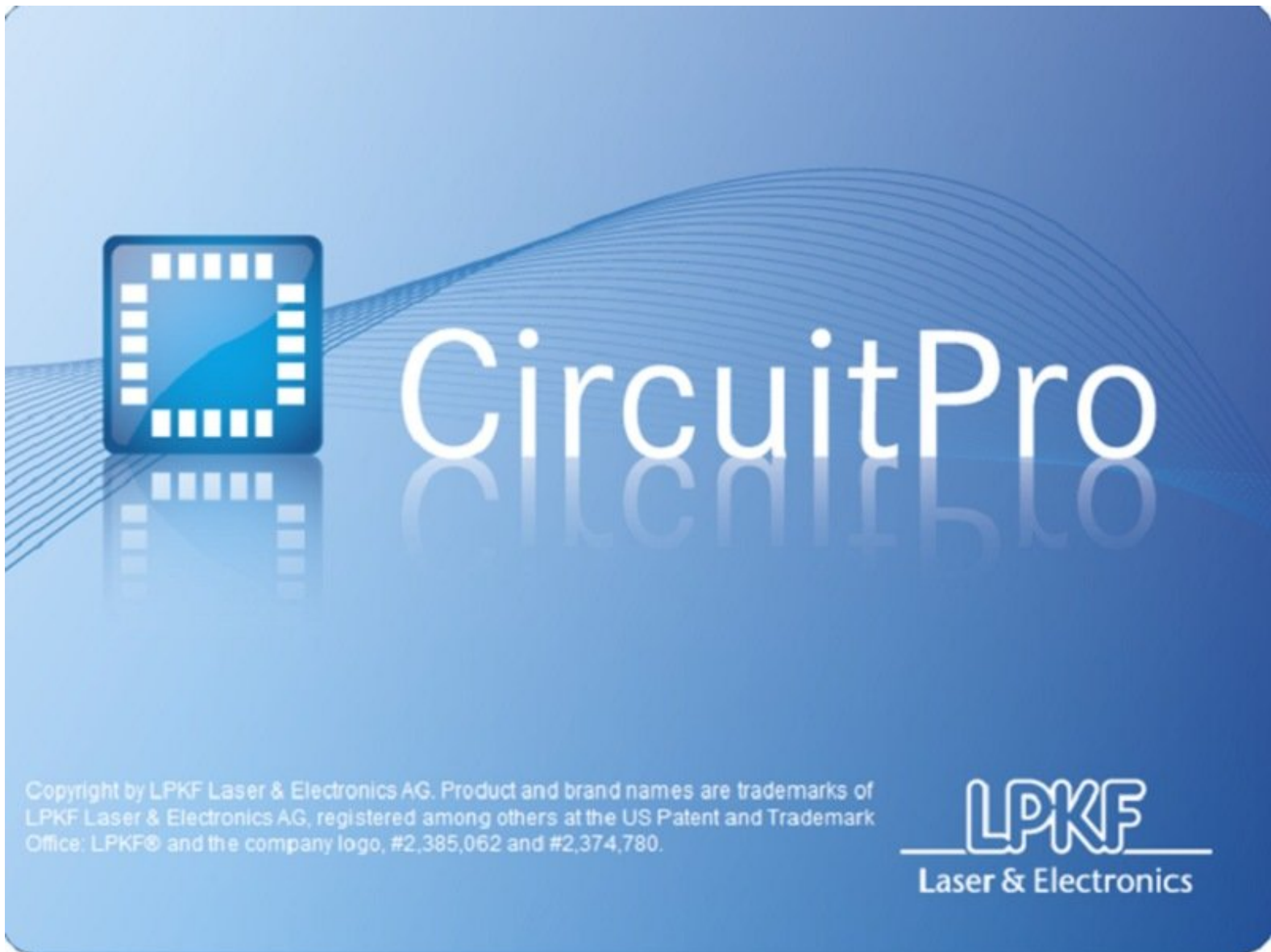




Setting Up Prints with CircuitPro

CircuitPro is the CAM software that is used to process and set up prints on the ProtoMat S64.

