

GCode for the EBF Machine with Explanation

General Notes

- The code you will use from the VCarve export should only be machine movement code.
- All G#, M#, S#, or T# codes from the export should be removed and replaced as shown below. Only one G#, M#, S# or T# command per line is allowed in WinCNC. Most VCarve GCode export formats put multiple commands on one line.
- 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

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 and is not necessarily how your tools will be laid out. It's helpful to create the tool list in your Gcode like the one below 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, the tool change macro that executes automatically raises the Z to the max height. 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 an S#, the new tool will spin at the same speed as the previous tool. Be careful of this if you're using tools of very different sizes.)

M3 (Start the spindle)

Footer

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

G53 Z0 X0 Y130 (This returns the machine to the park position. This is optional but reduces the chance of the user turning off WinCNC when the machine is not above the "Park" position and the spindle falling somewhere that causes damage.)

M12C7 (Turn the vacuum off)

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

GCode for the EBF Machine

Header

G20 (Inches)
G40 (Cancels cutter radius compensation)
G80 (Cancels canned cycle)
M11C7 (Turns the vacuum table on)

T1 (Get tool 1)
S10000 (Speed = 10,000)
M3 (Start the spindle)

M11C4 (Air blast cooling for spindle on)
M11C2 (Dust hood down)
G04 (Wait for Enter)

Tool Changes

T2 (Get T2)
S10000 (Speed = 10,000)
M3 (Start the spindle)

Footer

T0 (Unload Tool)
G53 Z0 X0 Y130 (Move machine to "Park" location)
M12C7 (Turn the vacuum off)
M2 (End Program)

EBF Checklist - Long

Prior to Turning On the EBF

- Check the oil in the vacuum pump through the sight glass on back of the unit. Use your phone to take a photo.
- Check dust collector bags outside and make sure they are not all full. Change if necessary.
- Move the control cabinet into the red taped area on the floor to be clear of the machine and the metal shop door
- Make sure that the large air compressor outside is on. Press the green button on the electrical box to the right as you're facing the EBF machine.
- Make sure that compressed air is supplied to the table. Connection is on the front lower left corner of the machine near the floor. The regulator is on the wall to the left of the machine and should read at least 70PSI.
- Load cutters/endmills into the tool holders and tool holders into the correct slots. They are numbered 1-8 left to right. This can be done at any time prior to setting tool heights.
- Adjust the dust hood if necessary via the allen head screws on the sides. This should not normally be necessary.
- If you are using the vacuum table (which is almost always the case), move rubber gasket material to the correct positions for the piece you will be vacuum holding to the table.

Turning Machines On

- Turn on the EBF on the main circuit breaker in the metal shop.
- Turn on the local power disconnect via the big red lever to the left of the machine.
- Make sure the rotary switch on back of the control cabinet is to 'Marche'
- Turn on the control cabinet power - black switch on left side of the control cabinet.
- Open the cabinet and push the power button on the computer.

Setting Up To Cut

- Launch WinCNC
- Make sure nothing is in the way of the machine's movement either on or next to the machine.
- Press "Home" on WinCNC
- Once the machine is homed, press the "Set X0 Y0 Z0" button or manually input a "G92 X0 Y0 Z0" command on the machine. This sets the initial 0,0,0 position.

