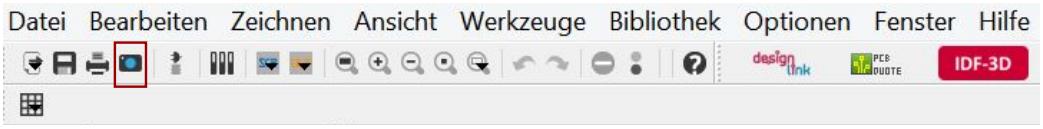
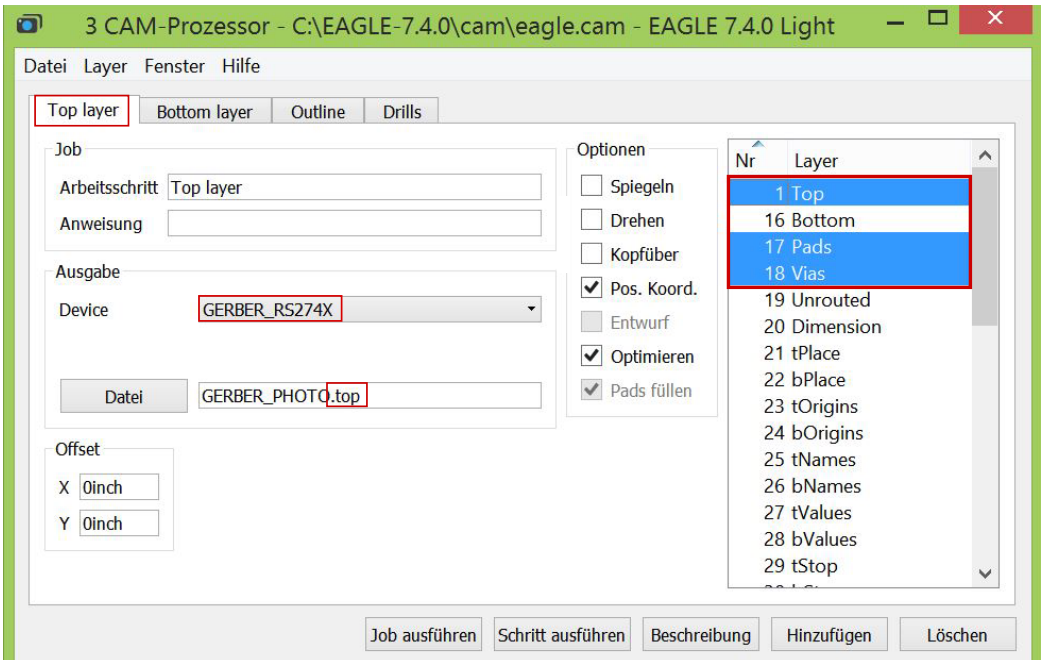
Step by Step Instruction