Written By: Cosimos Sivan Cendo

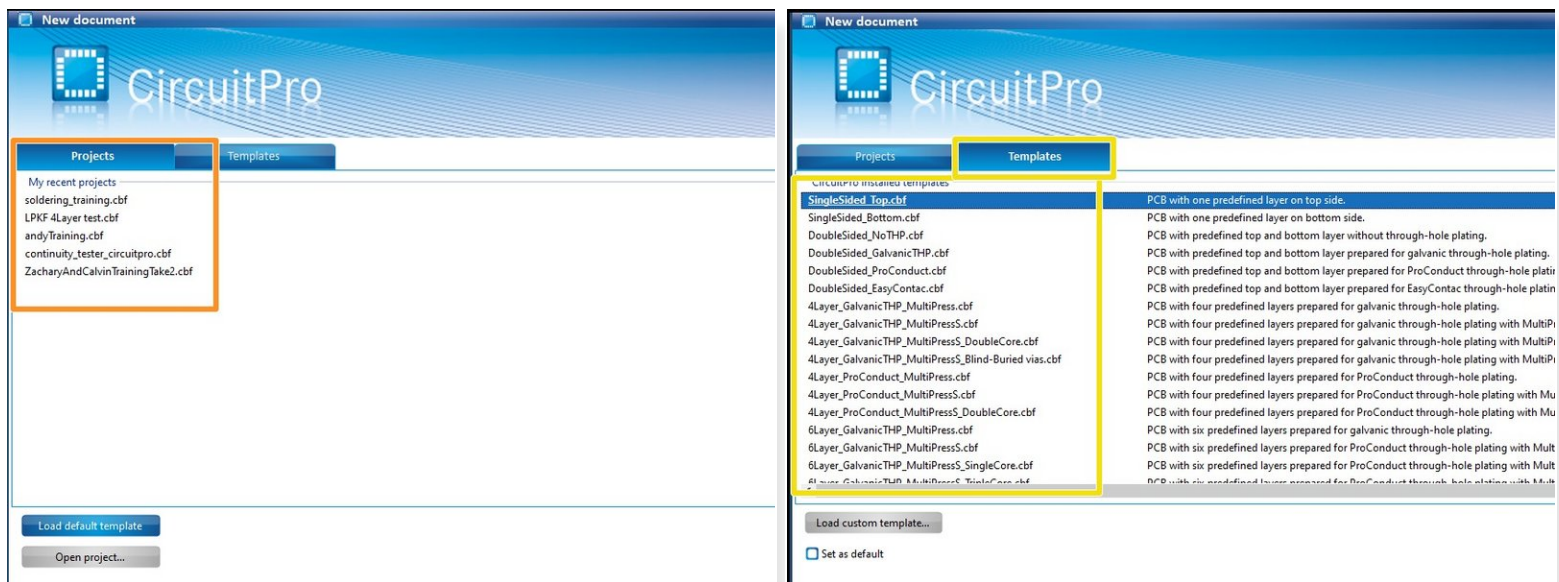


Step 1 — Intro



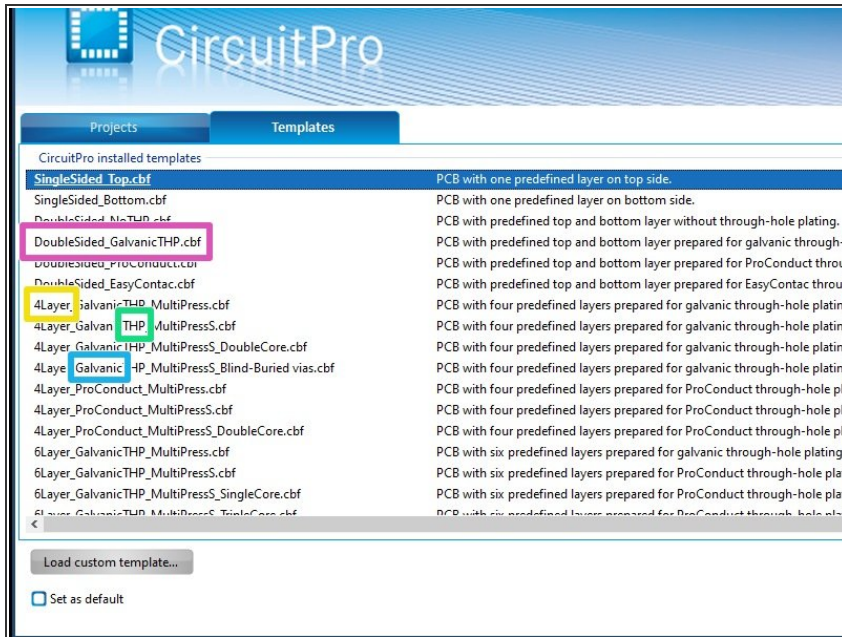
- Circuit Pro is the CAM software used to control the S64
- It is used to set up files for milling and send CNC commands to the S64