- Move the spindle out of the way so that you can position your spoil board/fixture. Use the arrow keys on WinCNC, the arrow keys on the keyboard, or issue a G00 with coordinates command.
- Position your spoilboard/fixture.
 - If using the machine pins for positioning
 - Be sure fixtures and spoil boards are clear of the pins.
 - Click “Pins Up” in WinCNC.
 - Position spoilboard/fixturing surface against side and front pins
 - If not using the machine pins, position spoil board/fixture as needed on the table.
- Position your material on your spoil board/fixture. This will be either with vacuum hold down, screws, or a fixturing table, depending on how you will secure your work piece.
- Load your tool table in WinCNC if needed (not frequently used).
- Touch-off each tool.
 - Place the touch-off button on the machine base (aluminum) on the side of the machine - the spot is marked. Load each tool one by one (T#) and move tool to touch-off position using the button “Move to X2 Y17” or issuing the command “G00 X2 Y17” and then click the “Set Tool Z” button on the WinCNC screen. Do this process for each tool using the T# command to change between tools.
- Determine the number of vacuum zones required and turn the switches to on (up) on the front of the machine. The zones are 1-4, front to back on the machine with 2 vacuum holes in the table per zone. The switches correspond to zones 1-4 from left to right.
- Cover any holes in the vacuum table that are not being used. These will be loose but will ensure that chips don't get down into the vacuum tubes.
- Turn on the vacuum by pressing the “Vacuum” button on WinCNC.
- Check the security of your spoilboard, fixture, and/or work piece.
- With your last tool (after touching off all tools) move the touch-off button to a suitable place on top of your work piece. This should be the highest point on your work surface. If your work surface is completely flat, you can set the touch-off button anywhere.
- Jog the spindle so that it is over the touch-off button, or move the button to underneath the spindle.
 - If you are using the vacuum table, make sure to turn on the vacuum before the next step.
- Press the “Set Material Z” button to set the Z height. As long as you touched off all of the other tools first, the machine will now know where the top of your material is for all tools.
- Jog the spindle to where you want your material X and Y zero to be.
- Note down the X and Y coordinates before the next step. This is not mandatory but is valuable if you need to restart your toolpath for some reason.
- Press the “Set Work X0 Y0” button or issue the command “G92 X0 Y0” to set the work piece X,Y zero position.

Loading the GCode and Cutting

- Save your GCode file from your flash drive to the computer. There is a USB hub on the left side of the machine.
- Load your GCode from the computer hard drive in WinCNC. (File, Open, yourfile) but DO NOT PRESS ENTER YET! Pressing Enter will start the toolpath.
- Turn on the main workshop dust collector. Make sure that all of the other gates in the shop are closed and that the gate above the machine is open. Turn on Vacuum (if not already on)
- Drop the machine pins if you raised the pins for aligning your work piece.
- If you have copied the suggested header into your GCode, the machine will turn on the vacuum table if it isn't already on, grab the first tool in your file, drop the dust hood, start the coolant, and wait for you to press Enter before it begins.
- Press Enter - Cutting will begin

Post Cut

- If you used the suggested Footer code, the T0 command turns off the spindle, returns the last tool used to the tool changer, the M12C7 turns off the vacuum table and the G53 command sends the machine to the Parking position.
- If you did not use the suggested Footer
 - Wait till the spindle stops spinning before doing anything.
 - Jog the machine out of the way of your material as needed
 - Turn the vacuum off via the button in WinCNC
 - Issue the T0 command to return the last tool used
- Turn dust collection off
- Press the "Go To Parking" button to move the machine to sit over top of the parking space at the back of the machine. If you turn off the computer, the spindle will drop quickly, so the parking position has had a pad added.
- Lower the spindle Z height to just over the pad in the parking position.
- Exit WinCNC
- Remove your material
- Vacuum top, under and around machine
- Empty dust collection bag(s) if full
- Restore/adjust tools in tool holders and tool order as needed.
- Shut down Windows
- Turn off cabinet (black switch on left side of cabinet)
- Turn switch on the back of the cabinet to "Arrete".
- Turn off power at the local breaker box behind the cabinet - big red lever.
- Turn off EBF breaker in the breaker box in the metal shop.

- Clear any custom fixtures and make machine ready for next user
- Move the PC cabinet back out of the way and so that the metal shop door can be opened.

EBF Checklist - Short

Prior to Turning On the EBF

- Check the oil in the vacuum pump.
- Check dust collector bags.
- Move the control cabinet into the red taped area.
- Turn on air compressor.
- Check that compressed air is supplied to the table - at least 70 psi.
- Load cutters/endmills into the tool holders and tool holders into the correct slots.
- Adjust the dust hood if necessary.
- Move rubber gasket material to the correct positions for your piece.

Turning Machines On

- Turn on the EBF at main circuit breaker.
- Turn on the local power disconnect via the big red lever to the left of the machine.
- Turn the rotary switch on back of the control cabinet is to 'Marche'
- Turn on the control cabinet power on left side of the control cabinet.
- Open the cabinet and push the power button on the computer.

Setting Up To Cut

- Launch WinCNC
- Press "Home" on WinCNC
- Press the "Set X0 Y0 Z0" button
- Position your spoil board/fixture and work material.
- Load your tool table in WinCNC if needed (not frequently used).
- Touch-off each tool - Set Tool Z
- Cover any unused holes in the vacuum table.
- Turn on the vacuum by pressing the "Vacuum" button on WinCNC.
- Check the security of your spoilboard, fixture, and/or work piece.
- Touch off the tool on the top of your workpiece - Set Material Z. Make sure vacuum is on.
- Jog the spindle to where you want your material X and Y zero to be.
- Press the "Set Work X0 Y0" button.

