

## INSTALLATION

Insert the PCB Engraver software CD into the PC

The installation menu should automatically start – if it doesn't, double-click the file **Start.exe**

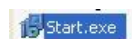
The software requires Microsoft NET Framework v2.0.50727 to be installed on the PC.

If this is not installed, then click the link to **Install the .NET 2 framework**

You may find that the .NET framework will not install with the current version of Windows Installer

– in this case, you need to upgrade using the link to **Upgrade to Windows Installer 3.1**

Now **install the PCB Engraver software** by clicking the link...



**Install .NET 2 Framework**

**Upgrade to Windows Installer 3.1**

**Install PCB Engraver Software**



When installation is complete, you will be asked to insert your software license media. This will normally be provided on floppy disk or USB memory key. Click the removable media option and browse for the file **seckey.tff**

**NB Keep the license disk safe.** We charge for replacement license files. However, the software CD can be downloaded free of charge from [www.denford.co.uk](http://www.denford.co.uk)

## CONNECTING TO THE PCB ENGRAVER

Connect the USB cable from the PC to the machine.

Connect the machine to power outlet using the supplied cable, and power it on.



At this stage, the PC may require USB drivers to be installed.

Do not automatically search for drivers, choose the option to locate them manually.

The Drivers are located on the CD in the **\Drivers** folder.

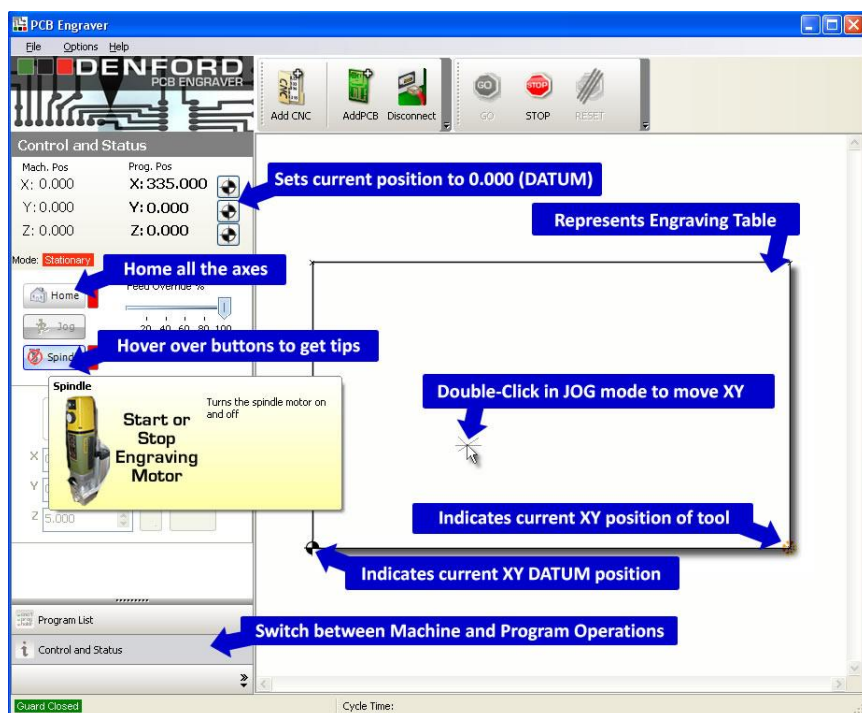
If a warning about windows logo testing appears, click **Continue Anyway**



**NB The software will try to connect to the machine as soon as it runs, so the machine should be powered up and connected before starting the software.**

## RUNNING THE SOFTWARE

From the All Programs menu, locate the Denford software folder and the PCB engraver shortcut:

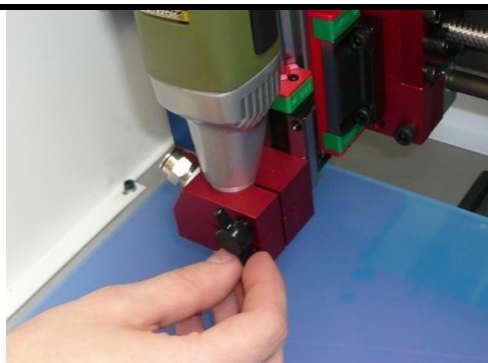


This screen shows the PCB Engraver software layout and some of its features.