Step 2 — Initial Start



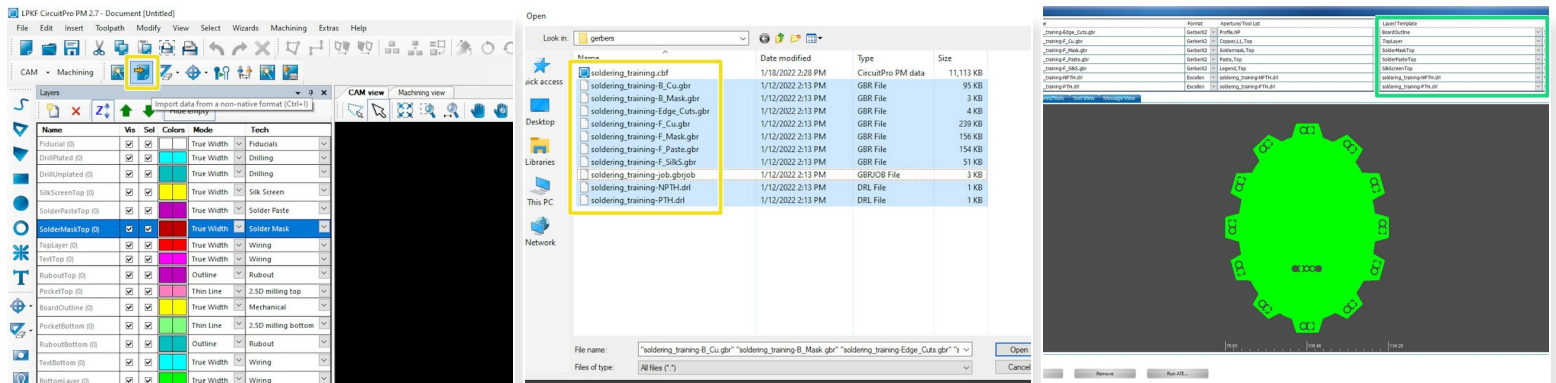
- On startup, the first screen will have 2 tabs: Projects and Templates
- **Projects** are used to resume work on a project
- **Templates** are used to start from scratch. They're essentially blank projects
- For the purpose of this guide, we will be starting from a template and saving it as a project so we can save progress as we work on it

