

GCode for the EBF Machine

General Notes

- The code you will use from the VCarve export should use the Big Blue machine.
- Gcode from other sources will probably need editing and addition of the Header and Footer below. The Fusion360 WinCNC machine definition may add extra stuff that the EBF will not recognize.
- Generally, WinCNC prefers comments in () rather than []
- Text in black below are actual machine commands.
- Text in red is explanatory text only and is not required in the GCode file. Any text within () is ignored by WinCNC
- Tool, Speed, and Spindle start (M3) need to all be on separate lines for the EBF

Header

(This section is made up of notes for the operator. None of this code actually makes the machine do anything)

(Name-of-project JDW 20190401 - This can be whatever you want it to be)

(Tools - The list below is for reference so that you can check that the right tools are in the right tool slots in the machine. You will use "T1", "T2" etc in the actual code so you need to make sure that you've instructed the machine to pick up the right tool.)

(T1 .250 end mill)

(T2 .375 end mill)

(T3 .500 end mill)

(T4 5mm end mill)

(T5 .1 end mill)

(T6 .750 dovetail cutter)

(The following are the actual commands that the machine will execute in the header. Extra line spaces have no effect and are present to aid visualization)

G20 (Make the machine use inches)

G40 (Cancels cutter radius compensation in case one was set - handled by software)

G80 (Cancels canned cycle in case one was set)

M11C7 (Turns the vacuum table on - may be redundant, but avoids and abort)

T1 (Tells the machine to get Tool 1 or whichever tool you need. Will invoke TU and TC macro.)

S10000 (Spin that tool at 10,000 RPM. Adjust this as required for your endmill. Generally bigger tools = slower speeds)

M3 (Start the spindle - will invoke the SPINON macro)

M11C4 (Air blast cooling for spindle on)

M11C2 (Dust hood down)

G04 (Tells the machine to wait for the Enter key to be pressed before beginning the job)

(Your gcode movement instructions exported from VCarve will go here. It's important to check your rapid movement and safe movement heights in VCarve before export as those heights will be in this code. VCarve also makes the machine move to X0 Y0 before moving to the first cut area. This code isn't necessary and can be removed if you wish but if you're unsure, just leave it in.)

Tool Changes

(If executing a tool change mid project. Make sure that the last line of the code before the tool change includes raising the tool to a safe movement height above the material. If you exported multiple toolpaths from VCarve, the name you gave each toolpath will be shown at the point in the file where the toolpath changes, so it is helpful to give your toolpaths descriptive names. You may need to remove other VCarve G#, M#, S, F or T commands between movement commands)

T2 (Tells the machine to go get tool 2, or whichever tool number you input here. The T# command will automatically stop the spindle and execute the machine commands necessary to get the new tool without damaging the machine)

S10000 (Spin the tool at 10,000 RPM. If you do not specify a S#, the new tool will spin at the same speed as the previous tool.)

M3 (Start the spindle)

Footer

T0 (Tells the machine to stop spindle, unload tool, and position spindle at the back of the machine.

M12C7 (Turn the vacuum off)

M2 (Tells WINCNC that its the end of the program)

Header and Footer without comments are on the next page

Header and Footer without all the comments

(I suggest at least one comment to identify the project)
(Tool and material descriptions are nice too)

G20
G40
G80
M11C7
T1
S10000
M3
M11C4
M11C2
G04

(Body of your Gcode goes here)

Footer

T0
M12C7
M2