DENFORD

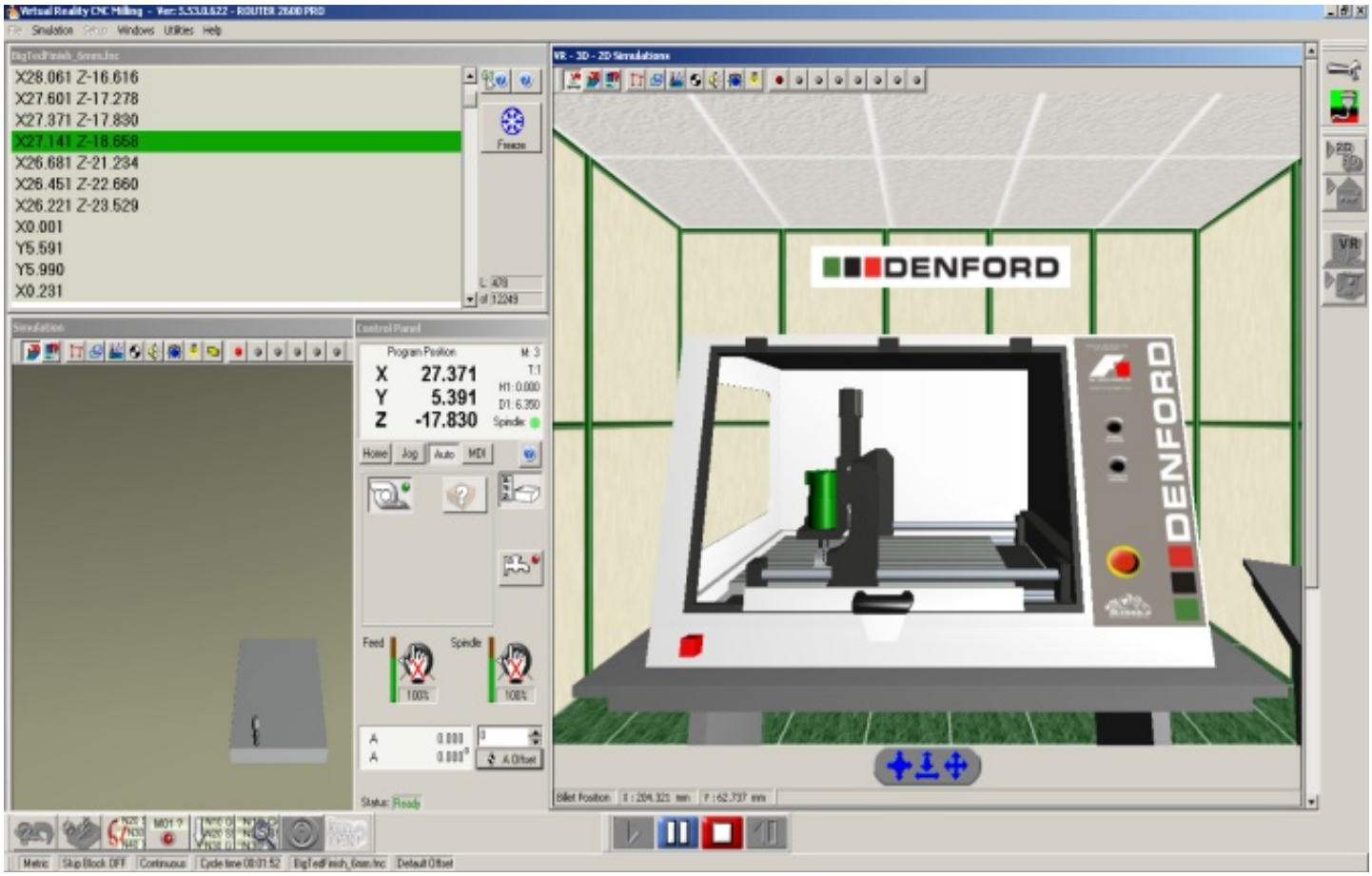
CAD/CAM Solutions & Projects for Education

VR CNC Milling V5

CNC Machine Control Software

Training Guide

VR Milling (V5.61)



IN SCHOOLS

LAND ROVER ABOVE & BEYOND

4 IN SCHOOLS technology challenge

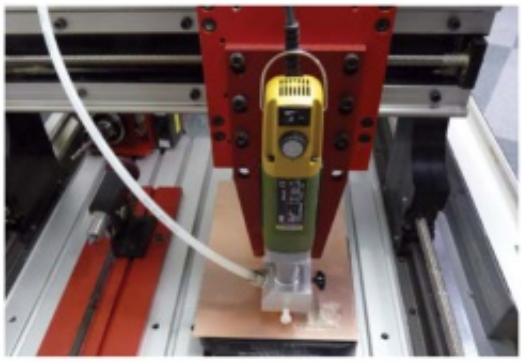
eim ENGINEERING IN MOTION INNOVATIVE EDUCATIONAL PROJECTS

JAGUAR Primary School Challenge

DENFORD

Router Accessories

FLOATING HEAD, AUTOMATIC TOOL CHANGERS, VACUUM BEDS, CLAMPING KITS, FIXTURES AND DUST EXTRACTION UNITS

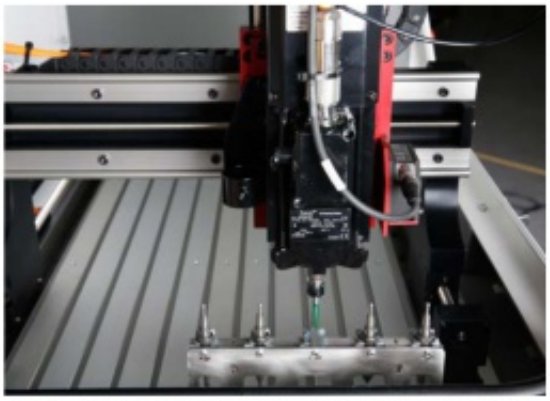


PCB Production with a Floating Head

Denford's 'Floating Head' option permits manufacture of PCB's and engraving of uneven surfaces, and is ideal for batch manufacture of PCB boards.

The floating head comes complete with a quick change facility for a swift interchange with the standard issue router motor.

The cutting tool profiles around the outside of the tracks creating an isolation gap. The weight of the spindle motor plunges the cutter into the PCB board, and depth is set by a plastic disc that floats on the material surface. A float up to 5mm is possible using this technology.



5 STATION AUTOMATIC TOOL CHANGER

The 5 Station Automatic Tool Changer is now **supplied as standard** on the new Router 2600 ATC and Router 6600 ATC and is NOT available as an optional accessory. It comes complete with 5 SK11 Toolholders and 8 collets and offers the following benefits:

- Saves time wasted in repeatedly setting tool offsets
- No additional software required as the 5 station ATC is compatible with all Denford 2D and 3D software

DUST PRO 100 EXTRACTION UNIT

Denford's Large Capacity Dust Extraction system is a purpose-designed dust control system for use with the MRC 40, Compact 1000 Pro, Router 2600/Pro/ATC & Router 6600/Pro/ATC. It can be used as a stand-alone unit, or incorporated within Denford's Universal Machine Bench, as shown above.

The unit is highly effective in removing airborne dust and light particles produced during machining, and is recommended for schools where MDF is regularly used. The unit comes ready to use including a removable / re-usable dust collection bag and separate HEPA filter.

Dimensions: H530mm W460mm D670mm
H20.87in W18.11in D26.38in

DUST PRO 50 EXTRACTION UNIT

Particle and dust extraction unit suitable for use with the MRC 40, Compact 1000 Pro and Router 2600/Pro/ATC. This purpose designed unit is ideal for extraction of airborne dust created during the manufacturing process, and also to vacuum the machine after the cutting process is complete.

The unit comes complete with castors, flexible hose and fittings.

Dimensions: H530mm W300mm D300mm
H20.87in W11.81in D11.81in

Contents

About 4
Introduction 4
Launching VR CNC Milling 5 5
Using the Graphical User Interface 6
Opening a Toolpath 6
The Editor Window 7
Utility and Output Toolbar 8
Utility Toolbar 8
Output Toolbar 9
- •3D Simulation Window 9
- •Post Processor Output 9
- •Virtual Machine 10
- •Real Machine 10

Options Toolbar 11
- •Units 11
- •Offsets / Tools 11
  - •Work Offsets 12
    - •Zeroing the Axis 12
    - •Datum Offsets 13
  - •Tooling Data 14
    - •Adding Tools to Tooling Data 14
  - •Tool Library 15
- •Block Skip 16
- •Optional Stop 17
- •Single Step / Continuous 18
- •Block Search 19
- •Program Info 20
- •EasyScan 3D 22

Control Panel 23
- •Info Window 23
- •Home Tab 23
- •Jog Tab 24
- •Auto Tab 25
  - •Turbo Mode 25
  - •Material Override Mode 25
- •MDI Tab 26

File Control Buttons 27
Opening a DXF 28
- •Follow Cut 31
- •Inside Offset Cut 32
- •Outside Offset Cut 33
- •Area Clearance Cut 34
- •Raster Clearance Cut 35
- •Drill Cut 36

Running a Program 39
Opening a Gerber 40

About VR CNC Milling 5

Virtual Reality (VR) CNC Milling 5 is an improved and updated version of our CNC machine control software, incorporating Denford PCB Manufacturing Software and 2D DXF import facilities, together with USB connectivity, delivering machining times up to 40% faster than before. Enhanced features provide the user with new machining capabilities, simplified options in datum setting, improved tool and work offset features and a new, powerful, virtual reality 3D simulation engine.

Introduction

The aim of this training guide is to teach you how to navigate your way around VR CNC Milling 5 and instruct you how to operate this software to operate your router or mill.
This guide will cover operation of your machine, setting offsets, operation of the virtual reality machine, opening tool-paths, and importing 2D DXF and GERBER files.

This guide makes use of screen shots where possible and will use the following conventions:

Instructions will be in this format

Text to be typed will be in this format

Any software buttons to be pressed, a picture of the button will follow the instruction

This guide assumes that your software has already been installed and your machine has been commissioned.
If any of the features described in this guide are not operating as described please check that the version number you are using is the same as that shown on the front cover.
Version is written on the title bar of the main software window.