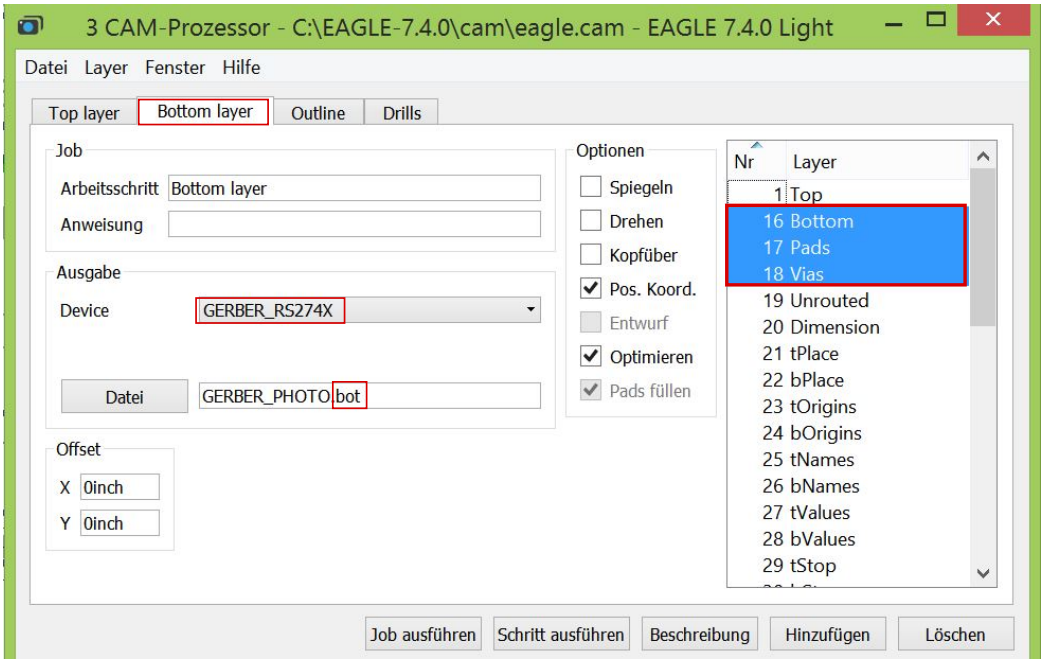
From Eagle file to Finished Board

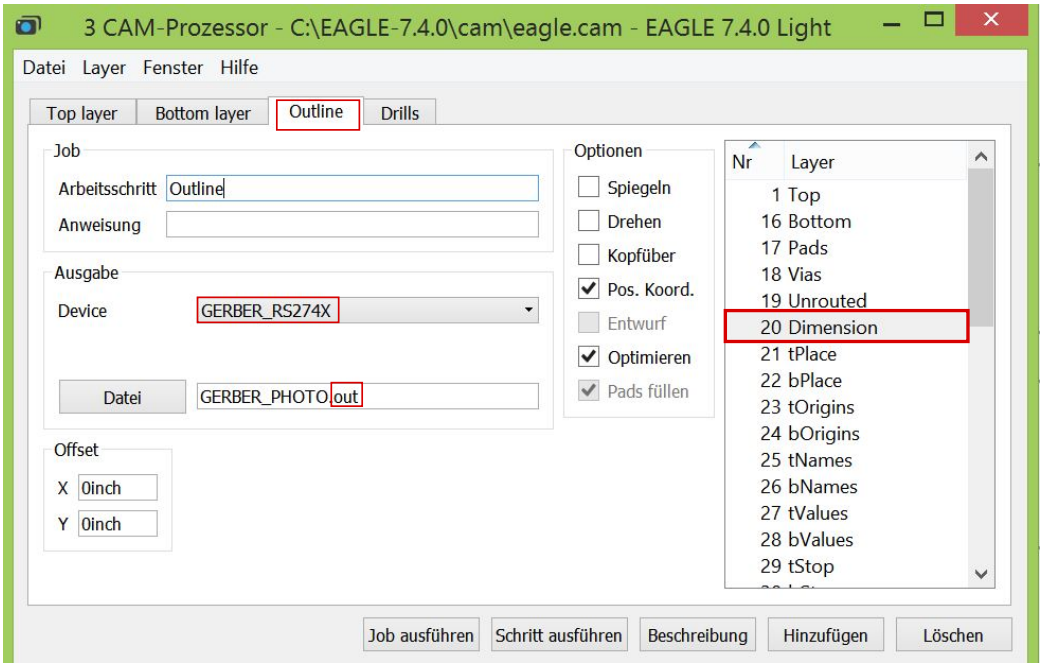
Export Eagle Files

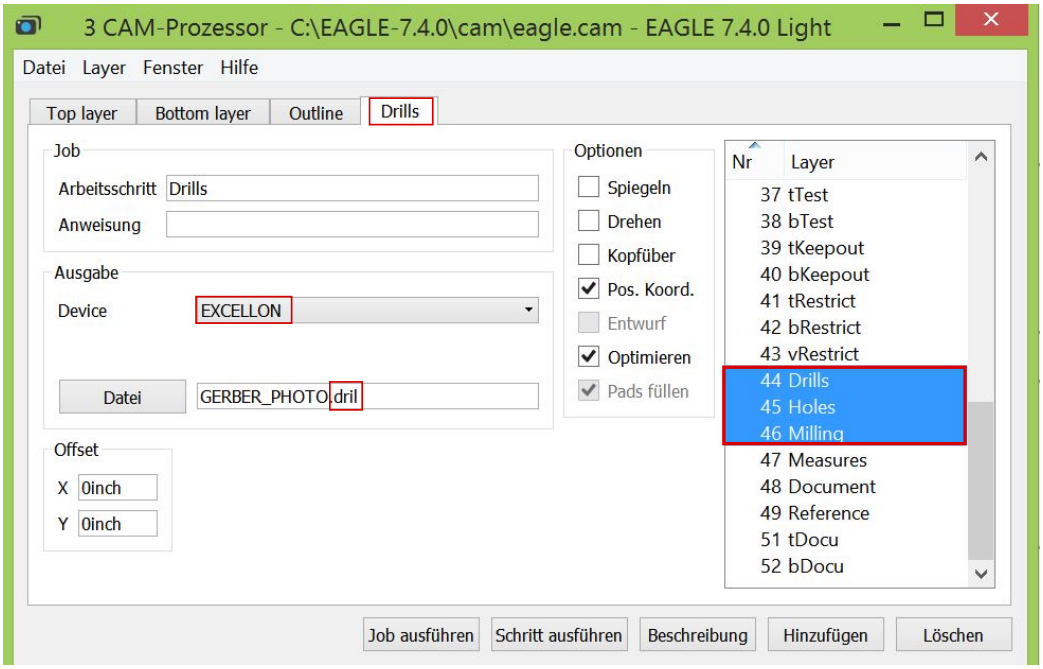
You need four files which you can export in *Eagle* with the *Datei* → *CAM Prozessor*. The relevant settings are shown in the figures below.






3 CAM-Prozessor - C:\EAGLE-7.4.0\cam\eagle.cam - EAGLE 7.4.0 Light
Datei Layer Fenster Hilfe
Top layer
Bottom layer
Outline
Drills
Job
Arbeitsschritt Bottom layer
Anweisung
Ausgabe
Device GERBER_RS274X
Datei GERBER_PHOTO bot
Offset
X 0inch
Y 0inch
Optionen
Spiegeln
Drehen
Kopfüber
Pos. Koord.
Entwurf
Optimieren
Pads füllen
Nr Layer
1 Top
16 Bottom
17 Pads
18 Vias
19 Unrouted
20 Dimension
21 tPlace
22 bPlace
23 tOrigins
24 bOrigins
25 tNames
26 bNames
27 tValues
28 bValues
29 tStop
Job ausführen
Schritt ausführen
Beschreibung
Hinzufügen
Löschen

