
vCarve is the software we use to set up the layout of the routing relative to the billet (piece of material), set the feed rates, speeds, depth and tool bits. Then vCarve exports gCode which communicates with the CNC route. The software that communicates with the Router itself is different from vCarve. The one we used is called 'NcStudio AD, Engraving Machine Control System'.

NcStudio software - Sending your files to the machine to cut

Open NcStudio Software. We need to alter the gCode as there are some lines in it that we don't need.

Step 1)

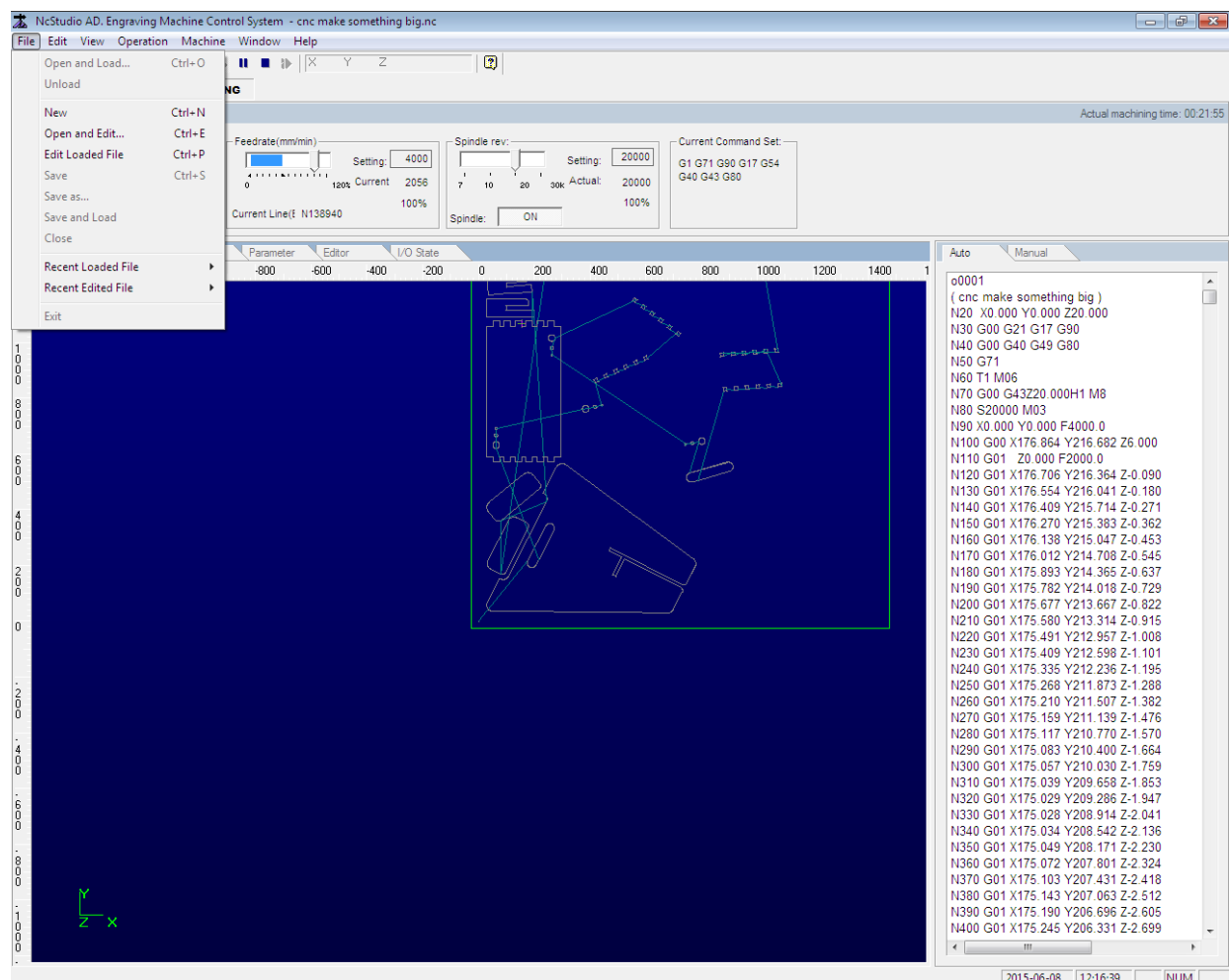
From the file menu - select 'open & Edit' to open the gCode file that you created from vCarve.