Loading the GCode and Cutting

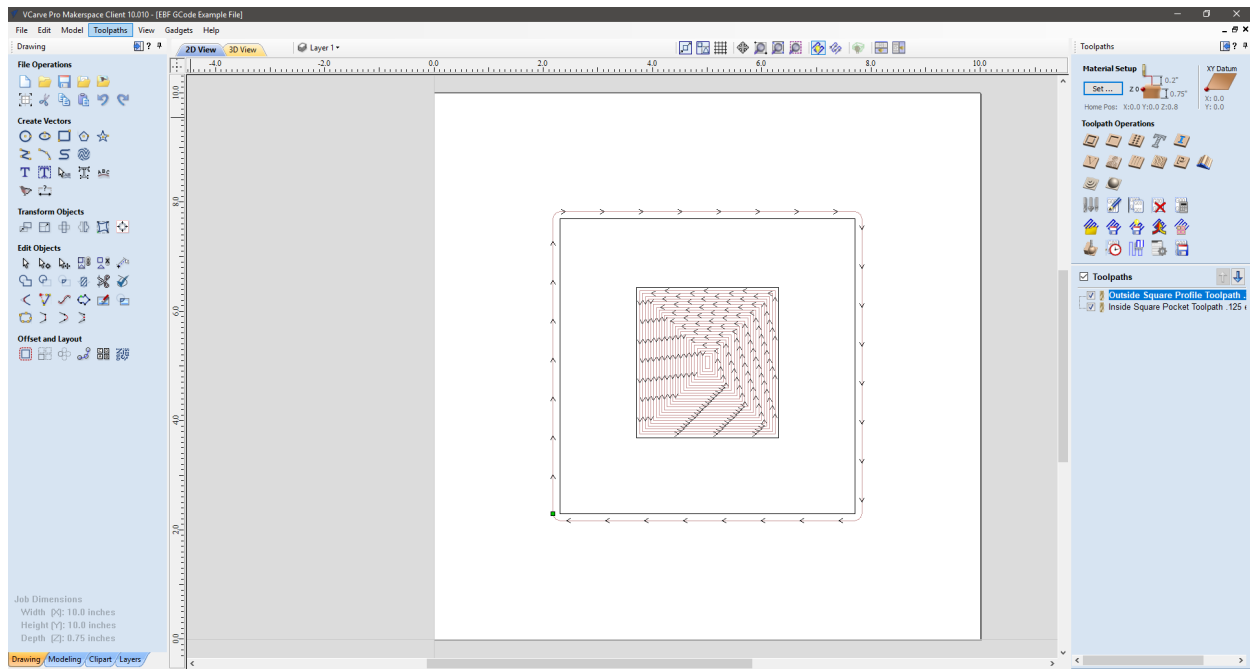
- Save your GCode file from your flash drive to the computer.
- Turn on the dust collector.
- Drop the machine pins if you raised the pins for aligning your work piece.
- Open the GCode file in WinCNC.
- Press Enter. This will pickup the correct tool and start the spindle.
- Once you hear the spindle spinning, press Enter again to start cutting.