Step 3 — Template Selection



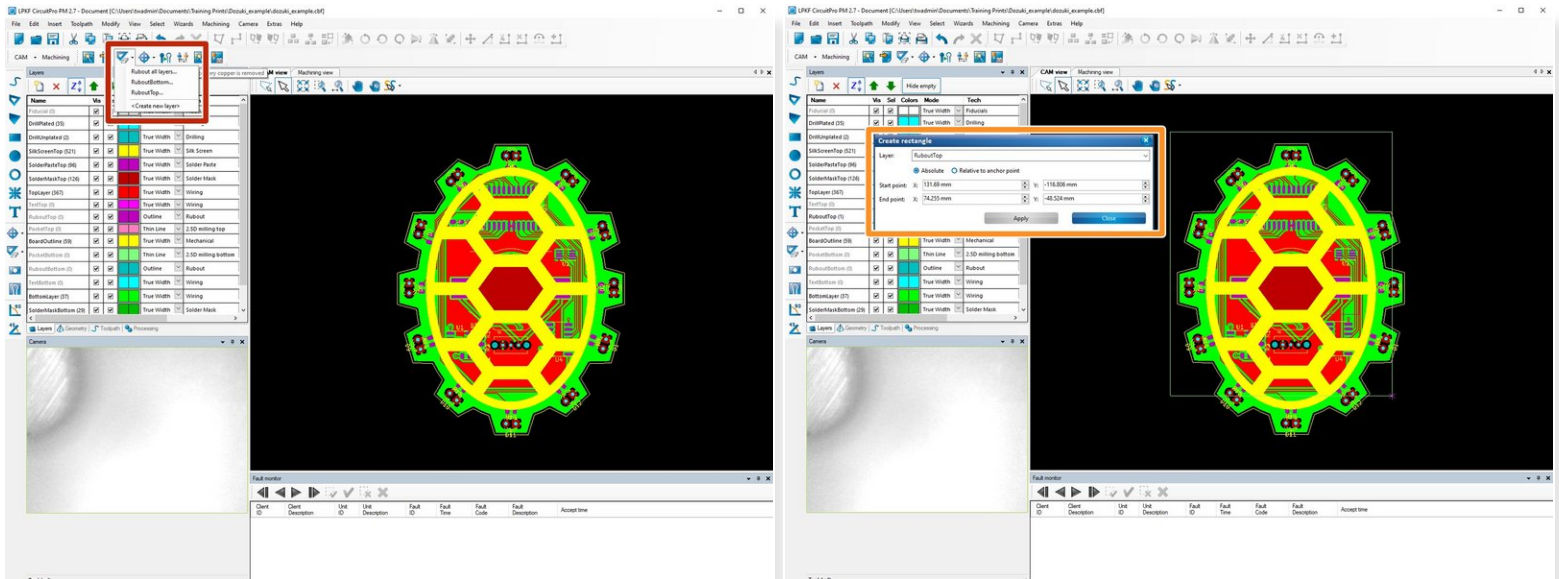
- The different templates detail the type of board and steps needed to make the board
- There's a long list of templates, each with different settings
 - Layers - the number of layers of the board. Most will be 2, but we can do up to 8 with the multipress
 - THP/No THP - Stand for Through Hole Plating. All vias require THP. If there are no vias, then no THP is required
 - Galvanic/ProConduct or EasyContac - the THP process. We have Galvanic, so make sure to select that
 - ① ProConduct and EasyContac are LPKF's other plating methods. Because we have the Contac S4, we use galvanic
- Click on a template to begin a project
 - ★ You'll probably want to use *DoubleSided_GalvanicTHP.cbf* most of the time

Step 4 — Importing Files



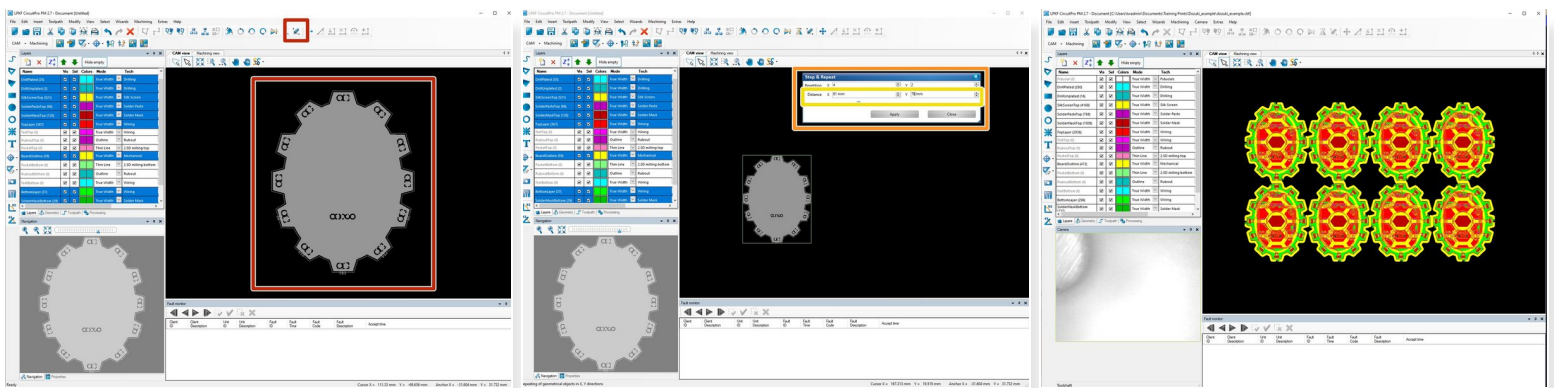
- The main type of PCB files are .gbr (gerber) and .drl (drill) files
- Each layer will have its own file
- Click the import button to import your gerber and drill files
- ❗ Disregard .gbrjob files when importing
- Make sure your files are assigned to the correct layers. CircuitPro will detect most by default but some must be assigned manually

