



DMSF PCB Router Operation Guide

A. <u>Preparation of NC files for PCB Router</u>

The input to the PCB router is NC file that contain commands and data being readable by the controller of the router's controller. These commands and data will instruct the router to move the cutting tool to perform drilling and isolation cutting.

For the machining of 2-side PCB, the following 4 NC files will be generated:

- 1. Alignment Holes Drilling NC file ("c:\DMSFApps\AlignHoleDrill.nc")
- 2. PCB Holes Drilling NC file ("01-Drill.nc")
- 3. PCB Top Layer Isolation Cutting NC file ("02-Top.nc")
- 4. PCB Bottom Layer Isolation Cutting NC file ("03-Bot.nc")

A.1 Alignment Holes Drilling NC file

- Double-click the Apps icon of CopperCAM on the desktop.
- If there is no previous data in CopperCAM working space, then CopperCAM will ask you to select the first layer of PCB:



Ctrl+S

Open			×					
→ 🔹 🛧 📙 « 0 - Test Data → 20170406	ٽ ×	Search 20170406	Q					
rganize 🔻 New folder								
Dropbox ^ Name	^	Date modified	Туре					
OneDrive 20170419		02/05/2017 14:45	File folder					
xxx-NotUsed		02/05/2017 14:45	File folder					
This PC 20170421.gbr		21/04/2017 16:45	GBR File					
Desktop T16155.GBL		29/03/2017 15:34	GBL File					
Documents T16155.GTL		29/03/2017 15:34	GTL File					
Downloads		29/03/2017 15:34	Text Docur					
h Music								
Pictures								
📲 Videos								
L OS (C:)								
DATA (D:)								
✓ <			>					
File name:	~	Gerber Scientific Flasher	r (*.GBR;*.GRB;*.TO	;;::::::::::::::::::::::::::::::::::::	CMP;*.SOL;*.GTL;	*.PHO;*.ART	r;*.GB*;*.GM*;*.T	XT;*.GTL;*.0
		All files (*.*)		$ \rightarrow $				
		CopperCAM file (*.PCB Gerber Scientific Flasher	r (*.GBR;*.GRP;*.To	;*.BOT;*.0	CMP;*.SOL;*.GTL	*.PHO;*.AR	T;*.GB*;*.GM*;*.	TXT;*.GTL;*.
		WinTypon-5 File (* WT	5; .GTL)			_		
					CopperCA	M . T1	6155 GPI	(lavor
If there is previous data	ata in Conner(AM working s	nace then	Con	CopperCA	IVI - 11	IOIDJ.GBL	Layer
-				File	Machin	e Edit	Display	Param
select pull-down mer	nu "File -> Clos	e layer" to cl	ose all					
layers.			🧱 Copr	e	Open			>

 After all layers are closed, then click "Open Project" File Mac button

Close layer...

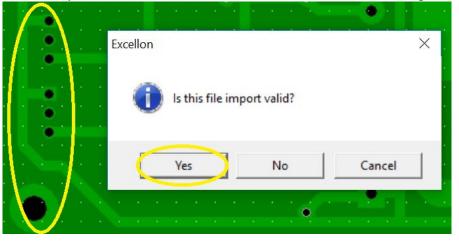


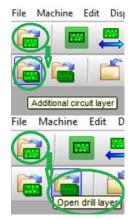


- Ensure that the file type of "Gerber Scientific Flasher" is selected, and then select the required PCB
 Top layer Gerber file (either *.GTL or *.GBL file).
- After the GTL or GBL file is opened, within the "Board dimensions" dialogue box, then specify the required "Reframe Margin", say, 5 mm.
 - "Reframe around existing circuitry with a margin of ..." is checked ON.
 - Default value of "Z Thickness" = 1 mm.
- Click "OK" button.
- For the "Warning" dialogue box, click the "No" button to keep the drilling depth of 2.8 mm.