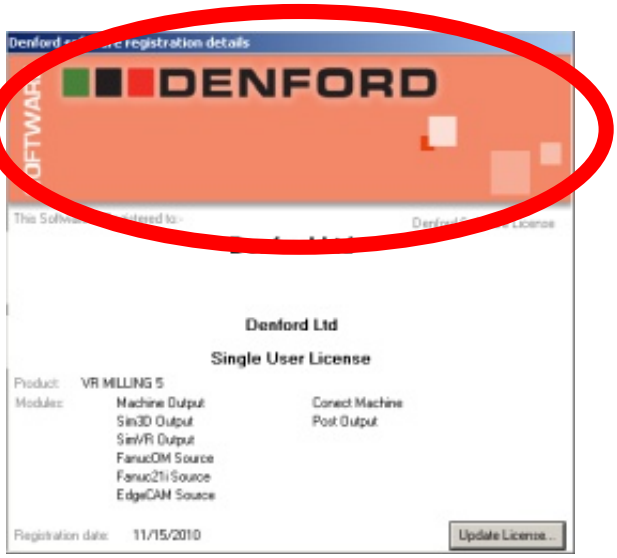
Denford provide machine training and it is recommended that you undertake the training and use this guide as a revision guide after completion of the machine training.

Launching VR CNC Milling 5

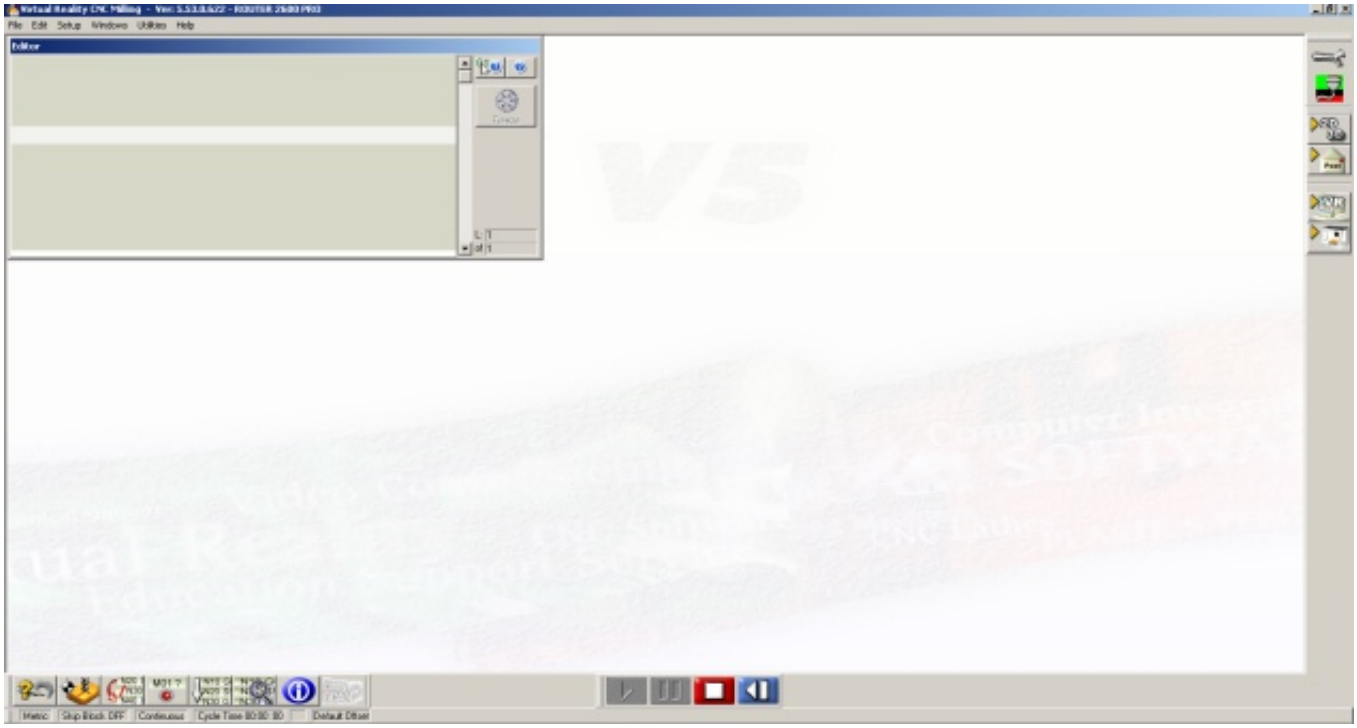
Open the "Denford Applications" folder.
"Double click" on the VR Milling V5 icon.

The screen shown on the right will be displayed and the software will take a minute or 2 to open.
You can force the software to open quicker by following the next instruction.

"Double click" on the area circled below.



The software will open and you will be greeted with the screen below.



Using the Graphical User Interface

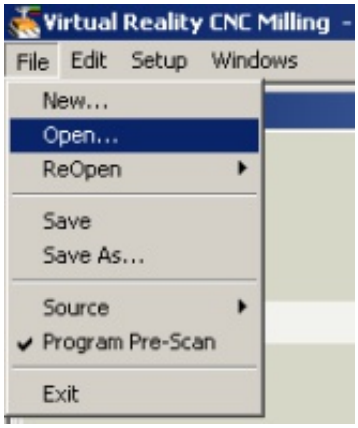
The software opens in a window which should be feel familiar to Windows Users.
There is a title bar at the top, the options shown here will vary according to which window is open in VR CNC Milling 5
Some of the buttons on the right hand side of the main window and some of the buttons along the bottom left of the main window will open new windows. Selecting the same button that opened the window will close it again.
All buttons have tool tips active, so hovering over them with the mouse will bring up a description of that buttons function.

All settings are saved to "c:/ProgramData/Denford/" this is a hidden folder and all users will require full read / write permissions in order to use this software.

Opening a Toolpath

To open a toolpath you need to use the toolbar at the top of the main window.

"Left click" File, then "Left click" Open

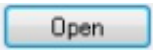


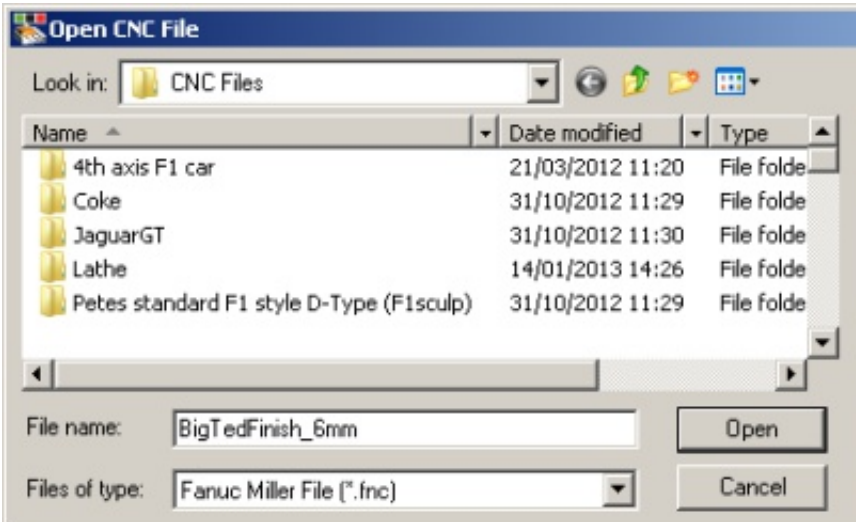
Sample files are located under "Documents/Denford/CNC Files/"
The file extension is .FNC (Fanuc Numerical Control)

Select an FNC file

It does not matter which one but you must have a file open to follow the rest of the tutorial.

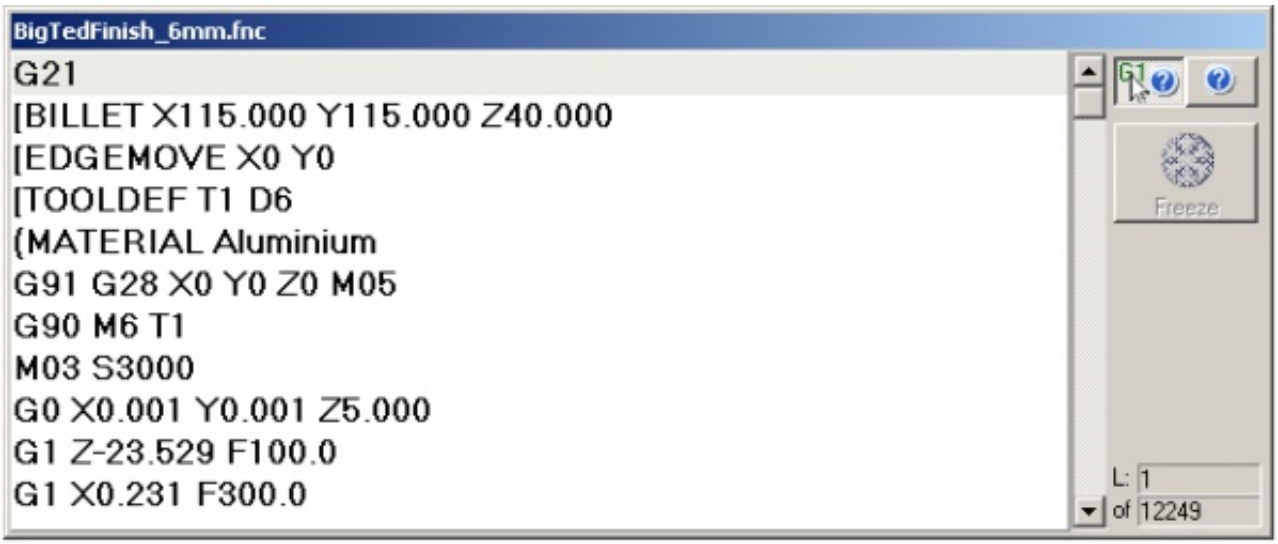
"Left click" the Open button





The Editor Window

This window displays the tool path you have just opened, you can edit the text here just as you would with any text editor.
The editor tells you how many lines the program has and the line number currently selected.