Step 2)

Deleted 2 lines G codes from editor (the machine or software in maklab doesn't like it). It's in one of the first 5 lines. Delete: G21 & G98 from the code

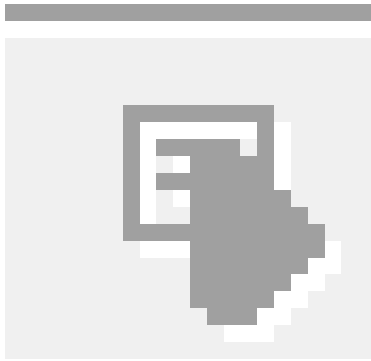
Step 3)

Then from the file menu, select save & load



Step 4)

Then we'll run simulate. Go into the trace window and run - simulate to see what it will cut



NcStudio AD: Engraving Machine Control System - cnc make something big.nc

File Edit View Operation Machine Window Help

AUTO RUNNING RUNNING

NC State Actual machining time: 00:17:35

Ax	M. Coord	W. Coord	Remains
X:	61.488	34.890	-2.371
Y:	68.229	43.681	24.732
Z:	-170.125	-19.000	0.000

Feedrate(mm/min) Setting: 4000
120% Current 3873
Current Line#: N131850

Spindle rev. Setting: 20000
7 10 20 30k Actual: 20000
Spindle: ON

Current Command Set:
G2 G71 G90 G17 G54
G40 G43 G80

Trace

Auto Manual

```
o0001
( cnc make something big )
N20 X0.000 Y0.000 Z20.000
N30 G00 G21 G17 G90
N40 G00 G40 G49 G80
N50 G71
N60 T1 M06
N70 G00 G43Z20.000H1 M8
N80 S20000 M03
N90 X0.000 Y0.000 F4000.0
N100 G00 X176.864 Y216.682 Z6.000
N110 G01 Z0.000 F2000.0
N120 G01 X176.706 Y216.364 Z-0.090
N130 G01 X176.554 Y216.041 Z-0.180
N140 G01 X176.409 Y215.714 Z-0.271
N150 G01 X176.270 Y215.383 Z-0.362
N160 G01 X176.138 Y215.047 Z-0.453
N170 G01 X176.012 Y214.708 Z-0.545
N180 G01 X175.893 Y214.365 Z-0.637
N190 G01 X175.782 Y214.018 Z-0.729
N200 G01 X175.677 Y213.667 Z-0.822
N210 G01 X175.580 Y213.314 Z-0.915
N220 G01 X175.491 Y212.957 Z-1.008
N230 G01 X175.409 Y212.598 Z-1.101
N240 G01 X175.335 Y212.236 Z-1.195
N250 G01 X175.268 Y211.873 Z-1.288
N260 G01 X175.210 Y211.507 Z-1.382
N270 G01 X175.159 Y211.139 Z-1.476
N280 G01 X175.117 Y210.770 Z-1.570
N290 G01 X175.083 Y210.400 Z-1.664
N300 G01 X175.057 Y210.030 Z-1.759
N310 G01 X175.039 Y209.658 Z-1.853
N320 G01 X175.029 Y209.286 Z-1.947
N330 G01 X175.028 Y208.914 Z-2.041
N340 G01 X175.034 Y208.542 Z-2.136
N350 G01 X175.049 Y208.171 Z-2.230
N360 G01 X175.072 Y207.801 Z-2.324
N370 G01 X175.103 Y207.431 Z-2.418
N380 G01 X175.143 Y207.063 Z-2.512
N390 G01 X175.190 Y206.696 Z-2.605
N400 G01 X175.245 Y206.331 Z-2.699
```

Step 5)

If you're happy with this then we need to set up the machine by:

Making sure the machine and software are in sych. We don't know what was run on the machine before or if the emergency stop was used so many the software and machine think they are in different places or one thinks it is running a job still and the other doesn't. To wipe this and make sure the software and machine is in sych you go into the 'Operation' drop down menu. Then go into (shown below). Then select all axis. This will ask the machine to move to the limit switches for x,y and z. Basically the limits of where the machine can travel to, where it hits the limits switches. This then tells the software where the machine's zero zero is for x,y,z. Then close the dialogue box on the screen.

The screenshot displays the NcStudio AD software interface. The 'Operation' menu is open, showing options such as 'Set Workpiece Origin', 'Set Workpiece Coordinate', 'Move to Workpiece Origin', 'Save the Current Workpiece Origin', 'Load the Saved Workpiece Origin', 'Start', 'Pause', 'Stop', 'Enter Simulation Mode then Start Simulating', 'Advanced Start...', 'Resume', 'Advanced MDI...', 'Jiggle...', 'Mobile Calibrator...', 'Move to Reference Point...', and 'Disable Mechanical Limits...'. The 'Start' option is selected. The main window shows a 3D model of a part being machined. The 'Current Command Set' is displayed as G1 G71 G90 G17 G54 G40 G43 G80. The 'Actual machining time' is 00:30:44. The 'Trace' window shows the G-code program, starting with 'o0001 (cnc make something big)' and ending with 'N400 G01 X175.245 Y206.331 Z-2.699'. The status bar at the bottom shows the date 2015-06-08, time 12:25:23, and the NUM key.