Step 5 — (Optional) Adding rubout



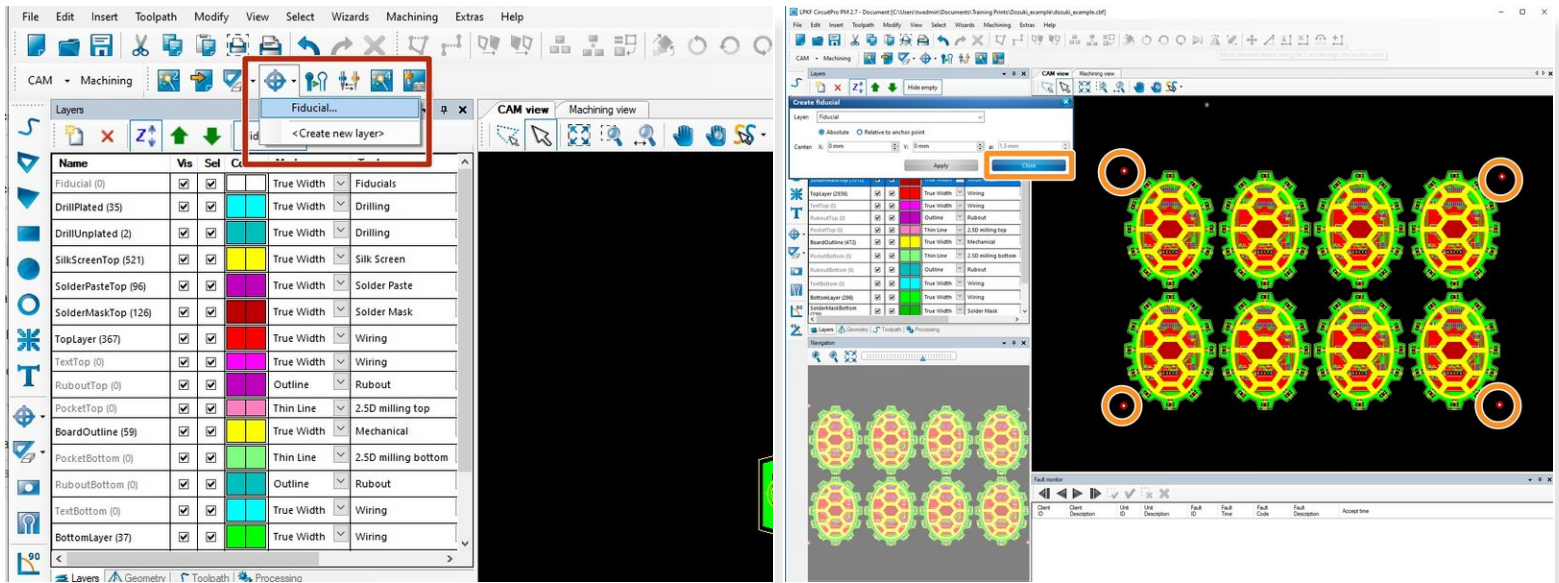
- You can add rubout to the top or bottom layers if you wish
- Draw a rectangle and all unused copper in that area will be removed

