

FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 1 of 12

1 Purpose

This SOP describes the standard workflow for fabricating a basic **double-sided printed circuit board (PCB)** using the LPKF ProtoLaser system controlled by CircuitPro software. Please note that the portions of the PCB procedure described here can be used to cut, drill, etch, ablate, or otherwise structure many materials for many purposes beyond PCBs.

The PCB procedure covers:

- Two copper layers: top and bottom
- No plated through-holes
- Standard FR4 laminate
- Laser drilling, copper isolation, and contour cutting

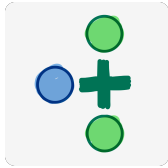
2 Scope

This SOP is intended for trained users in the Rodgers RLE Laboratory producing prototype circuit boards for research use.

This SOP does not cover multilayer boards, plated vias, solder mask, unusual laminates, flex circuits, or advanced process optimization. Please see Rodgers lab staff for advice beyond this SOP.

3 Required Materials and Equipment

- LPKF ProtoLaser system
- CircuitPro workstation
- FR4 double-sided copper clad board
- PCB manufacturing files:
 - Top copper Gerber



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 2 of 12

- Bottom copper Gerber
- Excellon drill file
- Mechanical outline / board contour file
- /textless 1.5 mm diameter poker for fiducial plugs
- Isopropyl alcohol and Kimwipes

4 Safety

- Only trained personnel may operate the ProtoLaser.
- Keep the enclosure closed during processing.
- Never bypass interlocks or defeat safety systems.
- If panels are off during maintenance, do not reach inside the machine.
- Stop operation immediately if excessive smoke, ignition, alarms, or abnormal machine behavior occur.
- Clean dust and debris after use.

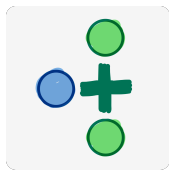
Report faults or abnormal operation to lab staff before continuing.

5 File Preparation

Export PCB manufacturing files from the CAD package.

Minimum required files:

- Top copper Gerber
- Bottom copper Gerber
- Drill file



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 3 of 12

- Board outline

Before import, verify:

- Units are correct and consistent.
- Board dimensions match the intended design.
- Bottom layer orientation follows the CAD export convention.
- Unused, duplicate, or drawing-only layers are removed.
- Drill holes are non-plated unless the design intentionally requires otherwise.

6 CircuitPro Workflow

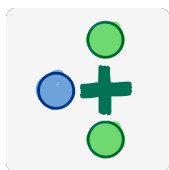
CircuitPro uses a guided workflow. For this SOP, use the following sequence:

New → Material → Import → Layout → Toolpaths → Processing

7 Procedure

7.1 Start New Job

1. Launch CircuitPro.
2. Select **New**.
3. Load the "Double-sided, Protolaser U4 (or R4), no through-hole plating" ProtoLaser PCB processing template.
4. Continue to the **Material** step.



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 4 of 12

7.2 Select Material

Choose a material profile matching the actual board stock:

- FR4 substrate
- Correct board thickness, 1.5 mm thickness
- Correct copper thickness, 18 μm thickness, top and bottom side
- Double-sided copper laminate with these dimensions is stored in the top left drawer of the bench behind you, as you face the Protolasers.
- Other materials must generally be supplied by the user

Use existing Material defaults whenever possible. When needed, it is best to make small changes from those defaults.

Do not modify existing Materials. Instead, duplicate the Material. A number will be automatically added to the end of the Material name. Change this number to your Kerberos ID, add other text if you like, and then modify the Material parameters and associated Tools to suit your needs. Here in the material tab, changes can only be made to the Material name, whether the Material is a laminate or not (more on that later), the thicknesses of the core and foils, and some notes to help you and other users make use of the Material settings.

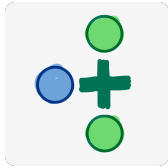
Later in the procedure you will see how to use and modify the Tools that are grouped under this Material.

7.3 Import PCB Files

In the **Import** step, add the Gerber and drill files.

Recommended layer mapping:

Input File	CircuitPro Layer / Function
Top copper Gerber	TopLayer
Bottom copper Gerber	BottomLayer
Excellon drill PTH file	DrillPlated
Excellon drill NPTH file	DrillUnplated
Edge cuts file	BoardOutline



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 5 of 12

After import, verify:

- Correct scale
- Correct board outline
- Drill holes centered on pads
- Top and bottom copper layers registered correctly
- No missing traces, pads, or apertures

7.4 Layout Review

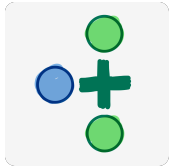
In the **Layout** view, inspect the design before toolpath generation.

Check:

- Fine-pitch footprints
- Narrow traces and gaps
- Drill-to-pad centering
- Clearance to the board edge
- Correct top and bottom side orientation

Correct file or layer errors before continuing. Then add fiducials to enable top to bottom alignment:

- Click on Automatic fiducial creation to open the Fiducials auto set dialogue.
- Deselect one of the four automatic fiducials by clicking on one of the numbers from 1 to 4.
- Enter 0 for the Distance, which will be corrected to 0.5 mm.
- Click OK and three fiducials will be automatically placed 0.5 mm outside of the BoardOutline data.



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 6 of 12

7.5 Review Workpackages

Proceed to the **Workflow** and enable at least the primary workpackages required for this basic PCB process:

1. In the drop-down, check that "Bottom side" is selected.
2. Review the Drill Fiducials work package.
3. Review the Drill Plated through-holes.
4. Review Structure Bottom.
5. Click "Top side."
6. Verify "Top side" is selected in the drop-down.
7. Click on "Read fiducials" and uncheck the box next to Scale.
8. Review Structure Top.
9. Review Drill Unplated through-holes.
10. Review Final contour cut

7.6 Generate Toolpaths

Toolpaths step.