Loading and running a CNC program

1. Using double-sided tape, stick your material to the table and set-up the tool in the floating head (see next sections for more detail)
2. Click **Home** to initialise the axes.
3. Click **Jog** to allow manual movement of the axes.
4. Using the keyboard arrow keys for XY and PageUp/PageDown for Z, jog the tool to the bottom left corner of the material.
5. Click the 3 DATUM set buttons next to the position readouts.
6. Click the **Add CNC** button at the top and choose an \*.fnc Denford G code file.
7. The main display will now preview the tool path.
8. Click the **GO** button at the top.

## SETTING UP THE TOOL AND MATERIAL



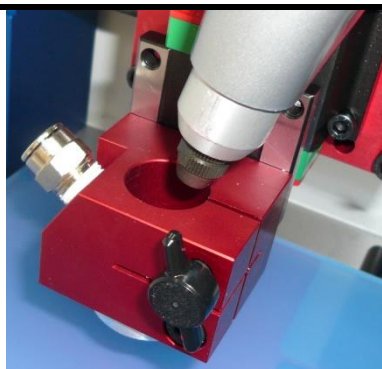
Remove the Spindle Motor from the machine using the thumbscrew as shown. ➡



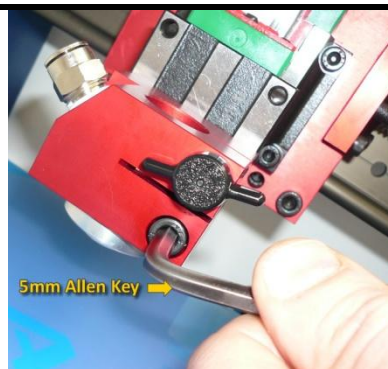
Release tool collet by pressing on the button and turning the spanner supplied. Insert engraving tool and very lightly tighten the collet. ➡



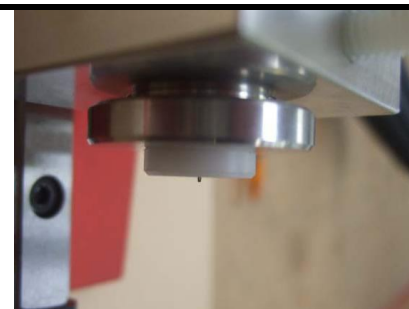
Using the tool setting gauge, push the tool into the spindle on the Z=0 setting. Now fully tighten the collet. ➡



Place the spindle motor back into the floating assembly. Make sure the clamp is clean and free from debris. Retighten the thumb screw. ➡



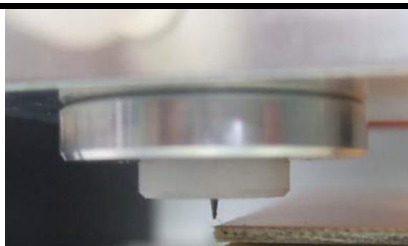
Loosen the cap screw and adjust the floating ring by turning until the tool tip protrudes... ➡




The tool should protrude just enough to break the surface of the engraving material or PCB. Now tighten the cap screw ➡

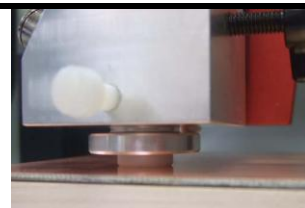



Now that the floating ring has been set, there should be no need to change this again. Simply use the tool depth gauge whenever changing tools in future. ➡

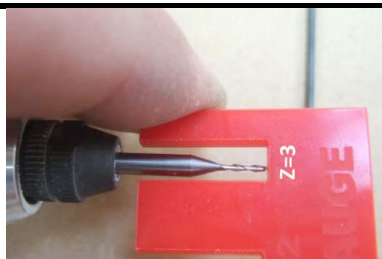


Now fasten the material to the table (usually with quality double-sided tape) and **JOG** the machine to the corner of the material.