Step 6)

Next we set up material and tool.

- screwing down the material
- set up the tool in router
- using jog settings and number pad to move the route until you find the 0,0,0 for z, x and y
- When you have them then press the x, y, z buttons to set them to zero

(***NOTE: first of set the z axis by itself then set the x & y at the same time)

NC State			
Ax	M. Coord.	W. Coord.	Remain
X:	56.602	30.003	-0.003
Y:	1004.094	979.546	79.754
Z:	-158.725	-7.600	0.000

Step 7)

When everything is set up and the extractor/s are on and then water pump for cooling the bit is running we are ready to go

Step 8)

Pressing play button will start the spindle running and the cutting job

Step 9)

If something goes wrong with the routing, to pause the machine via the software press F10 or the pause icon on the ribbon at the top of the software. Otherwise press the big red emergency stop button. As the machine is running the software will show you what the machine is doing, shown below.

The screenshot displays the NcStudio AD software interface. At the top, the title bar reads "NcStudio AD, Engraving Machine Control System - cnc make something big.nc". The menu bar includes File, Edit, View, Operation, Machine, Window, and Help. Below the menu bar is a ribbon with various icons for machine control. The main interface is divided into several sections:

- NC State:** Shows the machine is in "RUNNING" mode. The "Actual machining time" is 00:46:17.
- Coordinates and Parameters:** A table shows M. Coord. (X: 842.970, Y: 808.199, Z: -154.925), W. Coord. (816.372, 783.650, -3.800), and Remains (1.259, 0.272, 0.000). Feedrate is set to 4000 mm/min with a current of 1631. Spindle rev. is set to 20000 with an actual of 20000. Current Command Set includes G1 G71 G90 G17 G54 G40 G43 G80.
- Trace:** A 2D plot showing the tool path in blue on a dark blue background. The axes range from -1600 to 1600 on the X-axis and -1000 to 1000 on the Y-axis.
- Program Editor:** A list of G-code commands from O0001 to N400, including coordinates and feed rates.

```
O0001
( cnc make something big )
N20 X0.000 Y0.000 Z20.000
N30 G00 G21 G17 G90
N40 G00 G40 G49 G80
N50 G71
N60 T1 M06
N70 G00 G43Z20.000H1 M8
N80 S20000 M03
N90 X0.000 Y0.000 F4000.0
N100 G00 X176.864 Y216.682 Z6.000
N110 G01 Z0.000 F2000.0
N120 G01 X176.706 Y216.364 Z-0.090
N130 G01 X176.554 Y216.041 Z-0.180
N140 G01 X176.409 Y215.714 Z-0.271
N150 G01 X176.270 Y215.383 Z-0.362
N160 G01 X176.138 Y215.047 Z-0.453
N170 G01 X176.012 Y214.708 Z-0.545
N180 G01 X175.893 Y214.365 Z-0.637
N190 G01 X175.782 Y214.018 Z-0.729
N200 G01 X175.677 Y213.667 Z-0.822
N210 G01 X175.580 Y213.314 Z-0.915
N220 G01 X175.491 Y212.957 Z-1.008
N230 G01 X175.409 Y212.598 Z-1.101
N240 G01 X175.335 Y212.236 Z-1.195
N250 G01 X175.268 Y211.873 Z-1.288
N260 G01 X175.210 Y211.507 Z-1.382
N270 G01 X175.159 Y211.139 Z-1.476
N280 G01 X175.117 Y210.770 Z-1.570
N290 G01 X175.083 Y210.400 Z-1.664
N300 G01 X175.057 Y210.030 Z-1.759
N310 G01 X175.039 Y209.658 Z-1.853
N320 G01 X175.029 Y209.286 Z-1.947
N330 G01 X175.028 Y208.914 Z-2.041
N340 G01 X175.034 Y208.542 Z-2.136
N350 G01 X175.049 Y208.171 Z-2.230
N360 G01 X175.072 Y207.801 Z-2.324
N370 G01 X175.103 Y207.431 Z-2.418
N380 G01 X175.143 Y207.063 Z-2.512
N390 G01 X175.190 Y206.696 Z-2.605
N400 G01 X175.245 Y206.331 Z-2.699
```

Total time for cutting was 1 hour 17 minutes