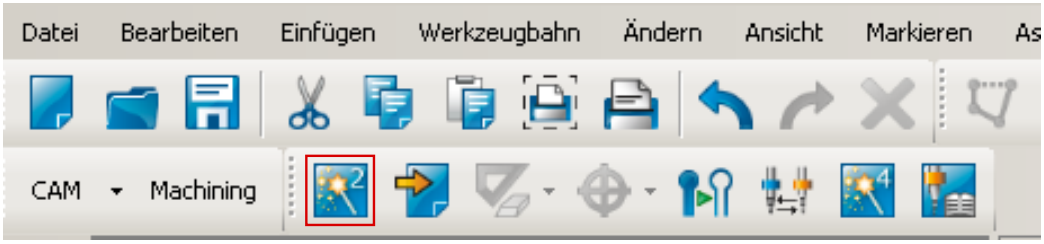

3 CAM-Prozessor - C:\EAGLE-7.4.0\cam\eagle.cam - EAGLE 7.4.0 Light
Datei Layer Fenster Hilfe
Top layer
Bottom layer
Outline
Drills
Job
Arbeitsschritt Outline
Anweisung
Ausgabe
Device GERBER_RS274X
Datei GERBER_PHOTO out
Offset
X 0inch
Y 0inch
Optionen
Spiegeln
Drehen
Kopfüber
Pos. Koord.
Entwurf
Optimieren
Pads füllen
Nr Layer
1 Top
16 Bottom
17 Pads
18 Vias
19 Unrouted
20 Dimension
21 tPlace
22 bPlace
23 tOrigins
24 bOrigins
25 tNames
26 bNames
27 tValues
28 bValues
29 tStop
Job ausführen
Schritt ausführen
Beschreibung
Hinzufügen
Löschen



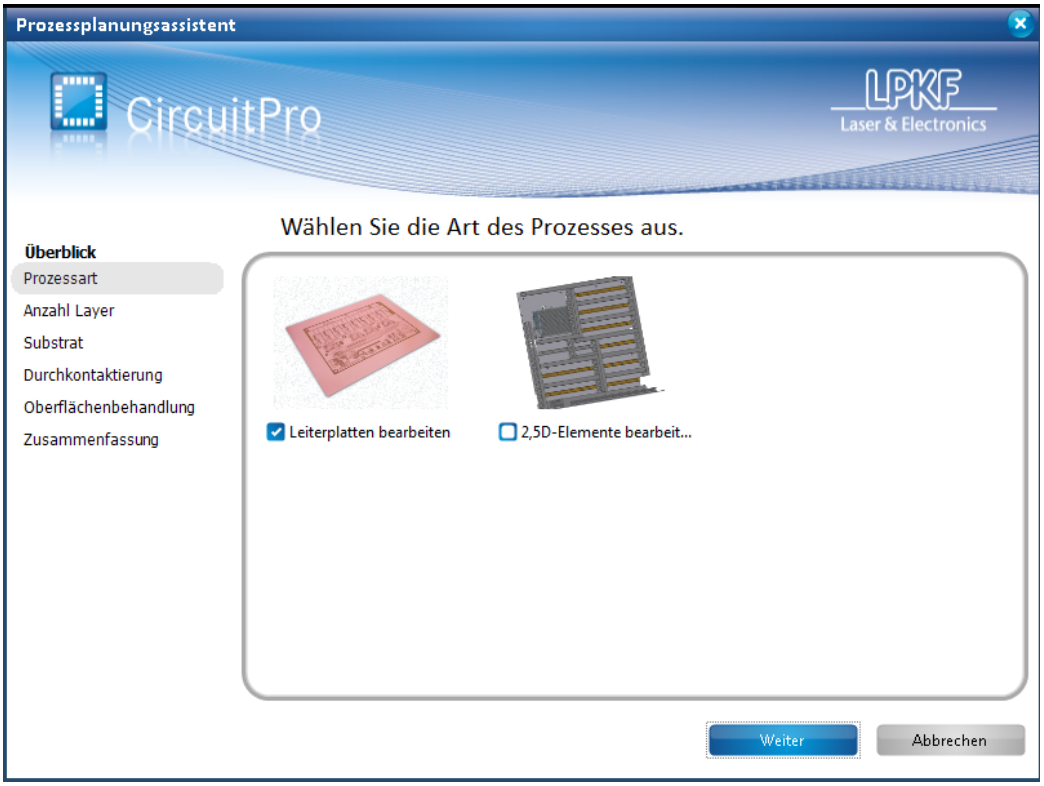
Mill

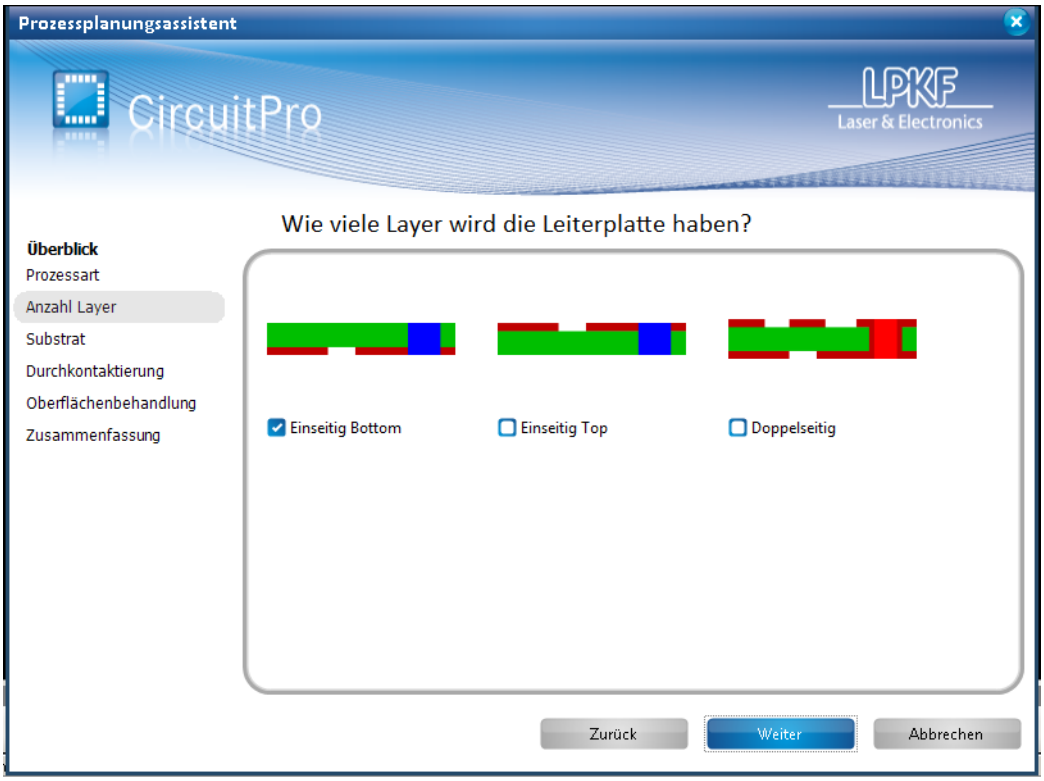
On the desktop of the provided PC is a link to the folder *Projektordner* where the files can be saved. Open the programme and choose any sample.