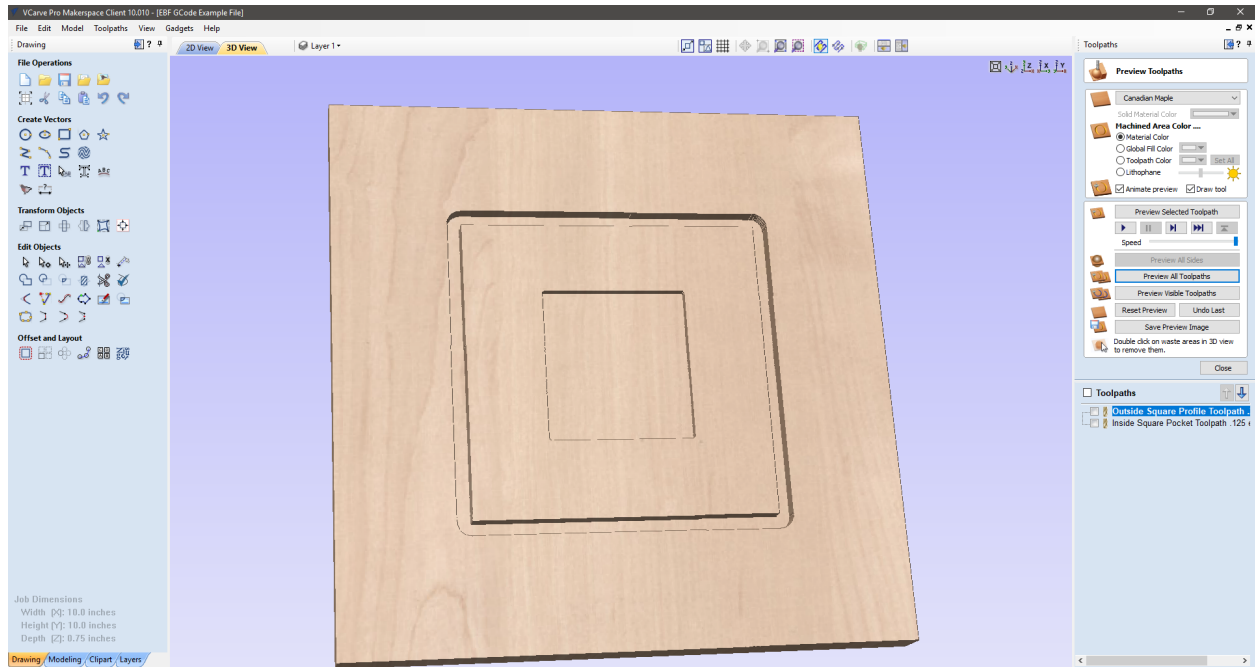
Post Cut

- Turn dust collection off
- Press the "Go To Parking" button.
- Lower the spindle Z height to just over the pad in the parking position.
- Exit WinCNC
- Remove your material
- Vacuum top, under and around machine
- Empty dust collection bag(s) if full
- Remove endmills from tool holders and replace all Makersmiths parts inside the cabinet.
Shut down Windows
- Turn off cabinet (black switch on left side of cabinet)
- Turn switch on the back of the cabinet to "Arrete".
- Turn off power at the local breaker box behind the cabinet - big red lever.
- Turn off EBF breaker in the breaker box in the metal shop.
- Move the PC cabinet back out of the way and so that the metal shop door can be opened.
- Put the large spoilboard back on top of the EBF to prevent shop dust etc. from getting into vacuum holes.

GCode Example From VCarve to EBF

The images below show what the code below looks like in VCarve. This is a very simple example project that uses two different endmills.





Create your toolpaths in VCarve as usual. Make sure that if you are using multiple tools, each different tool is given a different number.

Edit Tool

Name: End Mill (0.25 inch)


Notes:

Tool Type: End Mill

Geometry

Units: inches

Diameter (D): 0.25 inches



No. Flutes:

Cutting Parameters

Pass Depth: 0.125 inches

Stepover: 0.1 inches 40 %

Feeds and Speeds

Spindle Speed: 12000 r.p.m

Feed Units: inches/min Chip Load:

Feed Rate: 100 inches/min

Plunge Rate: 30 inches/min

Tool Number: 1

OK Cancel

Make sure in VCarve that you complete the Material Setup properly so that Rapid Z Gaps above Material and Home / Start Position are set correctly. These settings become part of the GCode that is exported and will impact your project.

Toolpaths

Material Setup

Thickness 0.75 inches

XY Datum Use Offset
 X: 0.0
 Y: 0.0
 Show detailed summary on toolpath tab

Z-Zero
 Material Surface
 Machine Bed

Model Position in Material

Gap Above Model
 0.0 inches
 Gap Below Model
 0.75 inches

Model Thickness 0.0 inches **Set...**
 Modeling Plane Z 0.0 inches

Rapid Z Gaps above Material

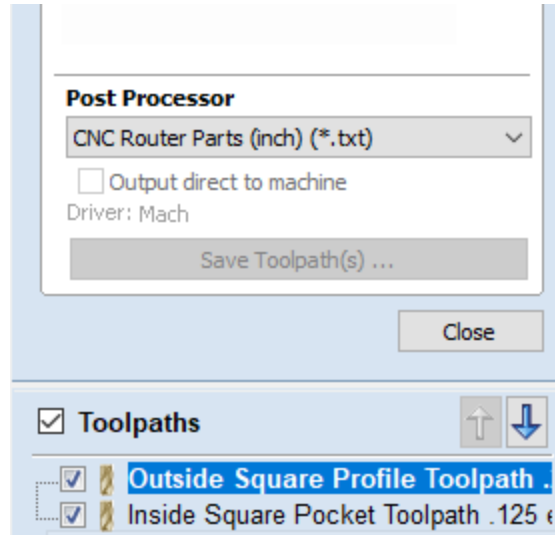
Z1 Clearance (Z1) 2 inches
 Plunge (Z2) 0.2 inches

