ProtoMat* 111
Fig. 105: Message *Processing phase: Mount core* 111
Fig. 106: Positioning holes in the right rear corner of the core 111
Fig. 107: List of processing phases 114
Fig. 108: Context menu *Processing* 115
Fig. 109: *Layout* view of the imported data 121

Fig. 110: Examples of a phase, work package and task 124
Fig. 111: Dialog *Import* and assigned layers 126
Fig. 112: Warning *Import* 126
Fig. 113: *Layout* view of the imported data 127
Fig. 114: Drop-down list layers 127
Fig. 115: Specifying the center point 128
Fig. 116: Specifying the radius 128
Fig. 117: Specifying the next center point 129
Fig. 118: Prepreg positioning holes 129
Fig. 119: Phase *Top Side (Standard)* 130
Fig. 120: Work packages *Drilling* and Cutting 131
Fig. 121: Tasks selected 131
Fig. 122: Context menu *Configure…* 132
Fig. 123: Dialog *Laser contour routing* 133
Fig. 124: Contour routing method *Basic* 133
Fig. 125: Work packages of the phase *Prepare Core* 134
Fig. 126: Context menu *Add task* 134
Fig. 127: Adding a task 135
Fig. 128: Added tasks 135
Fig. 129: Context menu *Compute toolpaths* 136
Fig. 130: Warning about deleting the toolpaths 138
Fig. 131: Scan fields displayed in the *Layout* view 138
Fig. 132: Context menu | *Hide scan fields* 139
Fig. 133: Context menu *Select by layer* 139
Fig. 134: *Specify scaling coefficient* 140
Fig. 135: Drop-down list *Individually* 141
Fig. 136: Preview of the scaled prepreg positioning holes 141
Fig. 137: List of layers 142
Fig. 138: Context menu *Select by layer* 143
Fig. 139: Filled rectangle 144
Fig. 140: Input field *Offset* value 144
Fig. 141: Preview of the expanded rectangle 145
Fig. 142: *Layout* view after expanding the rectangle 145
Fig. 143: Dialog *Laser contour routing* 146
Fig. 144: Pane *Layers* 147
Fig. 145: Warning exceeding the limits 148
Fig. 146: Dialog *Delete objects* 149
Fig. 147: Tasks in the work package *Prepare_Laminate* 150
Fig. 148: Setting the phase *Top Side (Standard)* to *Mirrored* 153
Fig. 149: Layout with computed and displayed scan fields 156
Fig. 150: Selected layers 157
Fig. 151: Preview of the pasted layers 158
Fig. 152: *Layout* view of pasted objects 158
Fig. 153: Dialog *Import* and assigned layer 163
Fig. 154: *Layout* view of the imported data 163
Fig. 155: Elements selected 164
Fig. 156: Combined open path 164
Fig. 157: Assigning combined path to the layer *BoardOutline* 165
Fig. 158: Board outline assigned to layer 165
Fig. 159: Top layer paths selected 166
Fig. 160: Holes selected 166
Fig. 161: Holes assigned to layer 167
Fig. 162: List of layers 168
Fig. 163: Filled objects 168
Fig. 164: Dialog *Import* 171
Fig. 165: Deselected files 172
Fig. 166: Dialog *Import* and *Format* undefined 173
Fig. 167: Selecting the appropriate file format 173

Fig. 168: Appropriate file format selected 174
Fig. 169: Unassigned layer 175
Fig. 170: Selecting the target layer 176
Fig. 171: Selected target layer 176
Fig. 172: Layout view of the imported files 177
Fig. 173: Unassigned layer 178
Fig. 174: Warning *Import* 178
Fig. 175: Warning about new layer created 178
Fig. 176: Layout view of the imported files 179
Fig. 177: List of layers 179
Fig. 178: Assigning objects 180
Fig. 179: Assigned drill holes 180
Fig. 180: *Import assignments* 181
Fig. 181: New import assignments 181
Fig. 182: Example of an import assignment 181
Fig. 183: Large layout size 182
Fig. 184: Selecting the appropriate unit 183
Fig. 185: Changed layout dimensions 183
Fig. 186: Incorrect preview of the layout data 184
Fig. 187: Correct preview of the layout data 184
Fig. 188: File *Tutor.BOA* selected 185
Fig. 189: Tab *Apertures/Tools* and sub-tab *Attributes* 185
Fig. 190: Selecting the type of aperture 186
Fig. 191: Preview of the modified aperture 186
Fig. 192: Layer name in the Tutor.BOA file 187
Fig. 193: Layer names not used in the Gerber file 188
Fig. 194: Activated check boxes 189
Fig. 195: Layer names from the Gerber file used 189
Fig. 196: Dialog *Options* with *Gerber* import options 190
Fig. 197: Selecting the value *True* for *Use layer name* 190
Fig. 198: Layer names defined in the Gerber files displayed 191

6.2 List of tables

Table 1: Tips for positioning and fiducial reading 12
Table 2: Required files/layers 22
Table 3: Size range according to drill hole type 23
Table 4: Required LPKF systems 28
Table 5: Usual copper thicknesses and tool libraries 53
Table 6: Required LPKF systems 64
Table 7: Table columns 171