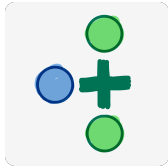
Select **Compute All** to calculate toolpaths.

Review the calculated paths before processing. If toolpaths appear incorrect, return to the layout and layer assignments before continuing.

8 Machine Setup

8.1 Prepare Work Surface

1. Clean the machine bed.



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 7 of 12

2. Remove previous fixtures, scraps, or debris.
3. Confirm dust extraction is available and active.
4. Confirm the work area is clear.

8.2 Mount PCB Blank

Place the FR4 copper-clad laminate flat on the machine bed.

Requirements:

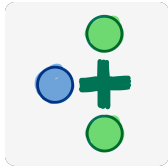
- Board is flat with no rocking or bowing.
- No debris is between the board.
- Copper surface is clean and dry.
- No debris is trapped beneath the board.

Poor material mounting can cause poor focus, incomplete copper removal, ragged edges, or side-to-side misregistration.

8.3 Focus / Z Height

Verify the Material thickness setting.

- Move to a location on the board with prominent features, such as an edge.
- Click on the Camera tab.
- Place the edge close to the edge of the ROI so that the camera view is primarily filled with the top copper surface.
- Make note of the initial stage Z position at the right of the screen.
- Click the AF button.
- Make note of the new Z position and subtract it from the initial Z position.



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 8 of 12

- Return to the Processing tab.
- Click on the Placement button, or right click in the copper-colored area of the Processing viewport.
- Make note of the "Material thickness" value in the Placement popup.
- If the difference between the initial and new Z positions is greater than 0.1 mm or less than -0.1 mm, then subtract that difference from the Material thickness value.
- Please note that if you have mounted your material on top of a glass slide or other secondary material, then the Material thickness value will be the total stack-up: the sum of the secondary material, any adhesive layer such as gel-pack, and your sample material. In short, the Material thickness value actually means "Move the laser focus this amount above the High Precision White Air Permeable backing material" which sits on the vacuum table.

Incorrect focus may cause:

- Incomplete copper isolation
- Excessively wide cuts
- Burned substrate
- Rough contour edges
- Reduced repeatability

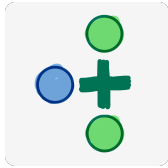
Refocus after changing material thickness, remounting the board, or flipping the substrate.

9 Processing Sequence

9.1 Process Top Side

Run the top-side process sequence:

1. Drill holes or registration holes as required.



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 9 of 12

2. Process top copper isolation.
3. Perform any optional partial contour operations, if used.

Observe the first features before allowing the full job to continue unattended.

9.2 Flip Board

After the top side is complete, flip the board using the approved alignment method:

- Registration holes
- Fixture edge
- Fiducials

Before starting the bottom side, verify:

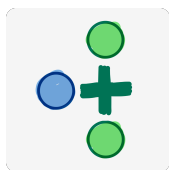
- Correct orientation
- Board seated flat
- No debris under the board
- Alignment features match expected locations

9.3 Process Bottom Side

Run the bottom-side process sequence:

1. Process bottom copper isolation.
2. Complete final contour cut.

Do not remove the board until all required machining steps are complete.



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 10 of 12

10 Post Processing and Inspection

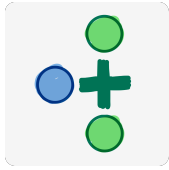
1. Remove the board carefully.
2. Vacuum dust and loose debris.
3. Clean copper with IPA.
4. Deburr board edges if needed.
5. Inspect the board under magnification.

Recommended checks:

- Continuity of important nets
- No shorts between adjacent traces
- Hole alignment acceptable
- Top-to-bottom registration acceptable
- Contour cut complete

11 Troubleshooting

Issue	Corrective Action
Copper remains after isolation	Recheck focus, material flatness, and process settings.
Burned or darkened substrate	Reduce energy input if authorized; check focus and extraction.
Top and bottom sides misaligned	Recheck flip orientation, registration holes, and seating.
Ragged contour edge	Improve hold-down, clean debris, and verify focus.
Wrong board scale	Check CAD export units and CircuitPro import settings.
Missing features	Verify imported Gerber layers and aperture interpretation.



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 11 of 12

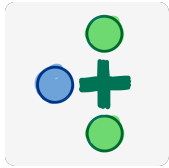
12 Documentation

Record the following in the lab logbook:

- User name
- Date
- Project name
- Material type and thickness
- Copper thickness, if known
- Any modified settings
- Pass / fail result
- Notes on defects or rework

13 References

1. LPKF Laser & Electronics AG. *CircuitPro PL 3.5 Online Help*.
https://tjr-lab.mit.edu/wp-content/uploads/2023/06/Protolaser-CircuitPro-PL-3.5_OH_V1.1_ENG.pdf
2. LPKF Laser & Electronics AG. *How-To Guides ProtoLaser*.
<https://tjr-lab.mit.edu/wp-content/uploads/2023/06/4207-how-to-guides-protolaser-u4-s4-r4-v4.pdf>
3. LPKF Laser & Electronics AG. *ProtoLaser: Optimizing Processing Quality*.
<https://tjr-lab.mit.edu/wp-content/uploads/2023/06/4506-protolaser-optimizing-processing-qua.pdf>



FABRICATION OF BASIC DOUBLE-SIDED PCB USING LPKF PROTOLASER / CIRCUITPRO

Rev.: A

Print Date:
August 17, 2026

Page: 12 of 12

14 Revision History

Rev	Date	Description
A	August 17, 2026	Initial Rodgers template release.