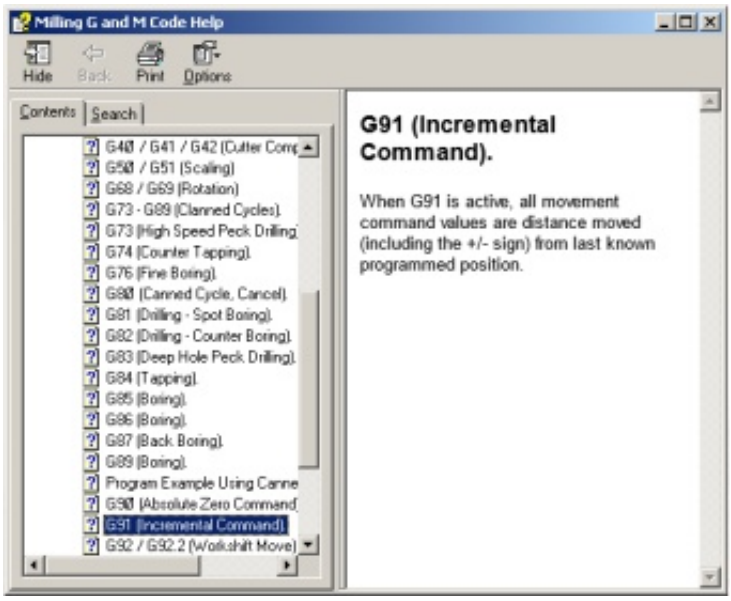
There are a few other options you can use here.

"Left click" this button

The mouse pointer will change to this

"Left click" on one of the "G" or "M" codes

This will launch the Milling G and M Code Help



"Left click" this button

This will launch VR Milling Help for the Editor

This button is greyed out

It is only active when running a program. It freezes the screen while running a program and will free up memory on older PC's and make the CNC machine run more smoothly.

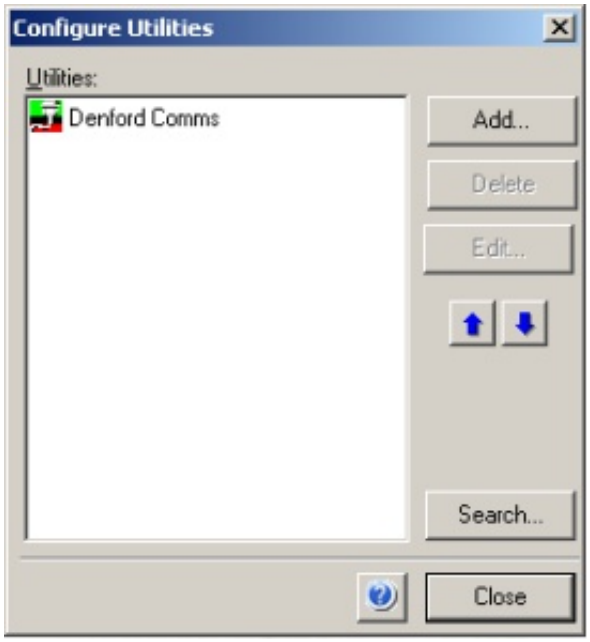
Utility and Output Toolbar

These are the buttons on the right hand side of the main window.
The top 2 buttons are the utility buttons and the remaining buttons are for output.

Utility Toolbar

This button is for configuring the utilities short cuts, you can use it to add short cuts to other programs.

"Left click" the button



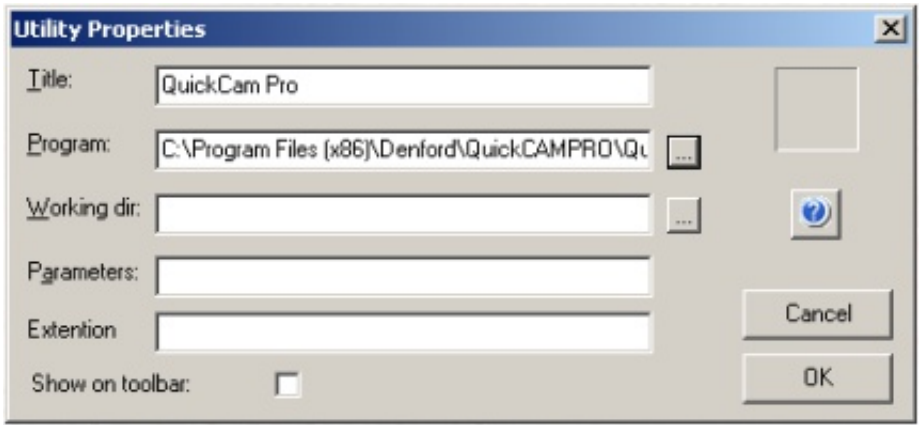
The window shown below will be launched
By default there is a short cut to Denford Comms, which is this button on the toolbar
If your machine is connected to your PC via USB it is not needed so delete it

To add another short cut follow the procedure below

Adding QuickCam Pro Short cut

"Left click" the Configure Utilities button
"Left click" the Add button

The window shown blow will appear



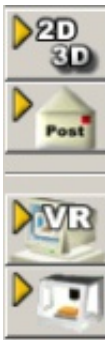
Under Title type the following:
QuickCam Pro
Under Program, enter the path to the QuickCam Pro application:
C:\Program Files (x86)\Denford\QuickCAMPRO\QuickCAMPRO.exe
Ensure that Show on toolbar is ticked
"Left click" the OK button

"Left click" the Close button

You have now added a short cut for QuickCam Pro to the utility toolbar, this process can be repeated for any other program you may wish to access from VR CNC Milling 5

Output Toolbar

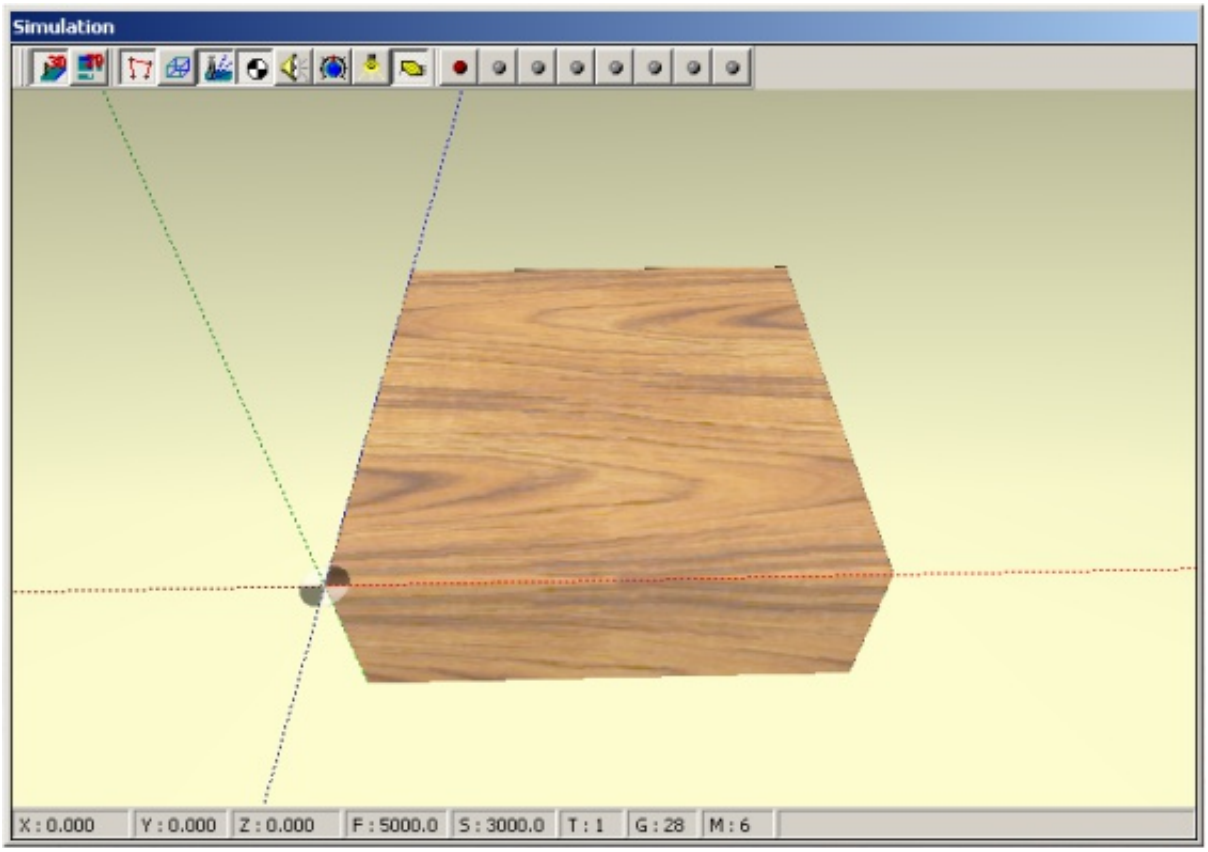
This toolbar has the 4 buttons shown on the right.
Listed below in descending order are their functions:



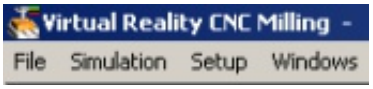
- 3D Simulation window
- Post Processor output
- Virtual machine
- Real machine

3D Simulation Window

This window will show you either a 2D or 3D simulation of the program that is loaded in the editor window, the window is shown below.



When the above window is selected the main window toolbar will show a "Simulation" option, this is used to adjust the settings of the simulation.

