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.
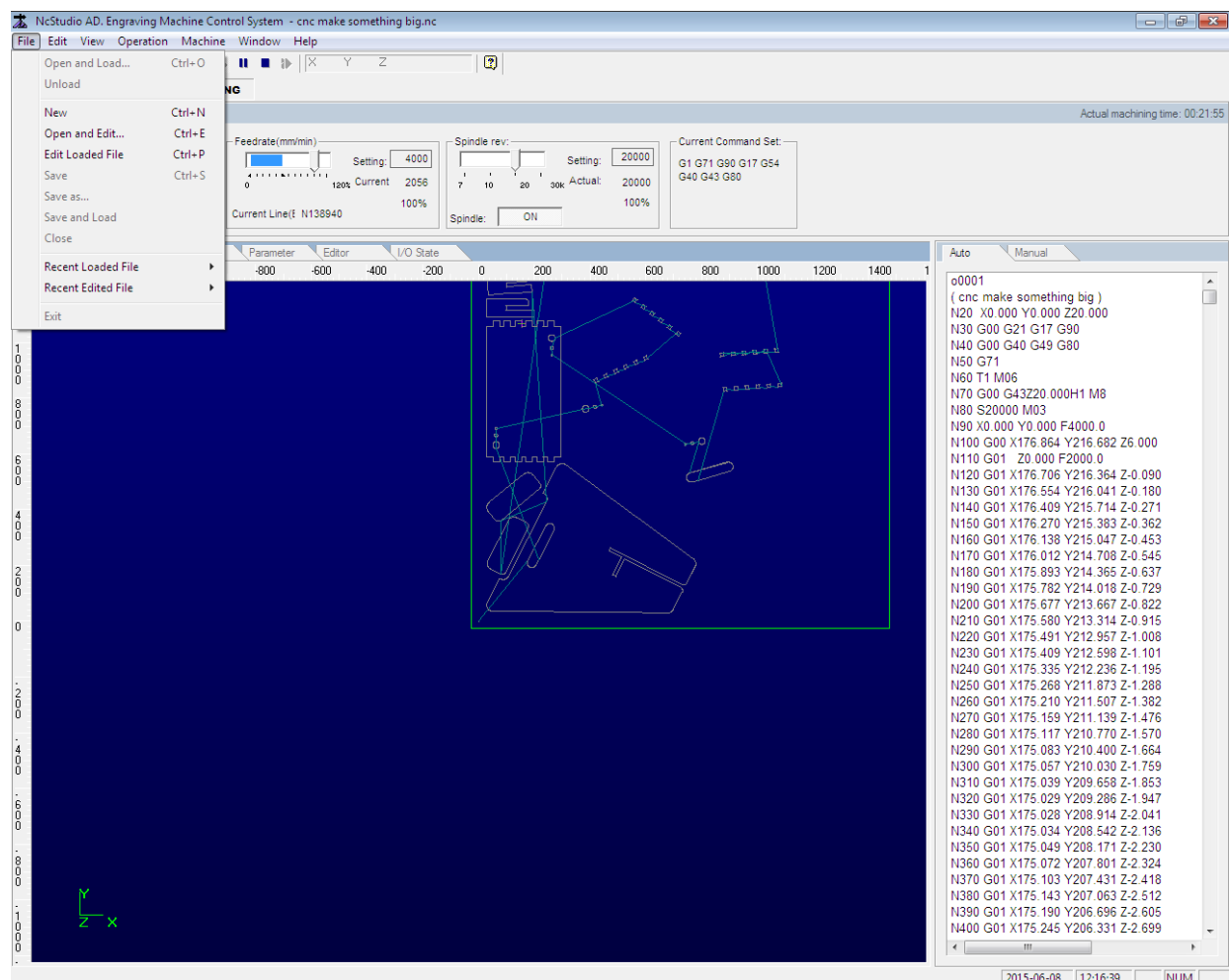
Step 1)

From the file menu - select 'open & Edit' to open the gCocde 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
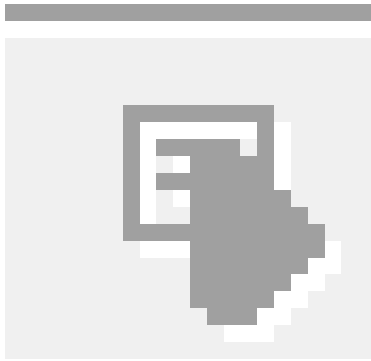
Step 3)

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

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 synch 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.

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			
A›	M. Coor.	W. Coor.	Remaine
✦ 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 spinal running and the cutting job

Step 9)

If something goes wrong with the routering, 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.

Total time for cutting was 1 hour 17 minutes