Home / Start Position

X: 0.0 Y: 0.0
Z Gap above Material 2

OK Cancel

When saving toolpaths from VCarve, use the “CNC Router Parts (inch) (*.txt)” type:



The code below was exported from VCarve using the “CNC Router Parts (inch) (*.txt)” format.

- Text with strikethrough should be removed from the GCode entirely
- Text in blue is optional whether you remove or leave it in
- Text highlighted in green is text that should be added
- Text in red is explanatory text only for the purposes of this example. It is in brackets so that if you accidentally leave it in your code, it will not impact the cutting process.

(Squares EBF GCode Example) (You can edit any of the blue text in this section for your own reference)

(File created: Friday February 14 2020 - 05:13 PM)

(for CNC Roter Parts Machines)

(Material Size)

(X= 10.000, Y= 10.000, Z= 0.750)

()

(Toolpaths used in this file:)

(Outside Square Profile Toolpath .25 endmill) (These are the toolpath names as specified in the toolpath in VCarve. This is why its helpful to give your toolpaths meaningful names in VCarve.)

(Inside Square Pocket Toolpath .125 endmill)

(Tools used in this file:)

(1 = End Mill {0.25 inch}) (These are the tool names as specified in VCarve)

(2 = End Mill {1/8"})

~~G00G20G17G90G40G49G80~~

G70G91.1

M07

G00G43Z0.8000H1

S12000M03

(Toolpath: Outside Square Profile Toolpath .25 endmill Tool: End Mill {0.25 inch})

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)

X0.0000Y0.0000F100.0 (This sends the machine to X0 Y0 before starting a toolpath and is not necessary, but it also doesn't hurt anything)

G00X2.1750Y2.3000Z0.2000

G1Z-0.1250F30.0

G1Y7.7000F100.0

G2X2.3000Y7.8250I0.1250J0.0000

G1X7.7000

G2X7.8250Y7.7000I0.0000J-0.1250

G1Y2.3000

G2X7.7000Y2.1750I-0.1250J0.0000

G1X2.3000

G2X2.1750Y2.3000I0.0000J0.1250

G1Z-0.2500F30.0

G1Y7.7000F100.0

G2X2.3000Y7.8250I0.1250J0.0000

G1X7.7000

G2X7.8250Y7.7000I0.0000J-0.1250

G1Y2.3000

G2X7.7000Y2.1750I-0.1250J0.0000

G1X2.3000

G2X2.1750Y2.3000I0.0000J0.1250

G00Z0.2000

M5

(Change to Tool: End Mill {1/8"})

T2M6

S16000M03

T2 (Insert new tool number)

S10000 (Insert tool speed)

M3 (starts the spindle spinning)

(Toolpath: Inside Square Pocket Toolpath .125 endmill Tool: End Mill {1/8"})