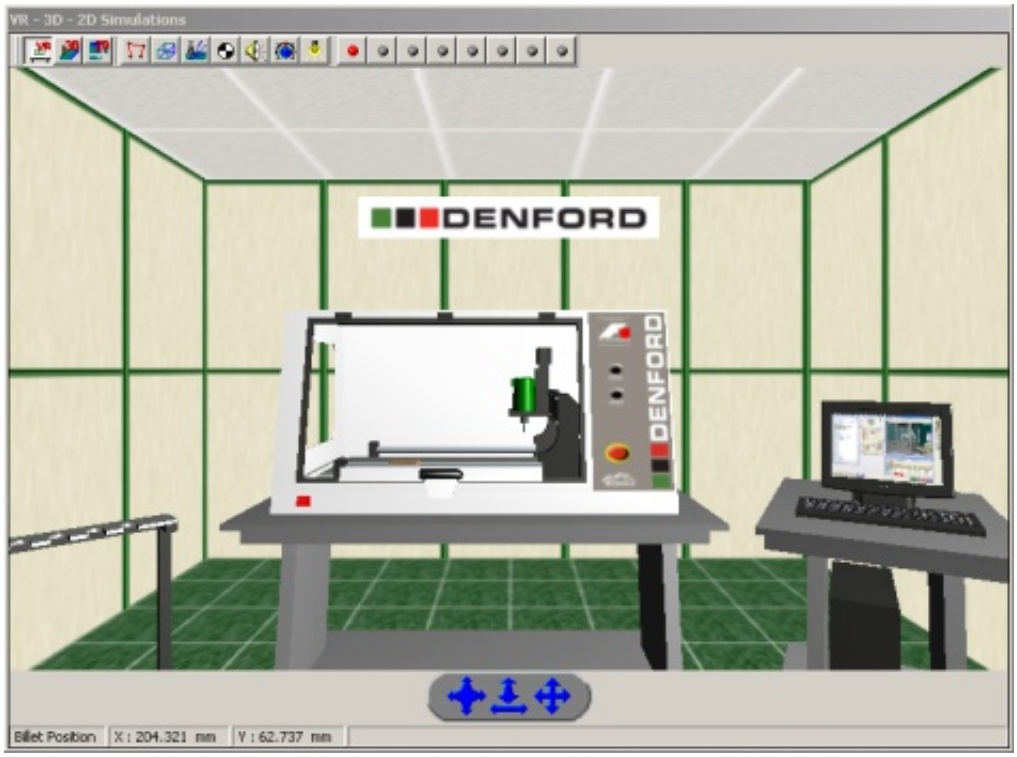


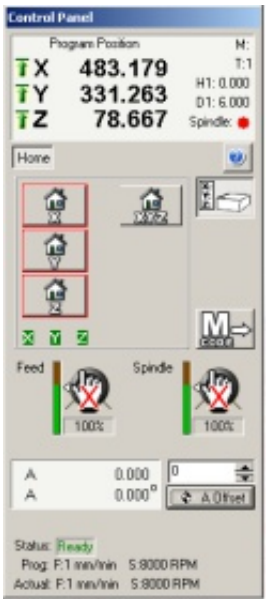
Post Processor Output

Selecting this option will enable you to save your tool path as a .GNC file which was used by some of our older machines.If you wish to create a tool path for a different make of machine, this can be done in QuickCam Pro and QuickCam 2D.

Virtual Machine

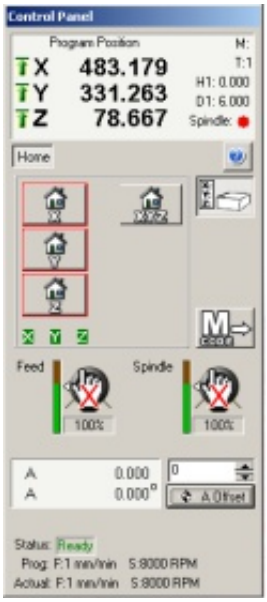
This will launch the windows shown below





This virtual environment enables you to simulate running a CNC machine without the added costs of materials and broken tooling.

Real Machine



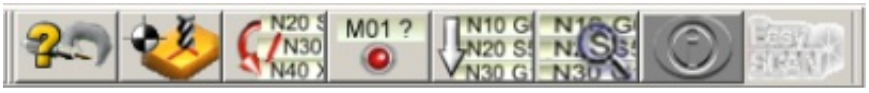
This will establish a connection to the real machine and launch the control panel shown on the right.

If the CNC machine is not connected to the computer and powered on at this stage you will get an error message if you attempt to establish a connection.

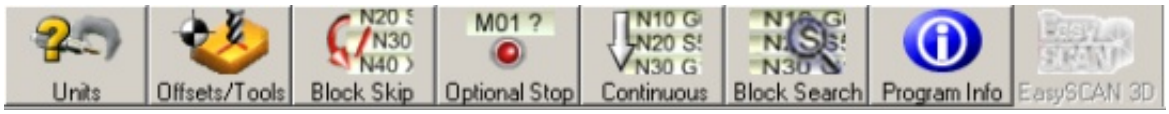
You will find a detailed description on how to use the Control Panel later in this manual

Options Toolbar

The toolbar on the bottom left of the screen is the options toolbar as seen below, with a description of each option from left to right.



"Right click" on the end of the toolbar to display toolbar captions

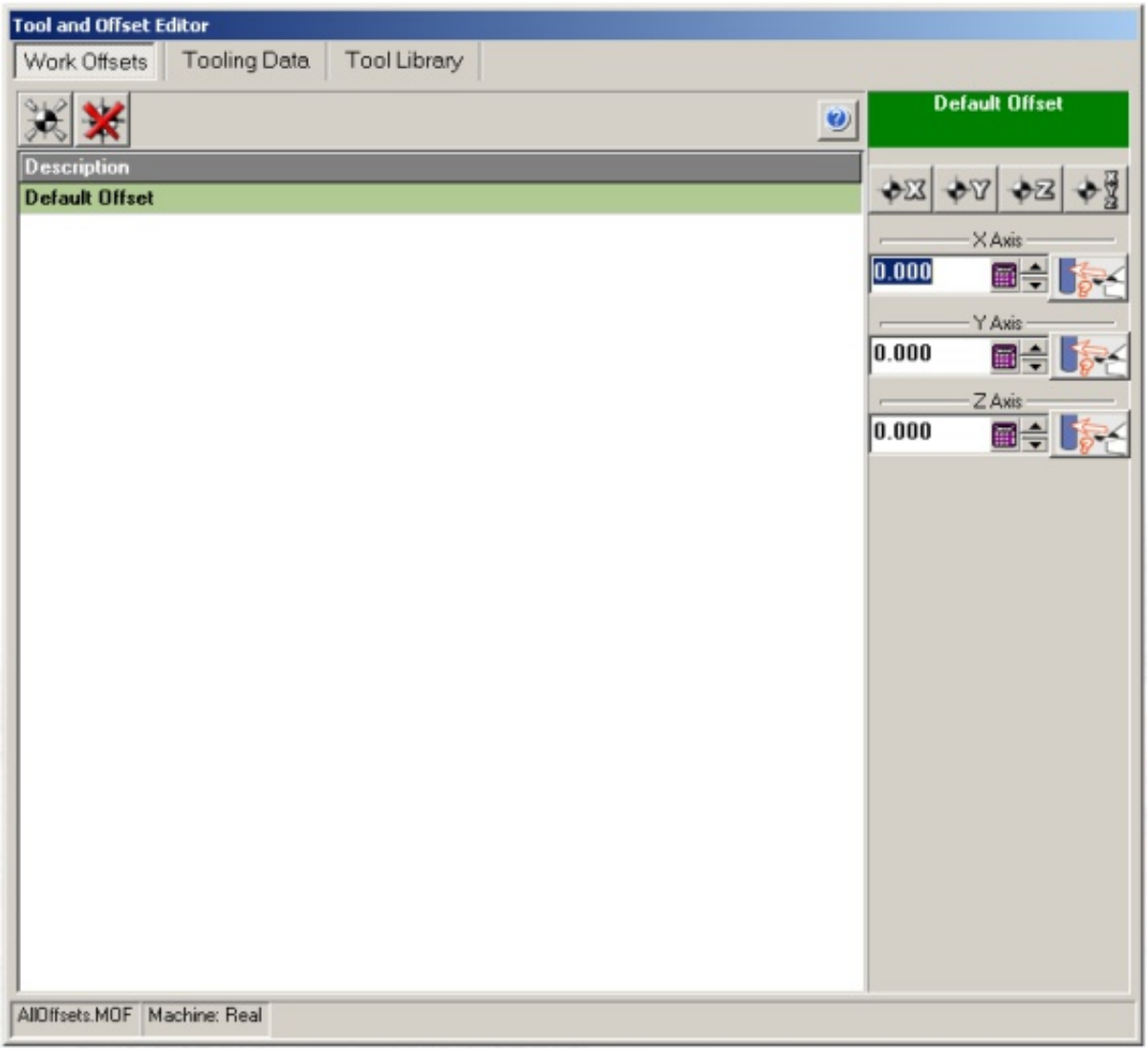


Units

This button changes from metric to inches

Offsets/Tools

This button opens the Tool and Offset Editor window shown below. This window has 3 tabs.



Work Offsets

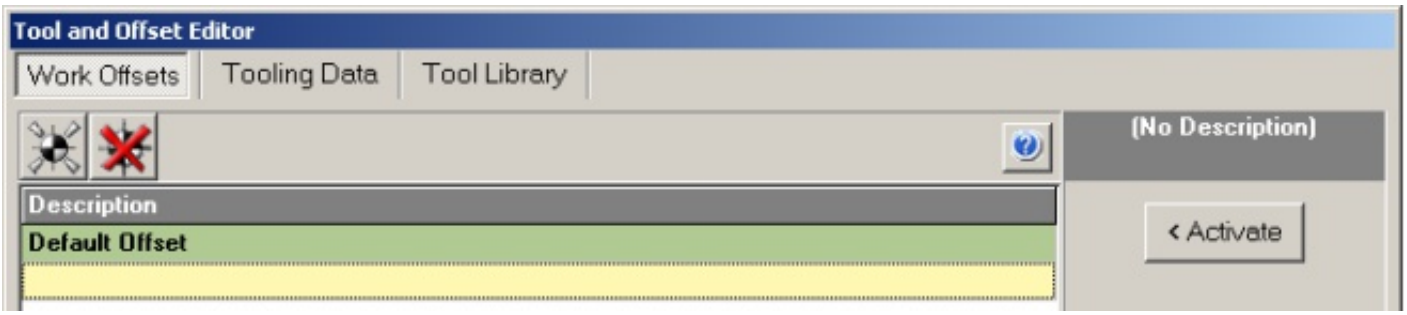
This is where the offsets are set. Depending on the bed size of your machine you can have a range of fixtures fitted to your machine and it is here that you create offsets for them.
It is necessary to create offsets so that the machine knows where the billet is. You can place the billet anywhere on the bed and it is only by setting the offsets that the machine can find the billet.

These 2 buttons can be used for creating or deleting an offset.

"Left click" this button to create an offset

A new line will appear below

"Left click" the new line and it will turn yellow, as shown below.