- Browse to "Open Project" button, and then select "Additional circuit layer" button.
- Select the required PCB Bottom layer Gerber file (either *.GTL or *.GBL file).
- Browse to "Open Project" button, and then select "Open drill layer" button.
- Select the required PCB hole drilling layer EXCELLON file (*.TXT).
- Check that the imported holes are matched with the position of the top and/or bottom layer, then click the "Yes" button of the "Excellon" dialogue box:

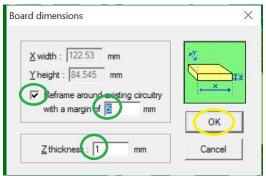


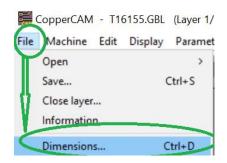




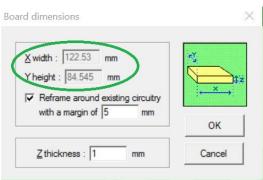


- Select pull-down menu "File -> Dimension..." or use "CTRL+D" to display the "Board dimensions" dialogue box.





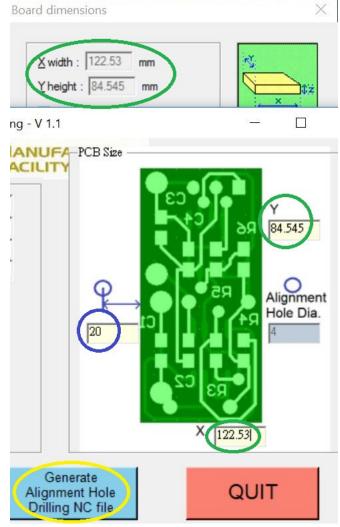
- Specify the required "Reframe Margin", say, 5 mm.
 - "Reframe around existing circuitry with a margin of ..." is checked ON.
 - Default value of "Z Thickness" = 1 mm.
- Click "OK" button.
- Again select pull-down menu "File -> Dimension..." or use "CTRL+D" to display the "Board Dimension" dialogue box.
- Please note the "X width" and "Y height" of the PCB.
- Double-click the Apps icon of DMSF-PCB-Drill-Alignment-Hole on the desktop.
- Enter the following values:
 - Required margin between alignment pins and boundary of PCB (say, 20 mm)
 - PCB Size X and Y (matches with values in "Board dimensions")







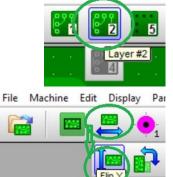




- Click "Generate Alignment Hole Drilling NC file" button to generate the Alignment Holes Drilling NC file.
 - The alignment hole drilling NC file will be created as "c:\DMSFApps\AlignHoleDrill.nc".

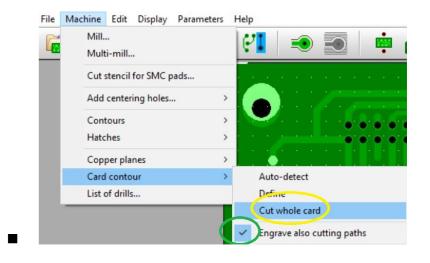
A.2 PCB Holes Drilling NC file

- Click the "Layer 2" button at the right-hand side of ribbon menu.
 - This will activation the layer 2 (i.e. bottom layer)
- Browse to "Flip ..." button and click "Flip Y" button once.
 - You will notice that now the holes and bottom layer are not aligned.
- Select pull-down menu "Machine -> Card contour"
 - Ensure that "Engrave also cutting paths" is checked ON.
 - Click "Cut whole card".



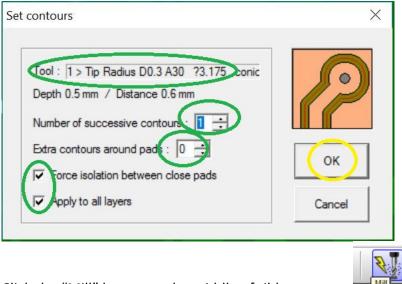








- Click the "Calculate contours" button around middle of ribbon menu.
- Ensure that the following shown values are correct, and the click the "OK" button of the "Set contours" dialogue box.