G00X4.9627Y4.9547Z0.2000

G1Z-0.0417F10.0

G1X5.0377F35.0

G1Y5.1797

G1X4.9627

G1Y4.9547

G1X4.9127Y4.9047

G1X5.0877

G1Y5.2297

G1X4.9127

G1Y4.9047

G1X4.8627Y4.8547

G1X5.1377

G1Y5.2797

G1X4.8627

G1Y4.8547

G1X4.8127Y4.8047

G1X5.1877

G1Y5.3297

G1X4.8127

G1Y4.8047

G1X4.7627Y4.7547

G1X5.2377

G1Y5.3797

G1X4.7627

G1Y4.7547

G1X4.7127Y4.7047

G1X5.2877

G1Y5.4297

G1X4.7127

G1Y4.7047

G1X4.6627Y4.6547

G1X5.3377

G1Y5.4797

G1X4.6627

G1Y4.6547

G1X4.6127Y4.6047

G1X5.3877

G1Y5.5297

G1X4.6127
G1Y4.6047
G1X4.5627Y4.5547
G1X5.4377
G1Y5.5797
G1X4.5627
G1Y4.5547
G1X4.5127Y4.5047
G1X5.4877
G1Y5.6297
G1X4.5127
G1Y4.5047
G1X4.4627Y4.4547
G1X5.5377
G1Y5.6797
G1X4.4627
G1Y4.4547
G1X4.4127Y4.4047
G1X5.5877
G1Y5.7297
G1X4.4127
G1Y4.4047
G1X4.3627Y4.3547
G1X5.6377
G1Y5.7797
G1X4.3627
G1Y4.3547
G1X4.3127Y4.3047
G1X5.6877
G1Y5.8297
G1X4.3127
G1Y4.3047
G1X4.2627Y4.2547
G1X5.7377
G1Y5.8797
G1X4.2627
G1Y4.2547
G1X4.2127Y4.2047
G1X5.7877
G1Y5.9297
G1X4.2127
G1Y4.2047
G1X4.1627Y4.1547

G1X5.8377
G1Y5.9797
G1X4.1627
G1Y4.1547
G1X4.1127Y4.1047
G1X5.8877
G1Y6.0297
G1X4.1127
G1Y4.1047
G1X4.0627Y4.0547
G1X5.9377
G1Y6.0797
G1X4.0627
G1Y4.0547
G1X4.0127Y4.0047
G1X5.9877
G1Y6.1297
G1X4.0127
G1Y4.0047
G1X3.9627Y3.9547
G1X6.0377
G1Y6.1797
G1X3.9627
G1Y3.9547
G1X3.9127Y3.9047
G1X6.0877
G1Y6.2297
G1X3.9127
G1Y3.9047
G1X3.8627Y3.8547
G1X6.1377
G1Y6.2797
G1X3.8627
G1Y3.8547
G1X3.8127Y3.8047
G1X6.1877
G1Y6.3297
G1X3.8127
G1Y3.8047
G1X3.7627Y3.7547
G1X6.2377
G1Y6.3797
G1X3.7627

G1Y3.7547
G00Z0.2000
G00X4.9627Y4.9547
G1Z-0.0833F10.0
G1X5.0377F35.0
G1Y5.1797
G1X4.9627
G1Y4.9547
G1X4.9127Y4.9047
G1X5.0877
G1Y5.2297
G1X4.9127
G1Y4.9047
G1X4.8627Y4.8547
G1X5.1377
G1Y5.2797
G1X4.8627
G1Y4.8547
G1X4.8127Y4.8047
G1X5.1877
G1Y5.3297
G1X4.8127
G1Y4.8047
G1X4.7627Y4.7547
G1X5.2377
G1Y5.3797
G1X4.7627
G1Y4.7547
G1X4.7127Y4.7047
G1X5.2877
G1Y5.4297
G1X4.7127
G1Y4.7047
G1X4.6627Y4.6547
G1X5.3377
G1Y5.4797
G1X4.6627
G1Y4.6547
G1X4.6127Y4.6047
G1X5.3877
G1Y5.5297
G1X4.6127
G1Y4.6047

G1X4.5627Y4.5547
G1X5.4377
G1Y5.5797
G1X4.5627
G1Y4.5547
G1X4.5127Y4.5047
G1X5.4877
G1Y5.6297
G1X4.5127
G1Y4.5047
G1X4.4627Y4.4547
G1X5.5377
G1Y5.6797
G1X4.4627
G1Y4.4547
G1X4.4127Y4.4047
G1X5.5877
G1Y5.7297
G1X4.4127
G1Y4.4047
G1X4.3627Y4.3547
G1X5.6377
G1Y5.7797
G1X4.3627
G1Y4.3547
G1X4.3127Y4.3047
G1X5.6877
G1Y5.8297
G1X4.3127
G1Y4.3047
G1X4.2627Y4.2547
G1X5.7377
G1Y5.8797
G1X4.2627
G1Y4.2547
G1X4.2127Y4.2047
G1X5.7877
G1Y5.9297
G1X4.2127
G1Y4.2047
G1X4.1627Y4.1547
G1X5.8377
G1Y5.9797