"Left click" in the yellow region and the bar will go white with a text entry cursor

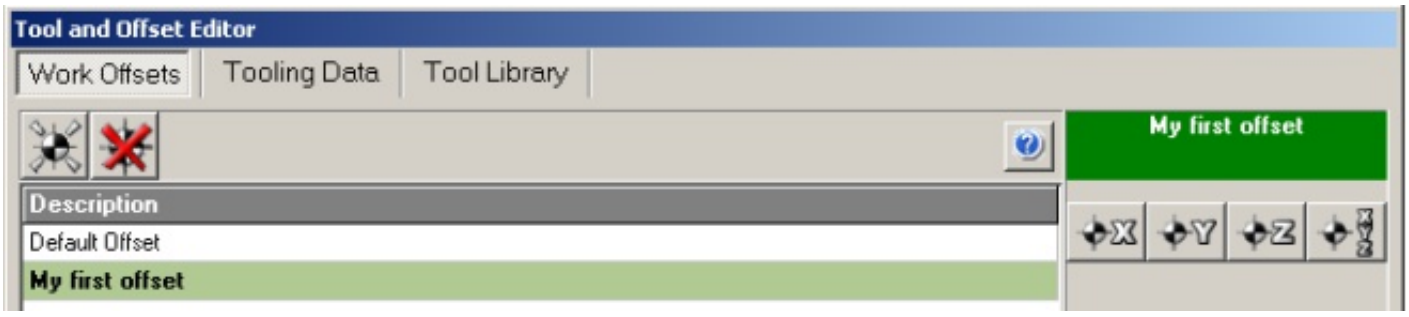
Name it the following

My first offset

"Press enter"

"Left click" the Activate button

"My first offset" should now be highlighted green, this shows that it is active

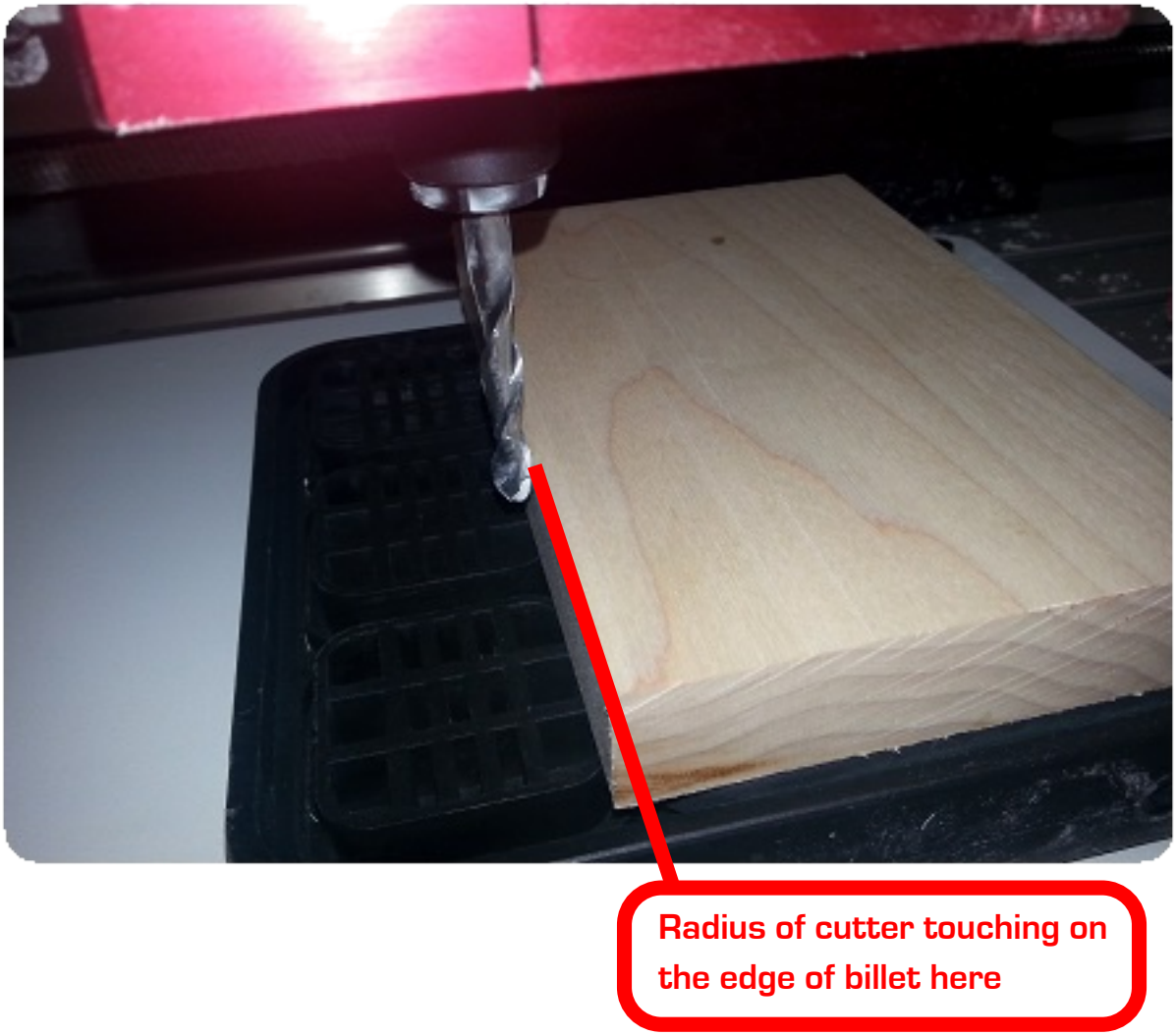


Zeroing the Axis

The buttons shown below will set the current axis position to zero which will be your datum point for the currently active offset.
There are 4 buttons which are in order of listing from left to right for the X, Y, Z, and XYZ axis.

Datum Offsets

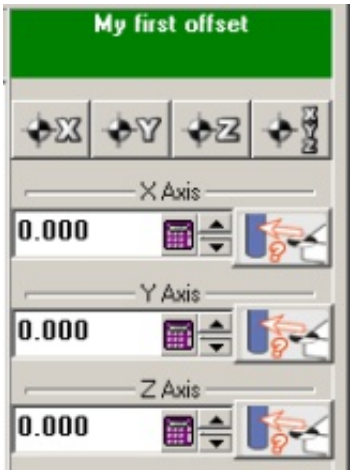
Zeroing the axis by setting the tool position to zero may be the right thing to do for some jobs but there will be other times when it is more accurate to touch the tool on the side of the work piece and offset the datum from this position by the radius of the tool as shown below.

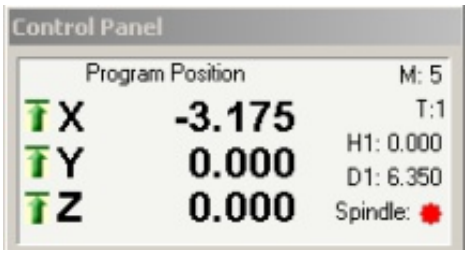


Now that the tool is touching the left hand side of the billet, you will need to set the datum with an offset of the tools radius, there is a button for this.

"Left click" the "Set datum offset from current position (X axis)" button

Datum for X axis is now set and current position is offset by the tool radius as can be seen by the display on the Control Panel

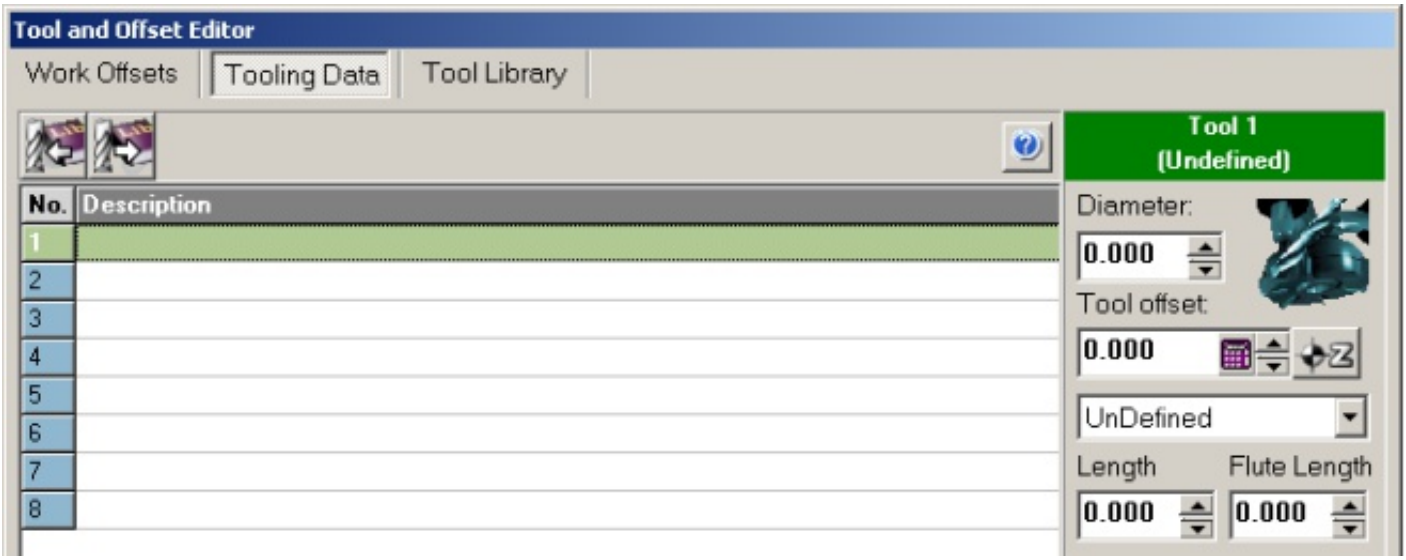




Now repeat this process to set the Y axis by touching on the front of the block

Tooling Data

The second tab on the Tooling and Offset Editor is the Tooling Data tab.



This is the area where you store the tooling to be fitted to your machine. If you have the automatic tool change (ATC) then the tool change will be done for you, if you do not have the ATC you will need a way of reliably setting the tool height to perform a manual tool change in programs that use multiple tools.