- Click the "Mill" button at the middle of ribbon menu.
- Enter "2.8" for "Drilling depth".
- Ensure that the following shown values are correct, and the click the "OK" button of the "Active tools" dialogue box.





Active tools		×
ENGRAVING TOOL	DRILLING TOOLS	
1 > Tip Radius D0.3 A30 ?3.175 conical 3 -	C Use one single tool for all drills, with circular boring	무무무
Depth : 0.5 mm => Radius 0.3 mm Margin : 0 mm => Distance 0.3 mm	> 3 > Drill D1 ?1 cylindrical	
Engraving speed : 500 mm/min	 Use for each drill the closest smaller tool, with circular boring Use for each drill the closest greater tool, without circular boring 	
HATCHING TOOL	Selected tools	
2 > EM D4R0 ?4 cylindrical	1 3 > Drill D1 ?1 cylindrical	
Depth : 0.1 mm => Radius 2 mm	2 · X	ок
Margin : 0 mm => Overlap 50%	3 × X	
Hatching speed : 800 mm/min	4 X	
	5 X	1
		Cancel
1 > Tip Radius D0.3 A30 ?3.175 conical 3 -	7 × X 8 × X	
Cutting depth : 1 mm	9 • • • • • • • • • • • • • • • • • • •	
Cutting speed : 500 mm/min	10 X	
CENTERING TOOL	Allow circular boring for all holes over 1 mm	
4 > Drill D4 - Alignment Hole ?4 cylindrical		
Hole diameter : 3.17 mm	Drilling depth : 28 mm Boring speed : 80 mm/min	
Extra depth : 6 mm	In case of a boning cycle : 🔽 Drill at centre 🔽 Lift-up at centre	

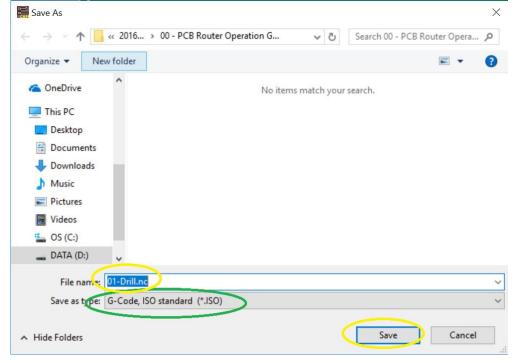




- Ensure that the following shown values are correct, and the click the "OK" button of the "Final output" dialogue box.

Final output			×
	(Tool #3 ?1)	Flip X	
Section #3 : Engra	ving layer #2	🗖 Flip X	C. C
Section #4 :	T	🗖 Flip X	
Section #5 :		Flip X	ок
Section #6 :	Ŧ	🗖 Flip×	
Section #7:	~	🗖 Flip×	Cancel
Section #8 :	Ÿ	🗖 Flip X	
XY-zero point North-West comer White cross South-West comer	North-East come Reference pad C South-East come	r O	Z-zero point Circuit surface Machine bed
	Advanced parameters	3 >>>	

- Enter the required NC filename (say, "<mark>01-Drill.nc</mark>") at your preferred folder.



Again click the "Mill" button at the middle of ribbon menu.

- Click the "OK" button of the "Active tools" dialogue box.
- Select "Engraving layer #1", and the click the "OK" button of the "Final output" dialogue box.
- Enter the required NC filename (say, "02-Top.nc") at your preferred folder.
- Again click the "Mill" button at the middle of ribbon menu.
- Click the "OK" button of the "Active tools" dialogue box.
- Select "Engraving layer #2", and the click the "OK" button of the "Final output" dialogue box.
- Enter the required NC filename (say, "03-Bot.nc") at your preferred folder.

B. Router Preparation

The router is enclosed by a safety cover.

- By opening the front door of the safety cover will trigger the machine to stop XYZ movement.
 - However, the spindle is still rotating!
 - Please remember to STOP the SPINDLE via the controlling software.
 - The side door of the safety cover is for emergency use only.
 - Please DO NOT open the side door!