G1X4.1627
G1Y4.1547
G1X4.1127Y4.1047
G1X5.8877
G1Y6.0297
G1X4.1127
G1Y4.1047
G1X4.0627Y4.0547
G1X5.9377
G1Y6.0797
G1X4.0627
G1Y4.0547
G1X4.0127Y4.0047
G1X5.9877
G1Y6.1297
G1X4.0127
G1Y4.0047
G1X3.9627Y3.9547
G1X6.0377
G1Y6.1797
G1X3.9627
G1Y3.9547
G1X3.9127Y3.9047
G1X6.0877
G1Y6.2297
G1X3.9127
G1Y3.9047
G1X3.8627Y3.8547
G1X6.1377
G1Y6.2797
G1X3.8627
G1Y3.8547
G1X3.8127Y3.8047
G1X6.1877
G1Y6.3297
G1X3.8127
G1Y3.8047
G1X3.7627Y3.7547
G1X6.2377
G1Y6.3797
G1X3.7627
G1Y3.7547
G00Z0.2000

G00X4.9627Y4.9547
G1Z-0.1250F10.0
G1X5.0377F35.0
G1Y5.1797
G1X4.9627
G1Y4.9547
G1X4.9127Y4.9047
G1X5.0877
G1Y5.2297
G1X4.9127
G1Y4.9047
G1X4.8627Y4.8547
G1X5.1377
G1Y5.2797
G1X4.8627
G1Y4.8547
G1X4.8127Y4.8047
G1X5.1877
G1Y5.3297
G1X4.8127
G1Y4.8047
G1X4.7627Y4.7547
G1X5.2377
G1Y5.3797
G1X4.7627
G1Y4.7547
G1X4.7127Y4.7047
G1X5.2877
G1Y5.4297
G1X4.7127
G1Y4.7047
G1X4.6627Y4.6547
G1X5.3377
G1Y5.4797
G1X4.6627
G1Y4.6547
G1X4.6127Y4.6047
G1X5.3877
G1Y5.5297
G1X4.6127
G1Y4.6047
G1X4.5627Y4.5547
G1X5.4377

G1Y5.5797
G1X4.5627
G1Y4.5547
G1X4.5127Y4.5047
G1X5.4877
G1Y5.6297
G1X4.5127
G1Y4.5047
G1X4.4627Y4.4547
G1X5.5377
G1Y5.6797
G1X4.4627
G1Y4.4547
G1X4.4127Y4.4047
G1X5.5877
G1Y5.7297
G1X4.4127
G1Y4.4047
G1X4.3627Y4.3547
G1X5.6377
G1Y5.7797
G1X4.3627
G1Y4.3547
G1X4.3127Y4.3047
G1X5.6877
G1Y5.8297
G1X4.3127
G1Y4.3047
G1X4.2627Y4.2547
G1X5.7377
G1Y5.8797
G1X4.2627
G1Y4.2547
G1X4.2127Y4.2047
G1X5.7877
G1Y5.9297
G1X4.2127
G1Y4.2047
G1X4.1627Y4.1547
G1X5.8377
G1Y5.9797
G1X4.1627
G1Y4.1547

G1X4.1127Y4.1047
G1X5.8877
G1Y6.0297
G1X4.1127
G1Y4.1047
G1X4.0627Y4.0547
G1X5.9377
G1Y6.0797
G1X4.0627
G1Y4.0547
G1X4.0127Y4.0047
G1X5.9877
G1Y6.1297
G1X4.0127
G1Y4.0047
G1X3.9627Y3.9547
G1X6.0377
G1Y6.1797
G1X3.9627
G1Y3.9547
G1X3.9127Y3.9047
G1X6.0877
G1Y6.2297
G1X3.9127
G1Y3.9047
G1X3.8627Y3.8547
G1X6.1377
G1Y6.2797
G1X3.8627
G1Y3.8547
G1X3.8127Y3.8047
G1X6.1877
G1Y6.3297
G1X3.8127
G1Y3.8047
G1X3.7627Y3.7547
G1X6.2377
G1Y6.3797
G1X3.7627
G1Y3.7547
G00Z0.2000

G00Z0.8000 (This moves the machine up to the safe clearance height that you set in VCarve. Its optional as the next command is a T0 which automatically raises the machine to full Z height before returning the current tool to the tool changer.)

G00X0.0000Y0.0000 (This will return the machine to X0 Y0 following the cut. This is optional)

M05

M09

M30

%

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

G53 Z0 X0 Y130 (Move machine to "Park" location)

M12C7 (Turn the vacuum off)

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