Step 6 — Duplicating the board



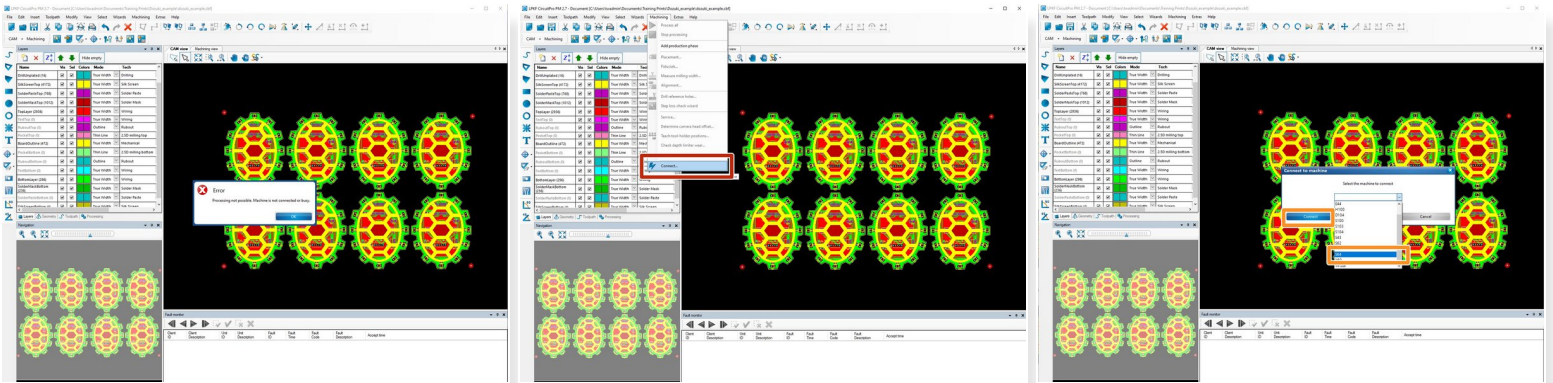
- Drag a box to select your entire board and click the duplicate button along the top row to create more copies
- The dialogue box will ask you how many copies you want and how much space should be between each copy
- By default, the edges of the copied boards will touch each other and this will cause problems when the board outline is being cut out. Add about 3mm to the x and y directions to give some space.
- ❗ Keep in mind that a sheet of FR4 is 9"x12" (228x305mm) so make sure your copied boards can fit

Step 7 — Adding fiducials



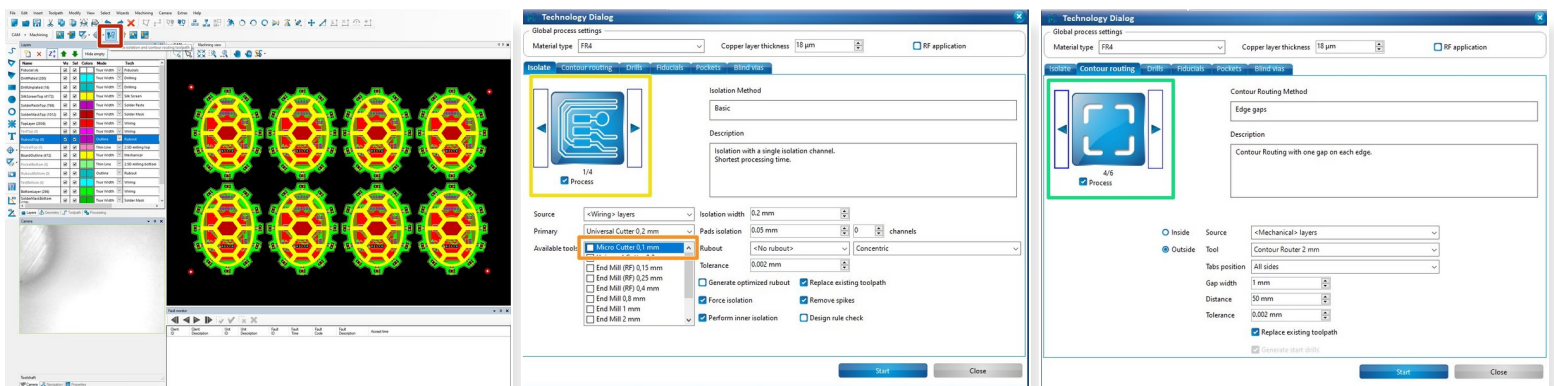
- Now that the board is copied we must add fiducials for the vision system to stay aligned
 - Click to make four fiducials in the corners of your design (these do not need to be precise)
- ⚠ Click **Close** when you're finished and NOT **Apply**; clicking **Apply** will create another fiducial at the xy-coordinates in the text box (probably 0,0)

Step 8 — (Optional) Connect machine



- i The ProtoMat may have connected automatically if you had it on when you started CircuitPro
- If not, then turn the machine on and navigate to **Machining->Connect**
- Select **S64** from the drop-down menu and click **Connect**