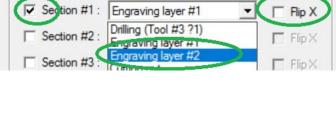
B.1 Turn ON the Router

Turn on the router by pressing the GREEN button at the front of the control box

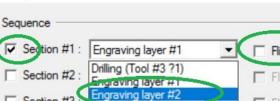
C. Launch Contoller App of Hand-wheel and Router

- Double-click the "HB03AutoEn" icon on desktop to launch the App of hand-wheel.
- Double-click the "NcStudioV5..." icon on desktop to launch the App of NC Controller.

















Engraving layer #1

miling (Tool #3 ?1)

Flip)

Final output

Final output

Sequence

ection #1 :

Section





File Edit View Operation Machine Window Help Image: Constraint of the second s	🕱 NcStudio AD. Engraving Machine Control System - 03-bot-02.nc	
JOC IDLE NC State A.s. M. Coor. Remaine Feedrate(mm/min) Setting: 2000 Current Command Set: Y. 188.216 155.388 0.000 Ourrent Line(f 100% Spindle rev: G0 G21 G90 G17 G54 Y. 188.216 155.388 0.000 Ourrent Line(f 0N 100% 100% G0 G21 G90 G17 G54 Spindle: ON 100% Spindle: ON ON G0 G21 G90 G17 G54 JOO 150 200 250 300 350 400 JOG 150 200 250 300 350 400 JOG 150 200 250 300 350 400 JOG Y Y Y Y Y Q 0 0.05mm Trace Log Manager Parama Editor 100 State 0.000 0.000 0.000 0.000 JOG Y Y Y Y Y		
NC State A> M. Coor. W. Coor. Remaine Feedrate(mm/min) Spindle rev. Setting: 18000 Current Command Set: Y: 188.216 155.388 0.000 0 Current Line(1 0 100% Spindle rev. Setting: 18000 G0 G21 G90 G17 G54 G40 G43 G80 V: 182.216 155.388 0.000 Current Line(1 0 100%	😂 🖳 🗊 💀 🦾 🛓 🖡 🕨 🖩 🖬 🖉 🕢 🛛 X Y Z	
A> M. Coor. W. Coor. Remaine Feedrate(mm/min) Spindle rev: G 021 G80 G17 G54 Y. 185216 155.388 0.000 Current Line(F 100% 100% G021 G80 G17 G54 Y. 168.216 155.388 0.000 Current Line(F 100% Spindle rev: 100% G021 G80 G17 G54 Y. 168.216 155.388 0.000 Current Line(F 00% Spindle: ON 0 Trace Log Manager Parama Edtor 100State MUTO MA 100 150 200 250 300 350 400 MITO MA 5 6 7 100 150 200 250 300 350 400 7	JOG IDLE	
• X: 104.309 • Y: 168.216 155.38 0.000 • Y: 168.216 155.38 0.000 • Urrent Line(f 100% Spindle: ON Go G21 G90 G17 G54 G40 G43 G80 • Z: -52.953 20.000 0.000 Current Line(f 100% Spindle: ON GN • JOG Manager Params Edtor ON 100% Spindle: ON GN JOG V+ (g)	NC State	
100 150 200 250 300 350 400 50 Image: Constraint of the second	Product Non-Strain Non-Strain	000 G0 G21 G90 G17 G54 000 G40 G43 G80
50 JOG (* Jog 50 (*) (*) 6 (*) (*) 7 (*) <td< th=""><th>Trace Log Manager Params Editor IOState</th><th>AUTO MA</th></td<>	Trace Log Manager Params Editor IOState	AUTO MA
	100 150 200 250 300 350 400 5 0	JOG (* Jog Y+ Z+ (8) (9) C 0.05mm X- X+ (4) (6) Z- Y- (1) (2) Tip: Press TURBO(or CTRL) key to jog C 10mm

D. <u>PCB Machining</u>

- Use the physical hand-wheel to control the X-, Y- and Z-axis movement for:
 - Workpiece coordinate system setting (i.e. X = 0, Y = 0)
 - Tool length setting (i.e. Z = 0)

D.1 Setup the PCB Board

- Take the raw materials of the PCB board.
- Apply double-side adhesive tape to the back of the board.
- Fix the board on the machinable resin block.
- Setup the workpiece coordinate system (0,0).