Step 1: Prozessplanungsassistent starten

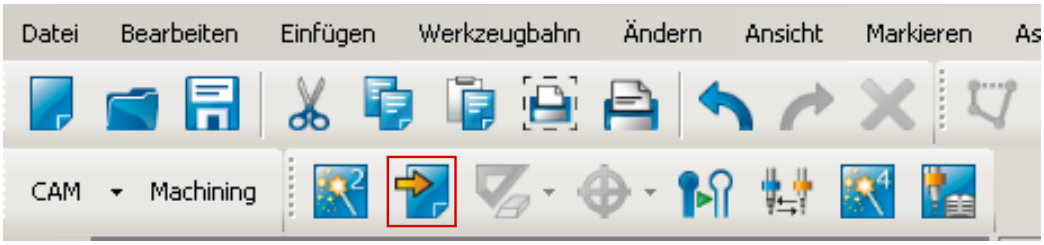


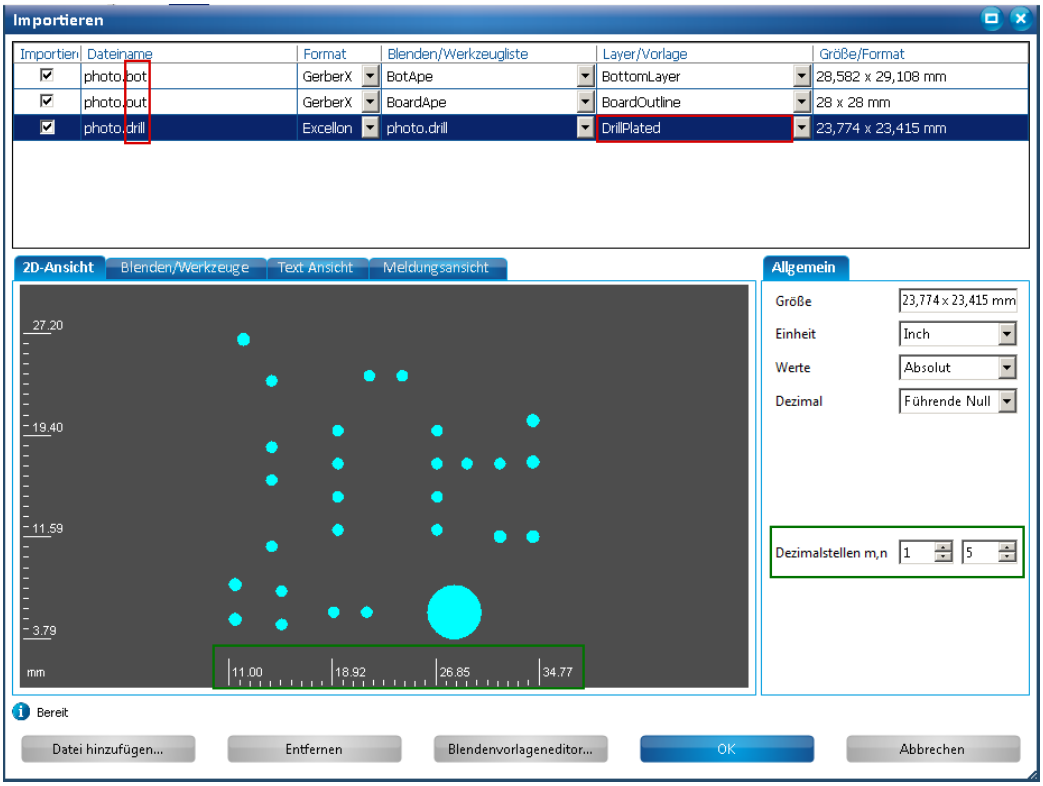
Choose *Leiterplatten bearbeiten* then the type of your designed borad and the used material *FR4/5*.

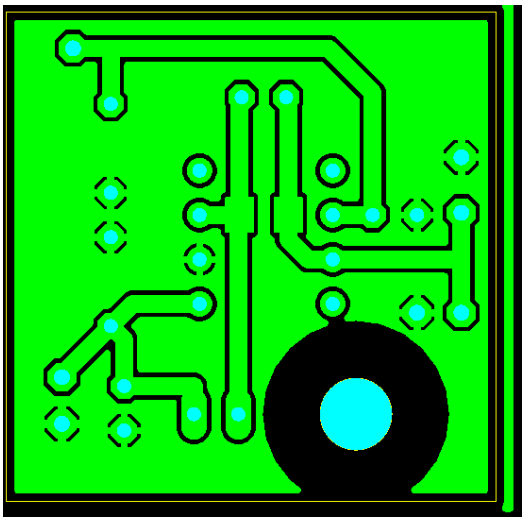

Prozessplanungsassistent
CircuitPro
LPKF
Laser & Electronics
Wählen Sie die Art des Prozesses aus.
Überblick
Prozessart
Anzahl Layer
Substrat
Durchkontaktierung
Oberflächenbehandlung
Zusammenfassung
Leiterplatten bearbeiten
2,5D-Elemente bearbeit...
Weiter
Abbrechen