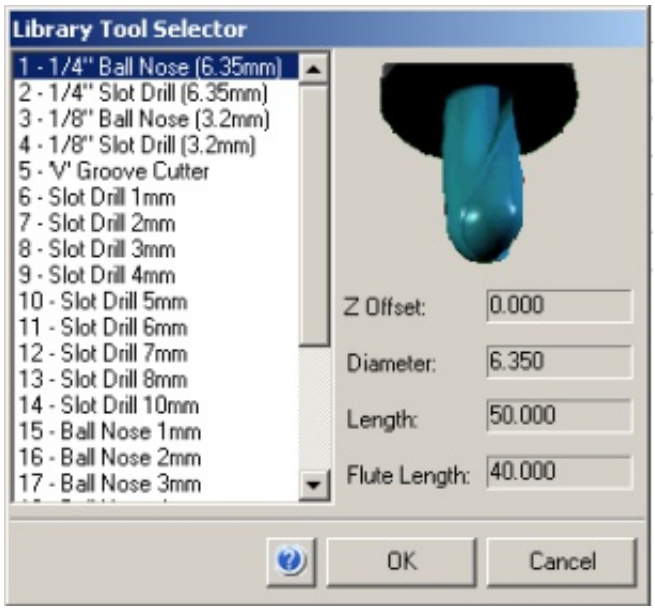
Adding Tools to Tooling Data

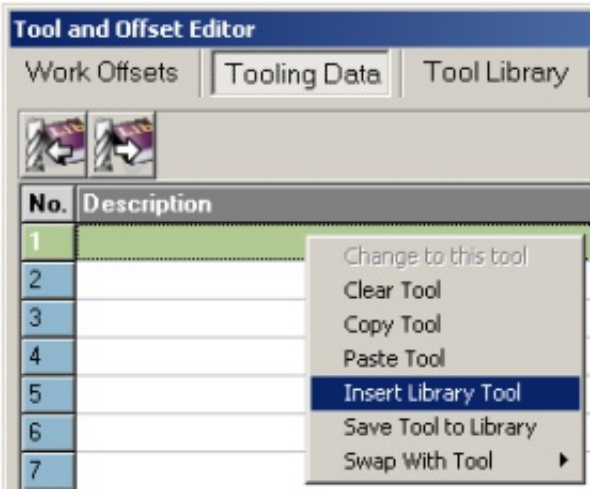
The most commonly used tool is a 1/4" (6.35mm) Ball Nose, add this to the tooling library by doing the following:

"Right click" on slot 1

"Left click" on Insert Library Tool

"Left click" on 1/4" Ball Nose





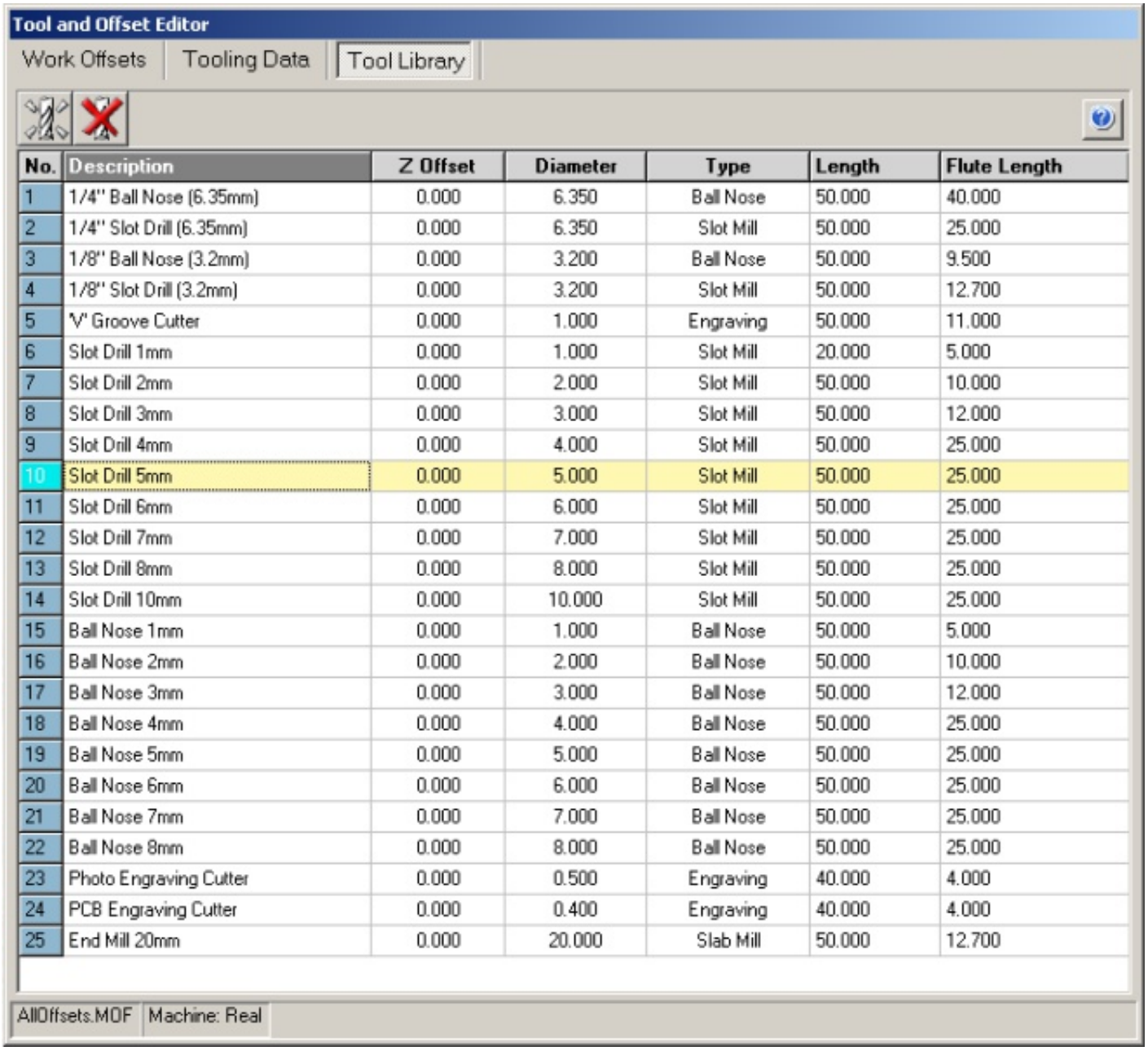
"Left click" the OK button

The 1/4" Ball Nose cutter is now fitted as tool number 1

Repeat this process for the other tools you have.

Tool Library

The third tab on the Tool and Offset Editor window is the Tool Library.



No.	Description	Z Offset	Diameter	Type	Length	Flute Length
1	1/4" Ball Nose (6.35mm)	0.000	6.350	Ball Nose	50.000	40.000
2	1/4" Slot Drill (6.35mm)	0.000	6.350	Slot Mill	50.000	25.000
3	1/8" Ball Nose (3.2mm)	0.000	3.200	Ball Nose	50.000	9.500
4	1/8" Slot Drill (3.2mm)	0.000	3.200	Slot Mill	50.000	12.700
5	'V' Groove Cutter	0.000	1.000	Engraving	50.000	11.000
6	Slot Drill 1mm	0.000	1.000	Slot Mill	20.000	5.000
7	Slot Drill 2mm	0.000	2.000	Slot Mill	50.000	10.000
8	Slot Drill 3mm	0.000	3.000	Slot Mill	50.000	12.000
9	Slot Drill 4mm	0.000	4.000	Slot Mill	50.000	25.000
10	Slot Drill 5mm	0.000	5.000	Slot Mill	50.000	25.000
11	Slot Drill 6mm	0.000	6.000	Slot Mill	50.000	25.000
12	Slot Drill 7mm	0.000	7.000	Slot Mill	50.000	25.000
13	Slot Drill 8mm	0.000	8.000	Slot Mill	50.000	25.000
14	Slot Drill 10mm	0.000	10.000	Slot Mill	50.000	25.000
15	Ball Nose 1mm	0.000	1.000	Ball Nose	50.000	5.000
16	Ball Nose 2mm	0.000	2.000	Ball Nose	50.000	10.000
17	Ball Nose 3mm	0.000	3.000	Ball Nose	50.000	12.000
18	Ball Nose 4mm	0.000	4.000	Ball Nose	50.000	25.000
19	Ball Nose 5mm	0.000	5.000	Ball Nose	50.000	25.000
20	Ball Nose 6mm	0.000	6.000	Ball Nose	50.000	25.000
21	Ball Nose 7mm	0.000	7.000	Ball Nose	50.000	25.000
22	Ball Nose 8mm	0.000	8.000	Ball Nose	50.000	25.000
23	Photo Engraving Cutter	0.000	0.500	Engraving	40.000	4.000
24	PCB Engraving Cutter	0.000	0.400	Engraving	40.000	4.000
25	End Mill 20mm	0.000	20.000	Slab Mill	50.000	12.700

There are 25 tools added to the library by default. Should you wish to add more tools to the library, or delete the tools you do not have, you can use the create or delete buttons.

Block Skip

There may be times when you may want to create 2 parts that are quite similar.
Rather than creating 2 separate programs for them you could use Block Skip to turn off certain parts of the program.

For example, imagine you are cutting a profile from a sheet of material and this is a part you make on a regular basis. Sometimes the part needs holes drilling in it, and sometimes it does not.
By editing the one program you could use Block Skip to turn off the drilling features by adding a forward slash to the lines that you wish block skip to miss out.
The code below has been edited, and with Block Skip on, the highlighted lines will be skipped.

```
G21
G90
[BILLET X150.000 Y100.000 Z10.000
[EDGEMOVE X0 Y0
G91 G28 X0 Y0 Z0 M05
[TOOLDEF T0101 D6.350
M5
G90 M6 T0101
M03 S23000
/(Drill 5.000mm Deep - T1 6.350mm Diam.
/G00 X100.000 Y50.000
/G00 Z2.000
/G01 Z-5.000 F1250.0
/G00 Z2.000
/G00 X50.000
/G01 Z-5.000 F1250.0
/G00 Z2.000
(Follow 3.000mm Deep - T1 6.350mm Diam.
G00 X0.000 Y0.000
G01 Z-3.000 F1250.0
G01 X150.000 F5000.0
Y100.000
X0.000
Y0.000
G00 Z2.000
(Follow 3.000mm Deep - T1 6.350mm Diam.
G00 X20.000 Y20.000
G01 Z-3.000 F1250.0
G01 X130.000 F5000.0
Y80.000
X20.000
Y20.000
G00 Z2.000
G0 Z2.000
G91 G28 X0 Y0 Z0 M05
G90
M30
```

Optional Stop

There may be times when you will want to stop the machine part way through a program to get access. For example, whilst cutting a deep pocket you may wish to stop several times to remove the swarf from the pocket, or you may wish to clean the part after roughing so that students can see what is happening on the finishing pass. For this we have the Optional Stop button shown below.

To use the Optional Stop feature simply add the following to your program where you wish the machine to stop:

M01

When the Optional Stop button is turned on the machine movement will stop. Pressing the "Play" button will restart the machine from where it was when the Optional Stop command was called.

The Optional Stop will only work when the Optional Stop button is active, as seen below.