D.2 Setup Drill bit of 4mm Diameter

- Take the 4mm collet and the 4mm diameter drill bit.
- Install the collet and drill bit at spindle head.
- Reset the "Z=0" for this drill bit.

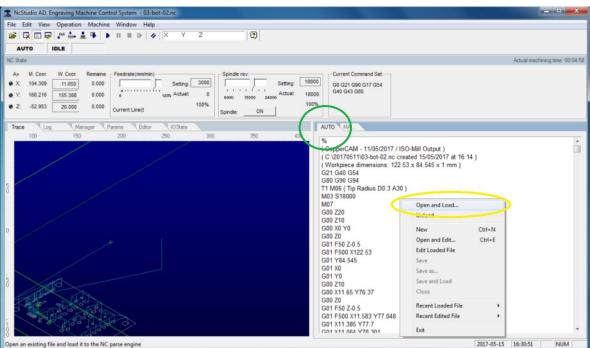
D.3 Alignment Hole Drilling

- Load the "AlignHoleDrill.nc" file into NcStudio.
 - Right-click in the "AUTO" page of NcStudio, then select "Open and Load...":









- Check that the loaded file is correct.
- Press the "Start / Pause" button on the hand-wheel to start the machining.
- Upon the completion of the machining:
 - Select the "Z" by the dial switch.
 - Move up the tool by turning the hand-wheel.

D.4 Fix PCB Board by 2 x Alignment Pins

- Take 2 x 4mm alignment pins.
- Press the pins into the alignment holes firmly.

D.5 Setup Drill bit of 1mm Diameter

- Dismantle the 4mm drill bit and the 4mm collet.
- Take the 3.175mm collet and the 1mm diameter drill bit.
- Install the collet and drill bit at spindle head.
- Reset the "Z=0" for this drill bit.

D.6 PCB Board Hole Drilling

- Load the "01-Drill.nc" file into NcStudio.
- Check that the loaded file is correct.
- Press the "Start / Pause" button on the hand-wheel to start the machining.
- Upon the completion of the machining:
 - Select the "Z" by the dial switch.
 - Move up the tool by turning the hand-wheel.

D.7 Setup V-bit of 0.2mm Tip Width, Angle=20°









- Dismantle the drill bit of 1mm diameter.
- Take the V-bit of 0.2mm Tip Width, Angle=20°.
- Install the V-bit into the collet.
- Reset the "Z=0" for this V-bit.

D.8 PCB Board Top Layer Engraving

- Load the "02-Top.nc" file into NcStudio.
- Check that the loaded file is correct.
- Press the "Start / Pause" button on the hand-wheel to start the machining.
 - Upon the completion of the machining:
 - Select the "Z" by the dial switch.
 - Move up the tool by turning the hand-wheel.

D.9 Prepare PCB Board for Bottom Layer Engraving

- Dismantle the 2 x alignment pins.
- Take out the PCB board from the machinable resin block.
- Clear the double-side adhesive tape at the bottom layer of PCB board.
- Apply double-side adhesive tape at the top layer of PCB board.
- Fix the PCB board on the machinable resin block, with bottom layer facing upward.
- Press the pins into the alignment holes firmly.

D.10 PCB Board Bottom Layer Engraving

- Load the "02-Bot.nc" file into NcStudio.
- Check that the loaded file is correct.
- Press the "Start / Pause" button on the hand-wheel to start the machining.
- Upon the completion of the machining:
 - Select the "Z" by the dial switch.
 - Move up the tool by turning the hand-wheel.

REMARKS:

For any assistance, please contact Mr. William Tai (2358.8097) / Mr. Anthony Yam (2358.8900) / Mr. Jeremy Tam (2358.8899).