Prozessplanungsassistent
CircuitPro
LPKF
Laser & Electronics
Wie viele Layer wird die Leiterplatte haben?
Überblick
Prozessart
Anzahl Layer
Substrat
Durchkontaktierung
Oberflächenbehandlung
Zusammenfassung
Einseitig Bottom
Einseitig Top
Doppelseitig
Zurück
Weiter
Abbrechen

Step 2: Datei aus Fremdvormat importieren



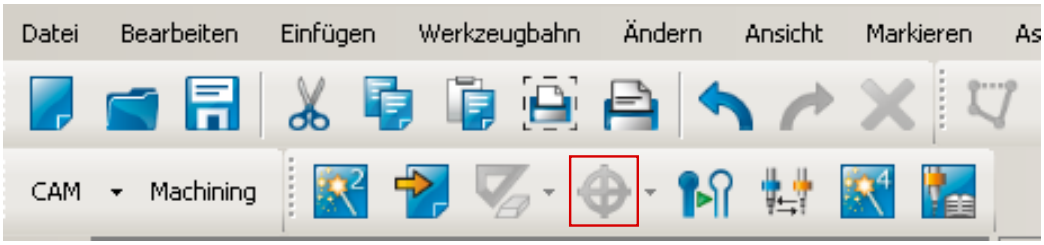


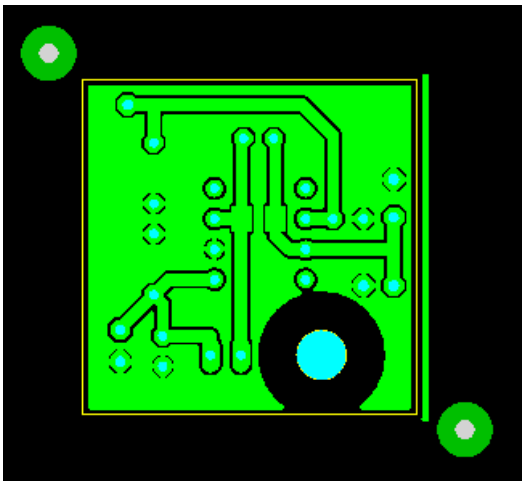


Now import the previous exported .top, .bot, .out and . drill files. Usually the .top, .bot, .out files are directly identified. However the .dril file is not, so that you have to choose *DrillPlated* in the *Layer/Vorlage* section. Sometimes the magnitude of the dimension is not correct. Therefore check the magnitude of your board and change the *Dezimalstellen* which is marked in green.

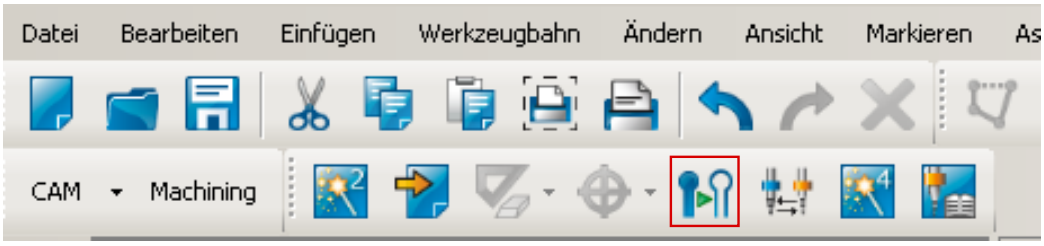
Step 3: Passermarken setzen

The *Passermarken* are orientation points for the double sided boards. Choose *Passermarken setzen* → *Fiduita* and place marks beside the left lower and right upper corner of the board.