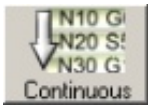
If you wish to stop the Optional Stop feature to get access to the machine you will need to stop the spindle before the door will unlock.

Add the following to your program:

G00 Z5 (Rapid movement to 5mm above billet
M5 (Spindle stop
M01 (Optional Machine stop
M3 (Spindle start
G01 Z-5 F1250 (Linear Interpolation, back to height tool was when it lifted at feedrate of 1250, this may not be Z-5 for your program

Single Step / Continuous

By default the program will be in Continuous mode, the Step button shown below will switch to Single Step mode.

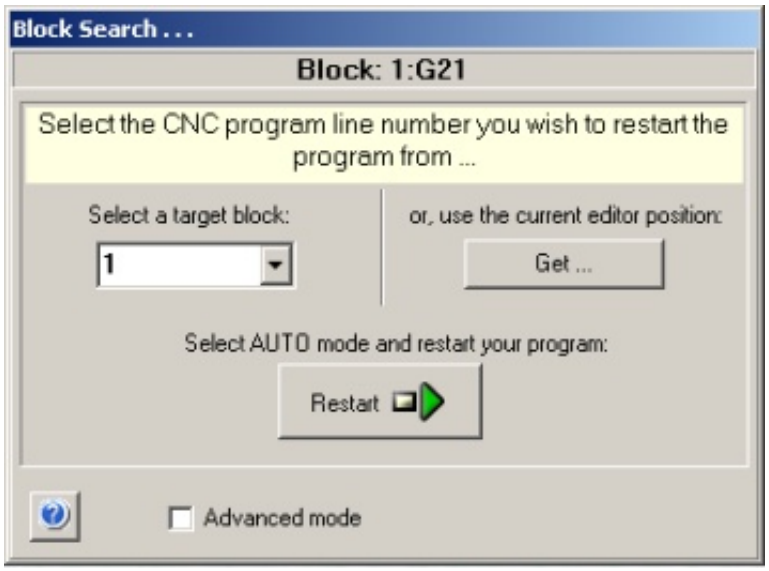


When Single Step mode is enabled the machine will pause after each line, to execute the next line you need to press the "Play" button

You may wish to run your program in Single Step mode if it is a new program that you have written manually without the aid of CAM (Computer Aided Manufacture) software program. I would not recommend using this mode on 3D contouring programs as they can be in excess of 20 000 lines and running in Single Step mode may be tiring.

Block Search

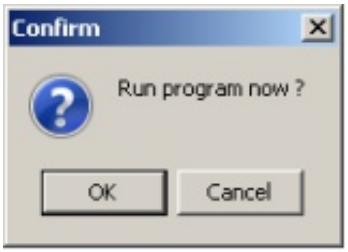
There may be times when you will have to stop the machine before a program is finished, if this is a particularly long program and you were a long way into it, you would be loath to start it again from the beginning.
This is where Block Search comes in. Pressing the Block Search button, shown below, will bring up the Block Search menu, also shown below.



To use Block Search, enter the line number you wish to restart from, enter it in the "Select a Target Block" form in the Block Search window, make sure you are in Auto mode on the Control Panel.

"Left click" the "Restart" button

The door will lock, the spindle will start, and the tool will move to the position it should be in for that line number. Then the Confirm window will pop up.



"Left click" Run to continue the program from this point.

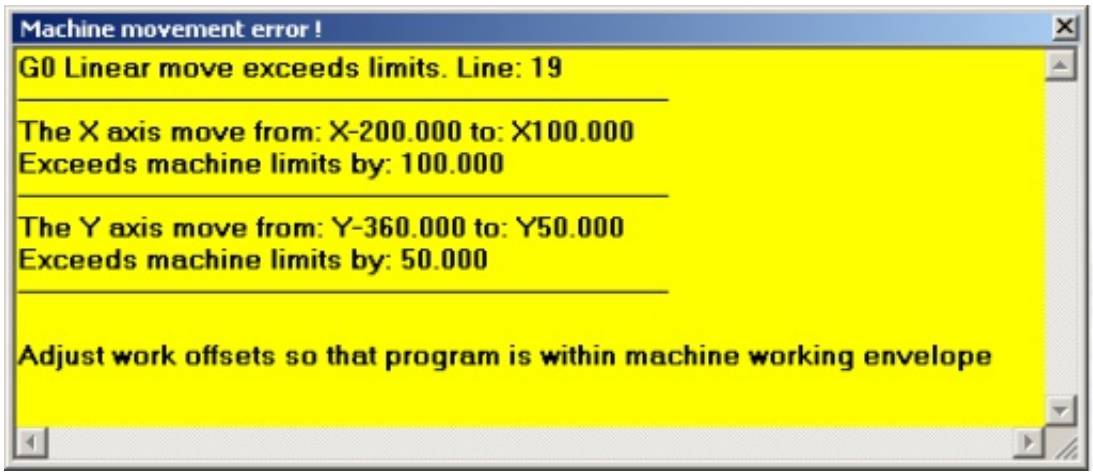
Program Info

This button will scan your program and ensure that the machine is able to move to all the coordinates that the program calls for, if it encounters a movement which exceeds the machines travel it will tell you.

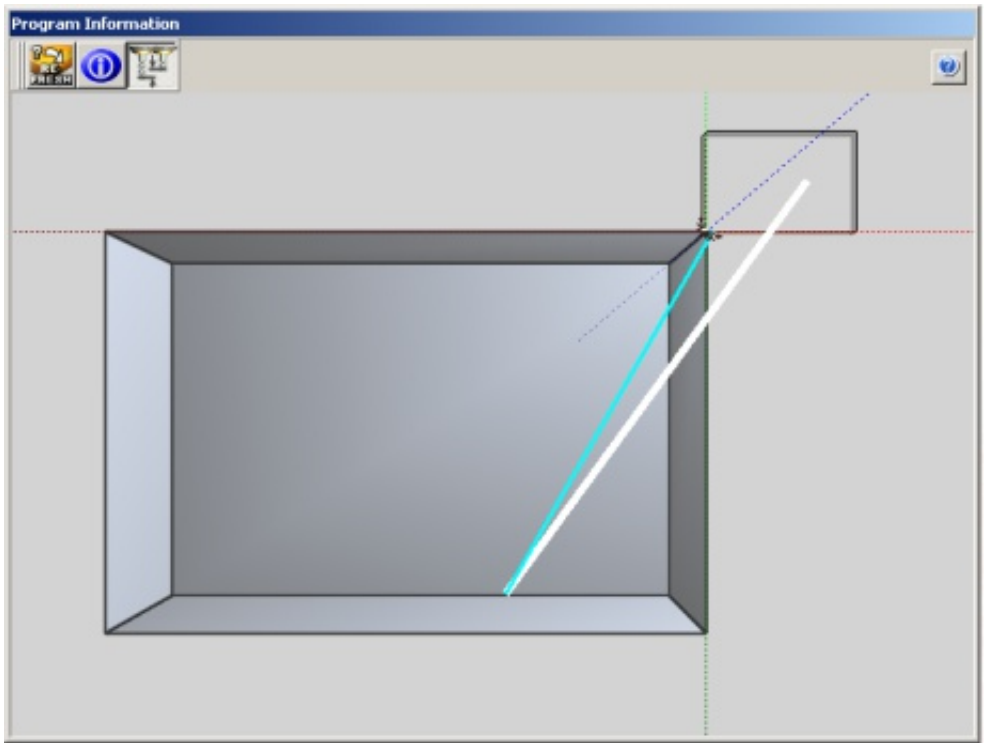
You should select this option after setting your Work Offset.

"Left click" the Program Info button

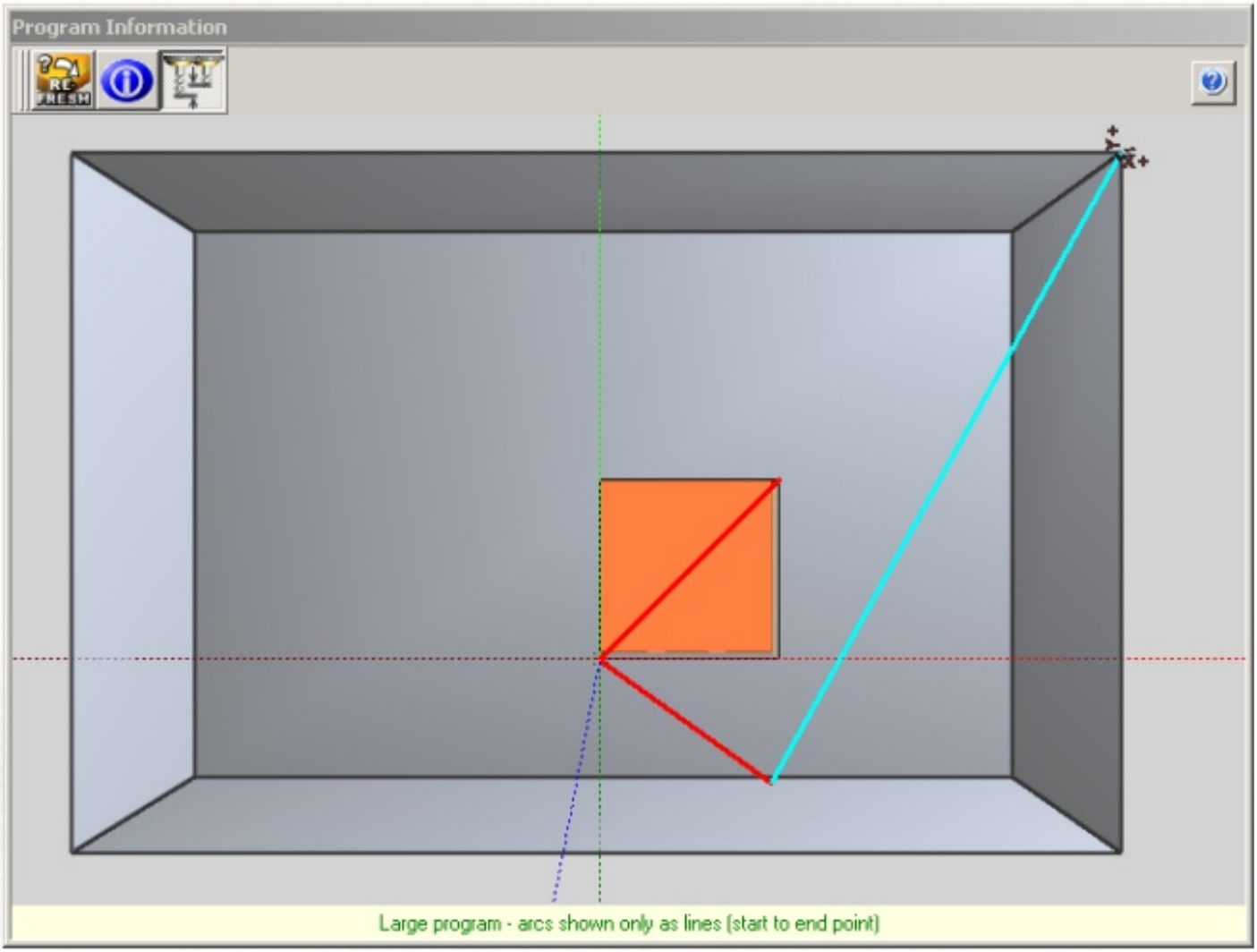
If the window below pops up, this means you have exceeded the machines limit of travel in your program



This has occurred because my Work Offset is set at the Machine Home position which makes the machine believe that my billet is outside of the working area as can be seen in the Program Information window below.