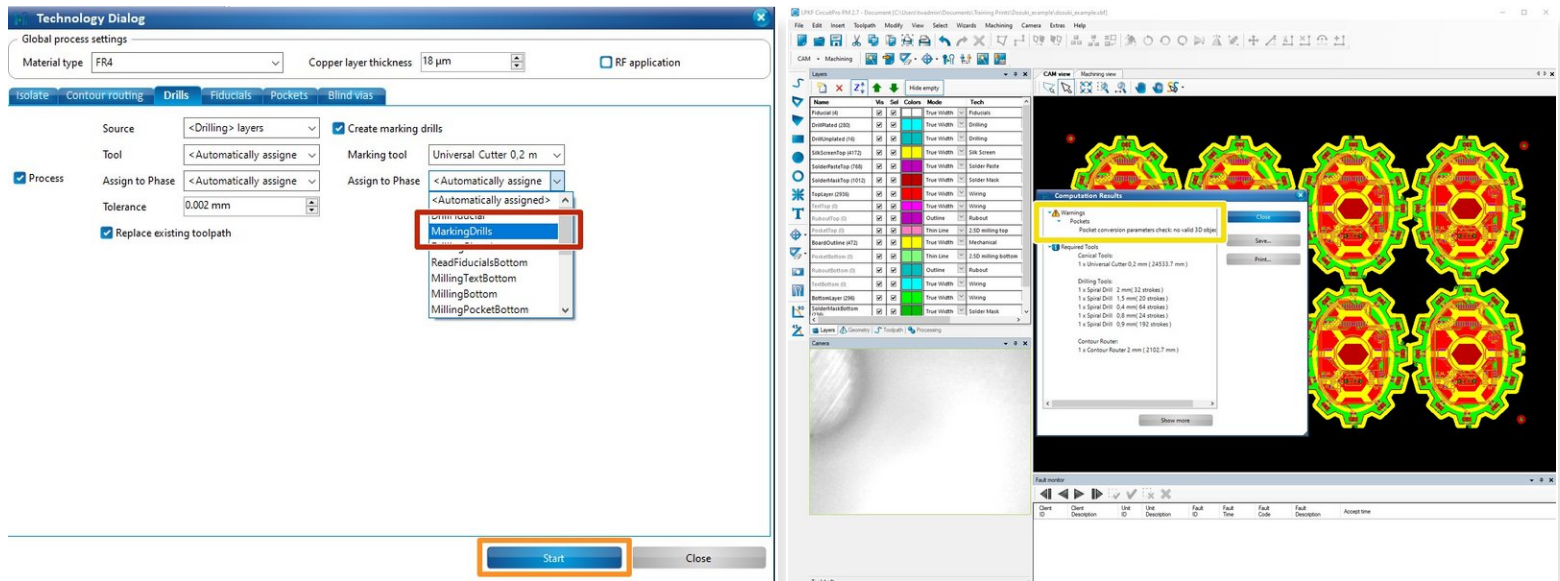
Step 9 — Generate toolpath



i Now we're going to generate the toolpath. This step is comparable to slicing for 3D printers - we are generating the code that the S64 will follow to route out the PCB