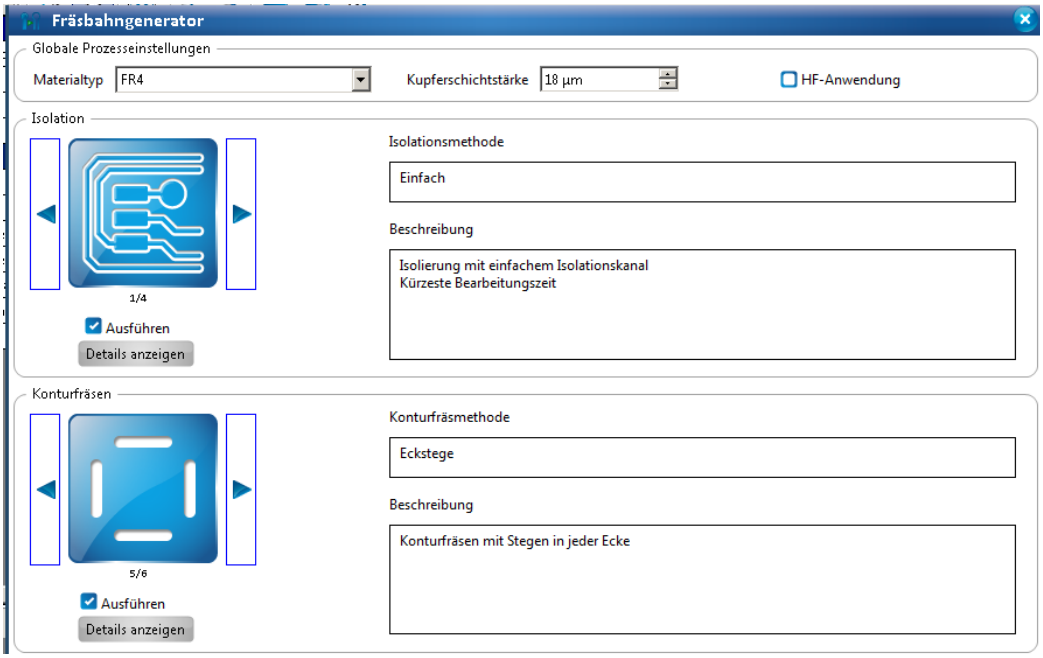


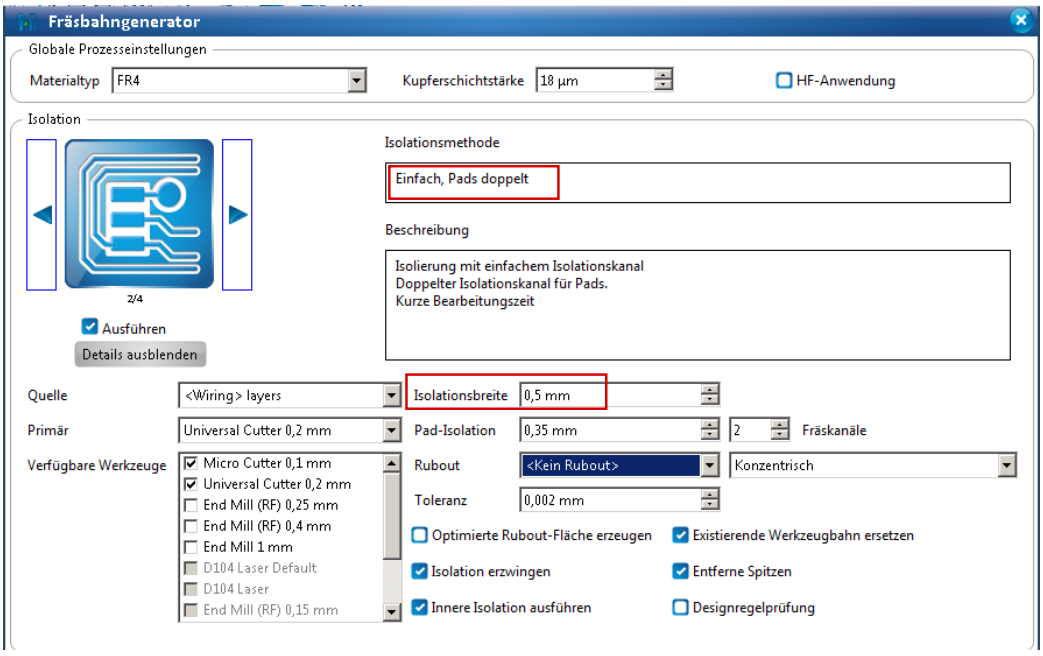


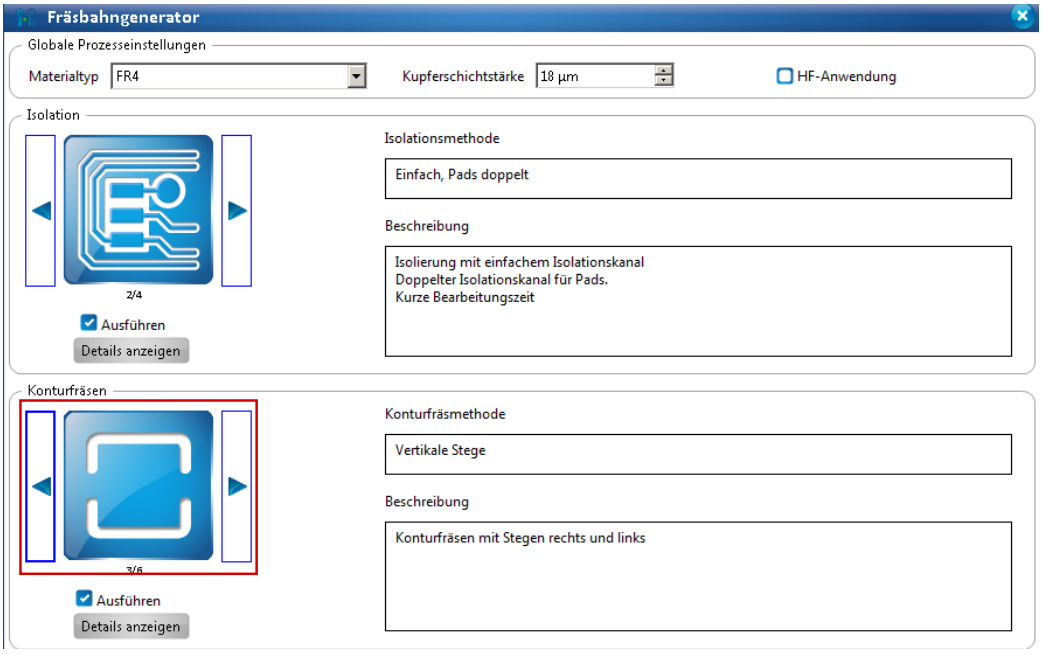
Step 4: Fräsbahngenerator



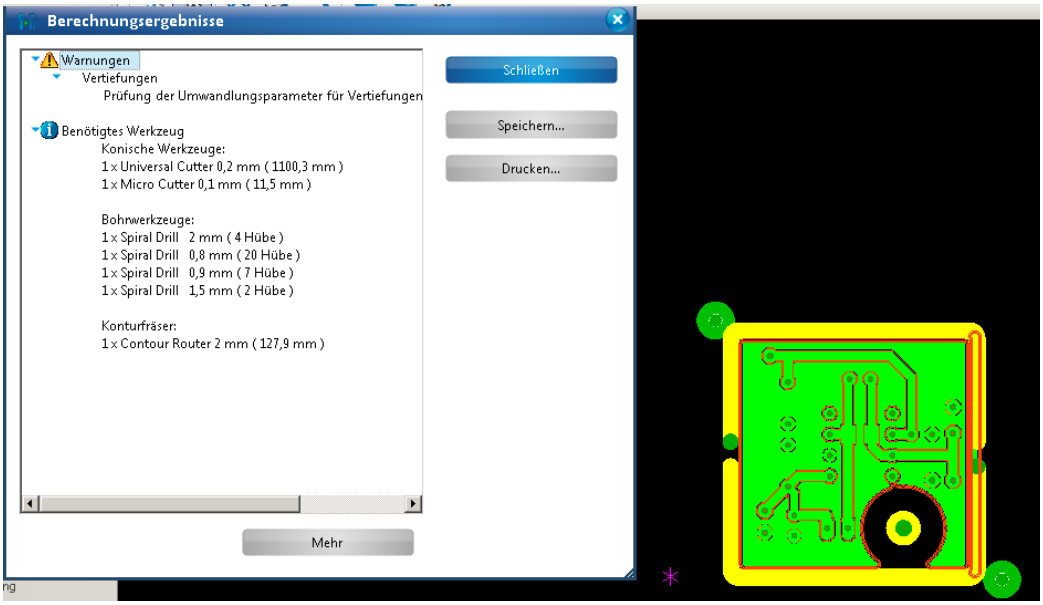
Here are a lot of settigs possible. The below shown settings are recommended.


Fräsbahngenerator
Globale Prozesseinstellungen
Materialtyp
FR4
Kupferschichtstärke
18 µm
HF-Anwendung
Isolation
1/4
Ausführen
Details anzeigen
Isolationsmethode
Einfach
Beschreibung
Isolierung mit einfachem Isolationskanal
Kürzeste Bearbeitungszeit
Konturfräsen
5/6
Ausführen
Details anzeigen
Konturfräsmethode
Eckstege
Beschreibung
Konturfräsen mit Stegen in jeder Ecke

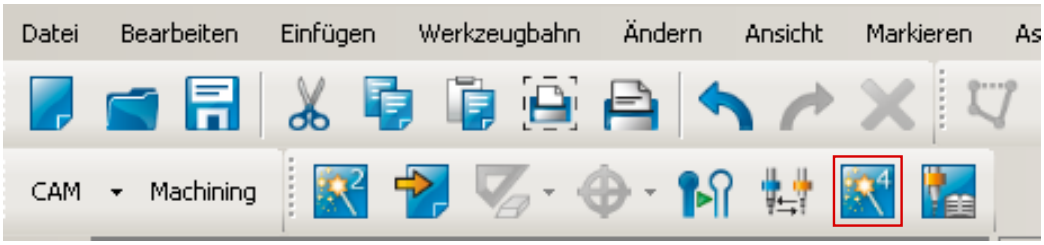

Fräsbahngenerator
Globale Prozesseinstellungen
Materialtyp
FR4
Kupferschichtstärke
18 µm
HF-Anwendung
Isolation
2/4
Ausführen
Details ausblenden
Isolationsmethode
Einfach, Pads doppelt
Beschreibung
Isolierung mit einfachem Isolationskanal