Set the X and Y DATUM positions here by clicking  for X and Y ➡



Now turn the spindle motor ON and jog down until the tool goes into the surface and the floating head lifts by a few mm. Now set DATUM for Z  ➡




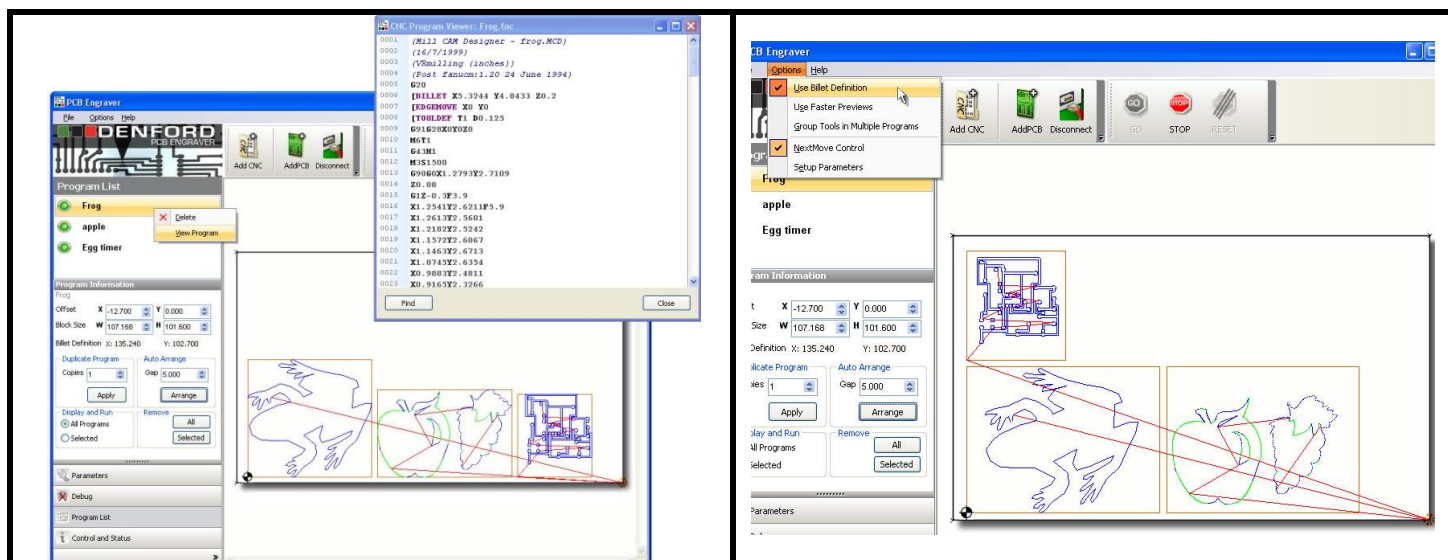
Setting up different tools is now simple-just set the tool with the gauge. If you want to use a drilling tool, then use the gauge to set how far the tool will protrude and thus travel into the material.

## OVERVIEW OF PROGRAM LIST

The PCB software allows as many CNC or Gerber files to be loaded, as can be fitted into the machine's working area.




Click **Add CNC**  to add Denford compatible G Code files

Click **Add PCB**  to start the Gerber PCB file import wizard (see additional tutorial file: *PCB Manufacture Tutorial.pdf* for more information on importing Gerber files via the wizard )



Here 2 CNC files and one Gerber PCB file have been loaded into the software. Right mouse clicking on a program in the Program List allows the program to be viewed or deleted. Various options like the gap between programs can be altered from within the Program List screen.

One option to note is "Use Billet Definition" – if the billet size has been programmed into the file, then this will be used. If not, then the extremes of the tool path are calculated and used to auto-position the files. (Note the difference between the previous and above screenshots).

-  A green LED next to the program/file name indicates that it fits ok and would be machined.
-  A red LED indicates that the program cannot fit into the working area (i.e. anywhere to the right and above the current DATUM position).
-  A yellow LED indicates that the program will fit ok, and is a duplicate of another program in the list.

Individual programs in the list can be selected (highlighted in gold) and re-positioned manually, removed or duplicated

## NOTES

- The PCB Engraving software ignores any programmed feedrates and sets them to a single value, which is defined in machine Parameters (Options menu).
- Even if you have more than one program loaded, you can choose to machine all programs, or any that are selected. (Hold down CTRL while clicking on the list to choose individual or multiple programs)
- When running multiple programs that use more than one tool, the software can combine the programs to reduce the number of tool changes required – to activate this feature, select: Options > **Group Tools In Multiple Programs**
- Visit the PCB Engraver section of the Denford forum for help and ideas: <http://www.denfordata.com/bb>