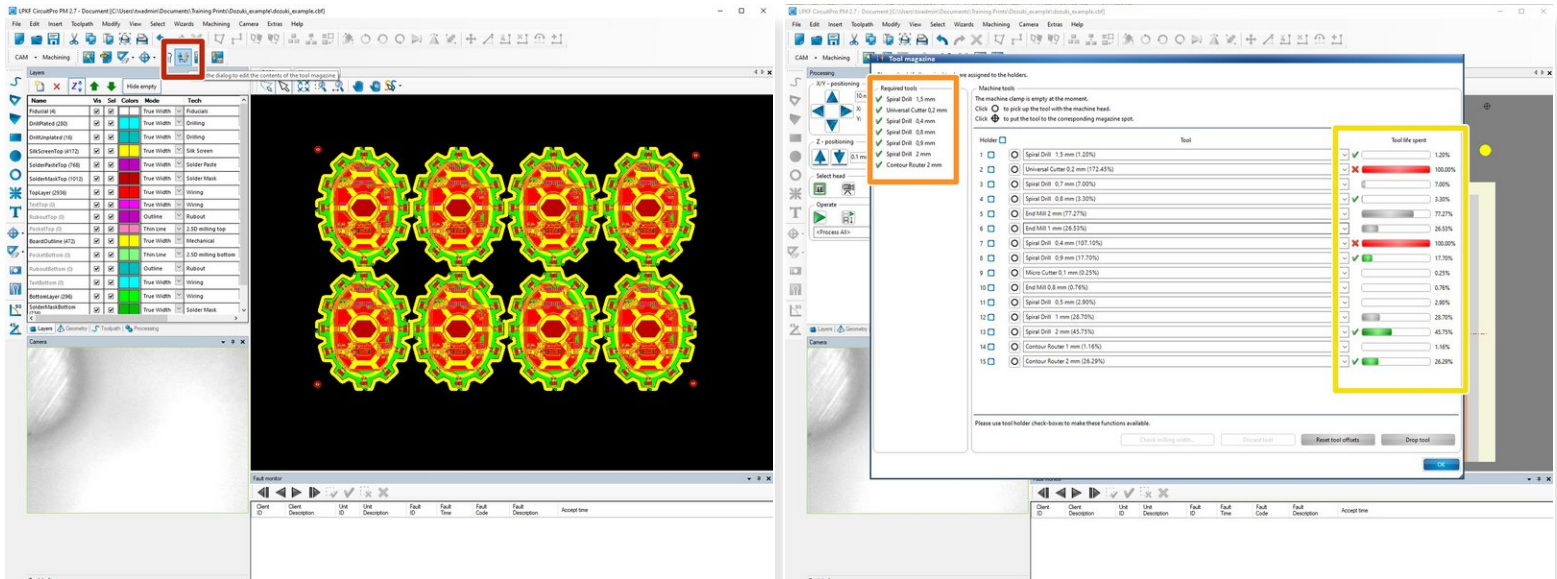
- Click the **Generate isolation and contour routing toolpath** button
- Uncheck *Micro Cutter 0,1 mm* from the list of available tools
 - You can change the **Isolation Method** if you want more space around your traces
- Choose whichever **Contour Routing Method** you want on the Contour routing tab

Step 10 — Marking drills



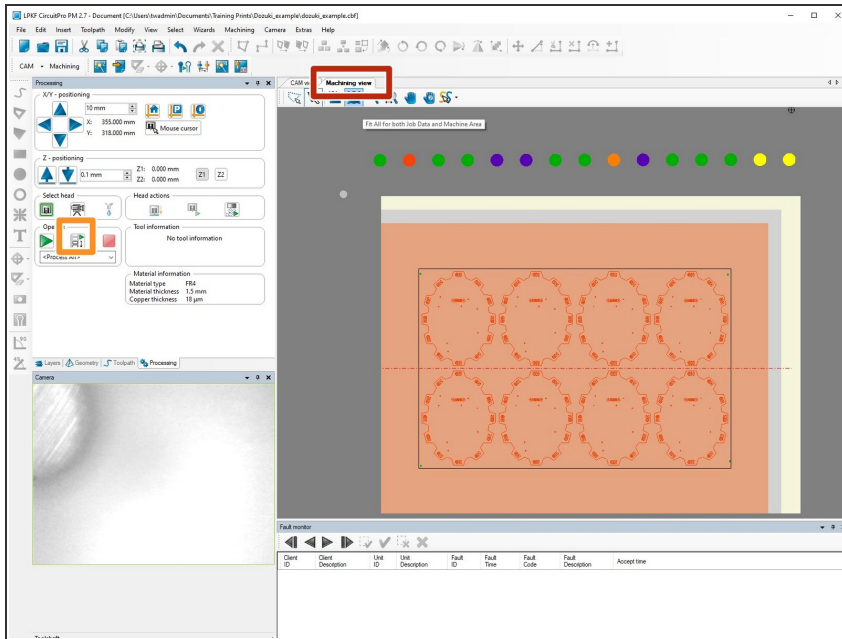
- Under **Marking tool** on the **Drills** tab, choose **MarkingDrills** for **Assign to Phase**
- Click **Start** to begin generating the toolpath
- Check that you don't have any warnings or errors before proceeding
- ① You can disregard the pocket conversion warning; it always appears

Step 11 — Tool magazine



- Check that all the required tools are in the machine
- Tools are listed on the left
- Tool life is listed on the right
- Use this window to put needed tools into the machine and replace spent ones

Step 12 — Start PCB process



- You can see where your board will be routed on the FR4 under **Machining view**
- Click the middle button above **<Process All>** to begin routing
- CircuitPro will guide you through the whole process