Doppelter Isolationskanal für Pads.
Kurze Bearbeitungszeit
Quelle
<Wiring> layers
Isolationsbreite
0,5 mm
Primär
Universal Cutter 0,2 mm
Pad-Isolation
0,35 mm
2
Fräskanäle
Verfügbare Werkzeuge
Micro Cutter 0,1 mm
Universal Cutter 0,2 mm
End Mill (RF) 0,25 mm
End Mill (RF) 0,4 mm
End Mill 1 mm
D104 Laser Default
D104 Laser
End Mill (RF) 0,15 mm
Rubout
<Kein Rubout>
Konzentrisch
Toleranz
0,002 mm
Optimierte Rubout-Fläche erzeugen
Existierende Werkzeugbahn ersetzen
Isolation erzwingen
Entferne Spitzen
Innere Isolation ausführen
Designregelprüfung



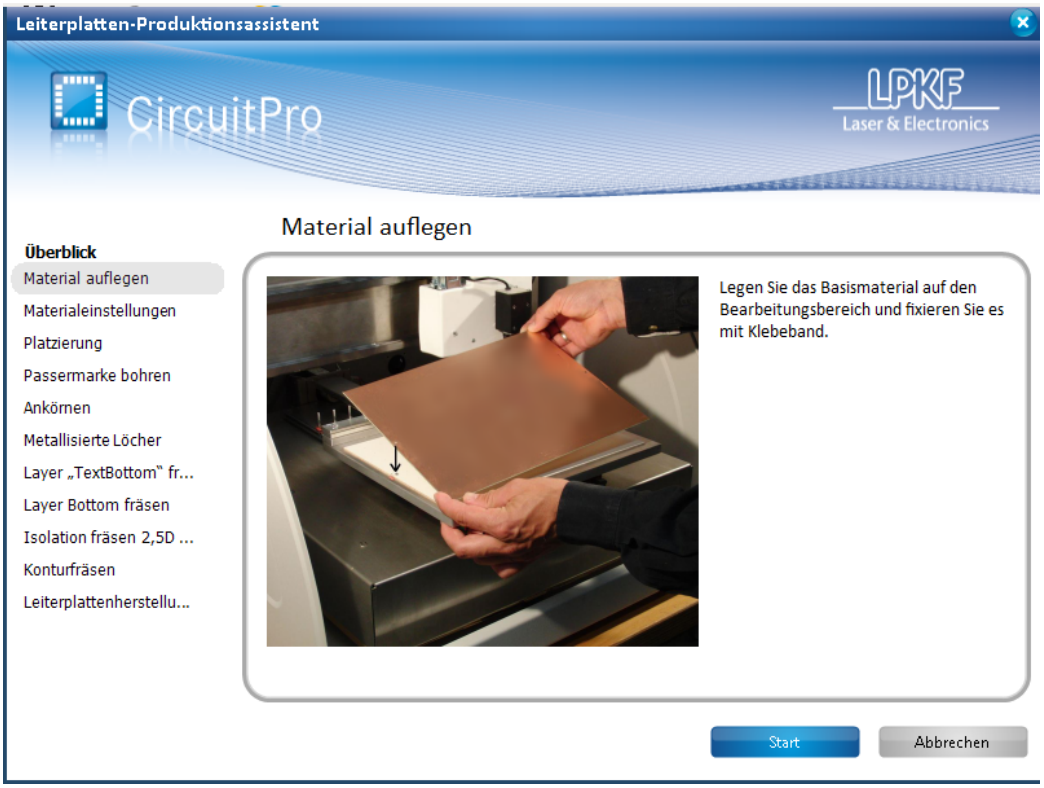
Now you find a list of the needed and used tools. It can happen that you seem to need a tool which is not available. E.g. for holes with a diameter of 0.6 mm. If the size of the hole is not needed to be 0.6 mm but can also be 0.7 mm then choose manuelly the best tool to replace it with. If there is an error which seems to be important ask someone.



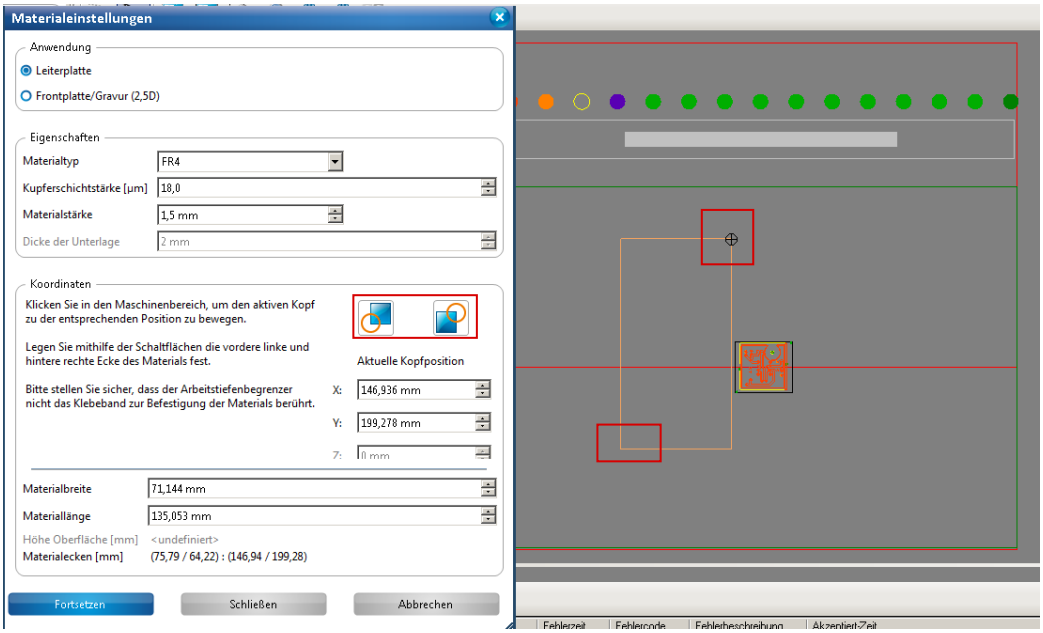
Step 5: Produktionsassistenten starten

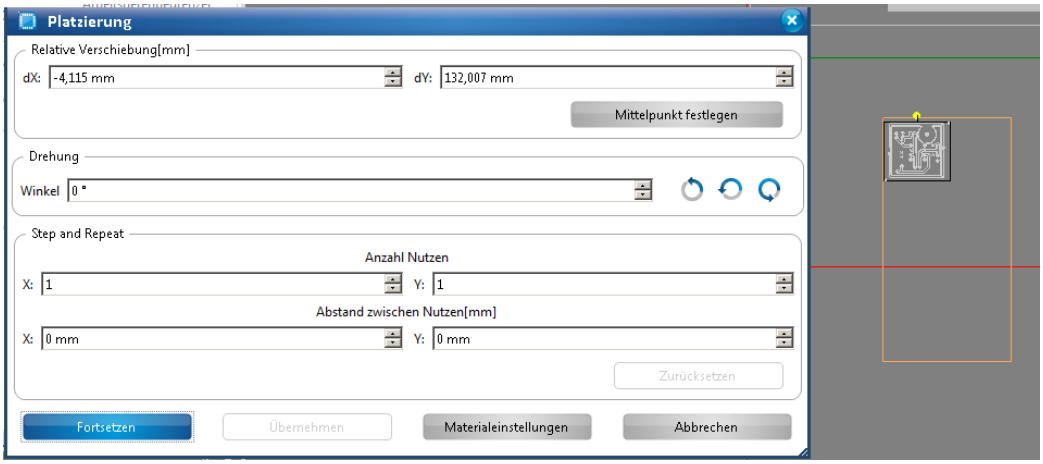


Mount the board material with the provided tape.



Now move the milling head by clicking with your mouse on the grey screen to set the limits of the material. You set the limits by defining the lower left and upper right corner as before for the *Passermarken*. Afterwards mark the board and place it in the area. If you have a double sided board you have to turn the material upside down manually. Therefore follow the given instructions.





Step 5: Check the Result

An often occurred problem was that not all pads for SMD parts were separated properly. You might want to check especially small areas carefully before soldering.