If your Work Offsets are correctly set then the Program Information window should look like the image below and the Machine movement error! window will not pop up.

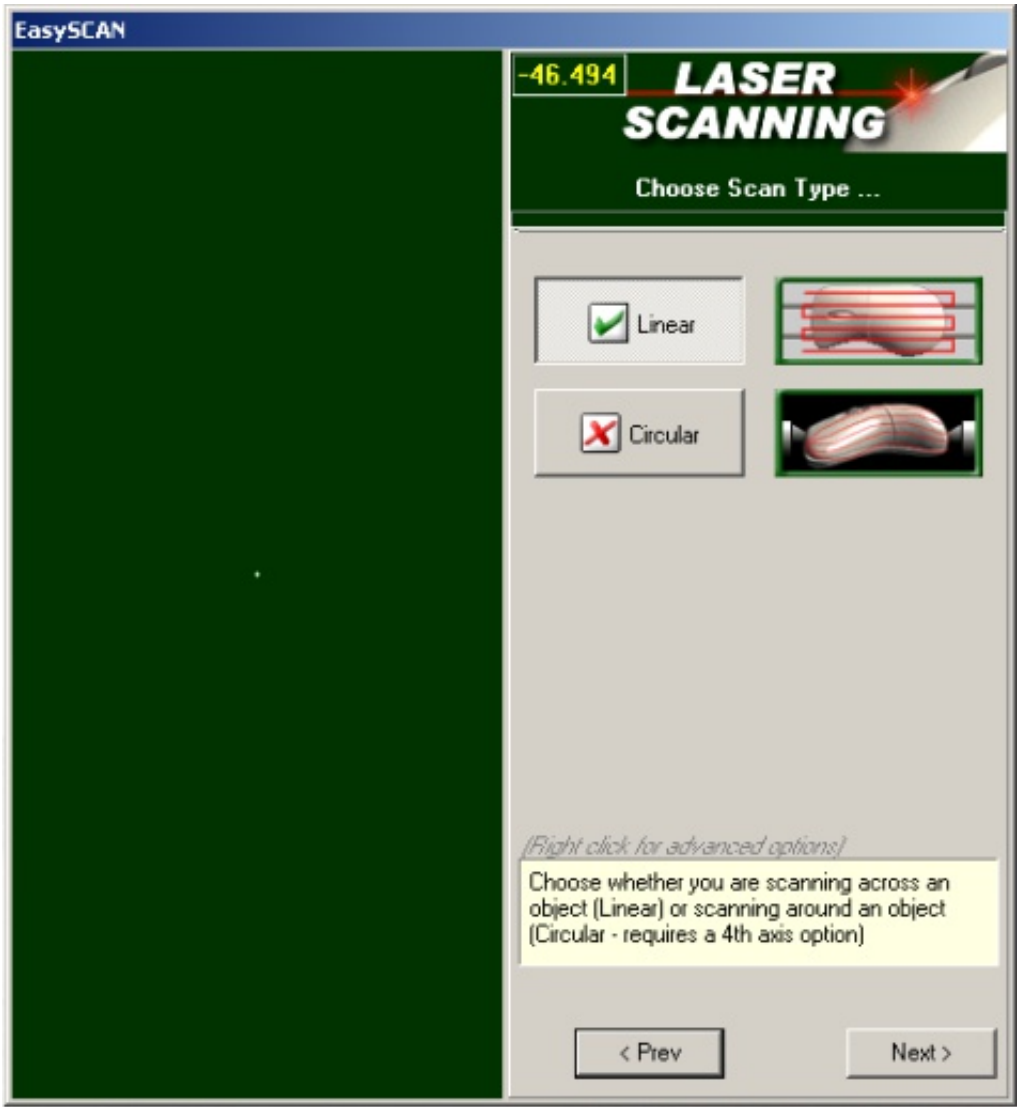


EasyScan 3D

The button to the right opens up the 3D laser scanning application.

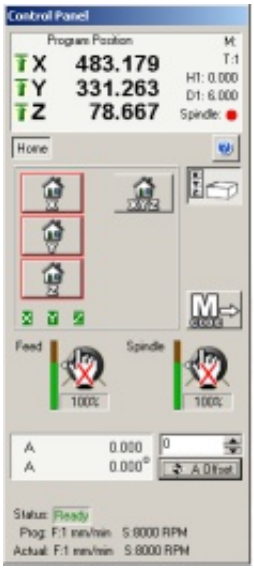
The 3D scanning application will only open if the 3D scanner has been fitted to your machine, if not fitted this option will be greyed out like the image below.

After the Easyscan button has been pressed the screen below will open.



To operate the laser scanning application see the Easyscan Tutorial.

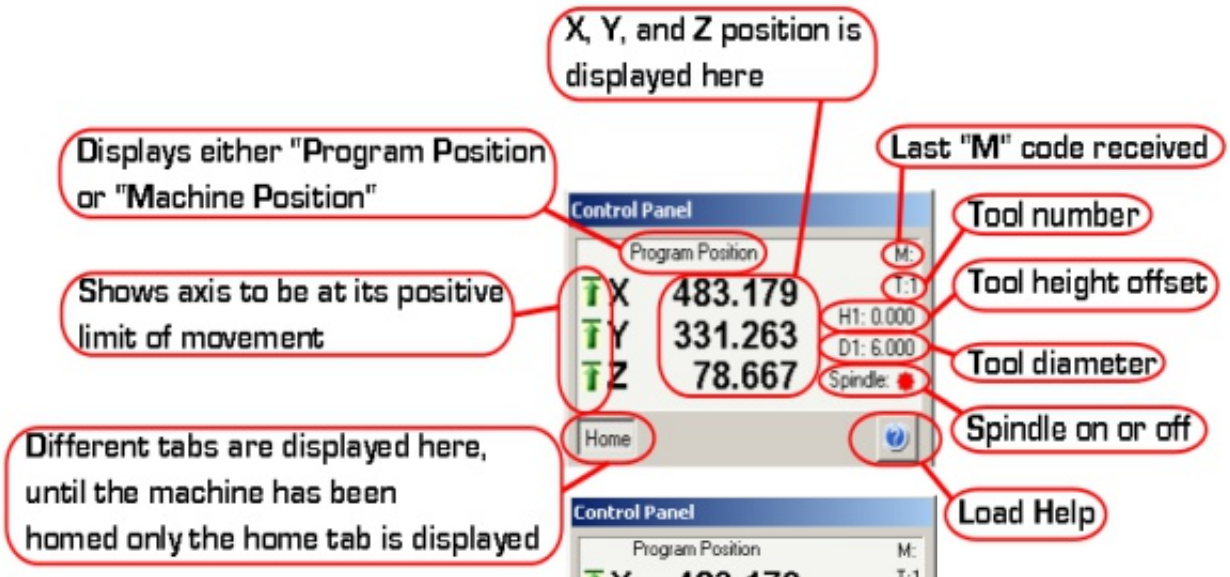
Control Panel



The Control Panel shown on the right is the interface window you will use to control the CNC machine.
This section will describe all features of the Control Panel and explain how to use it.

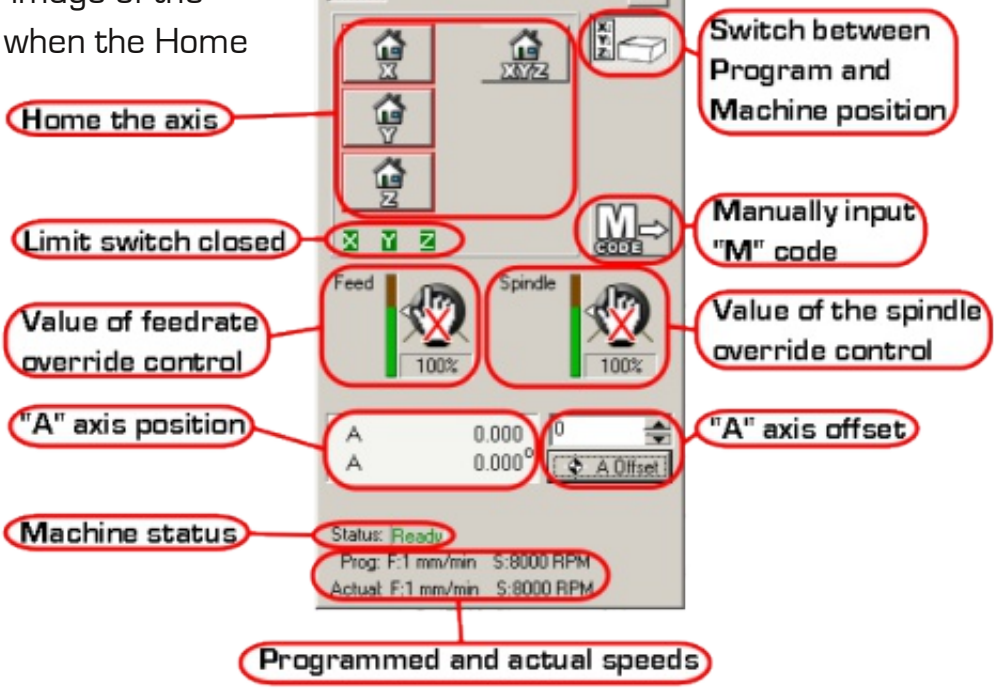
Information Window

The information window displays information on the status of the machine as shown below.



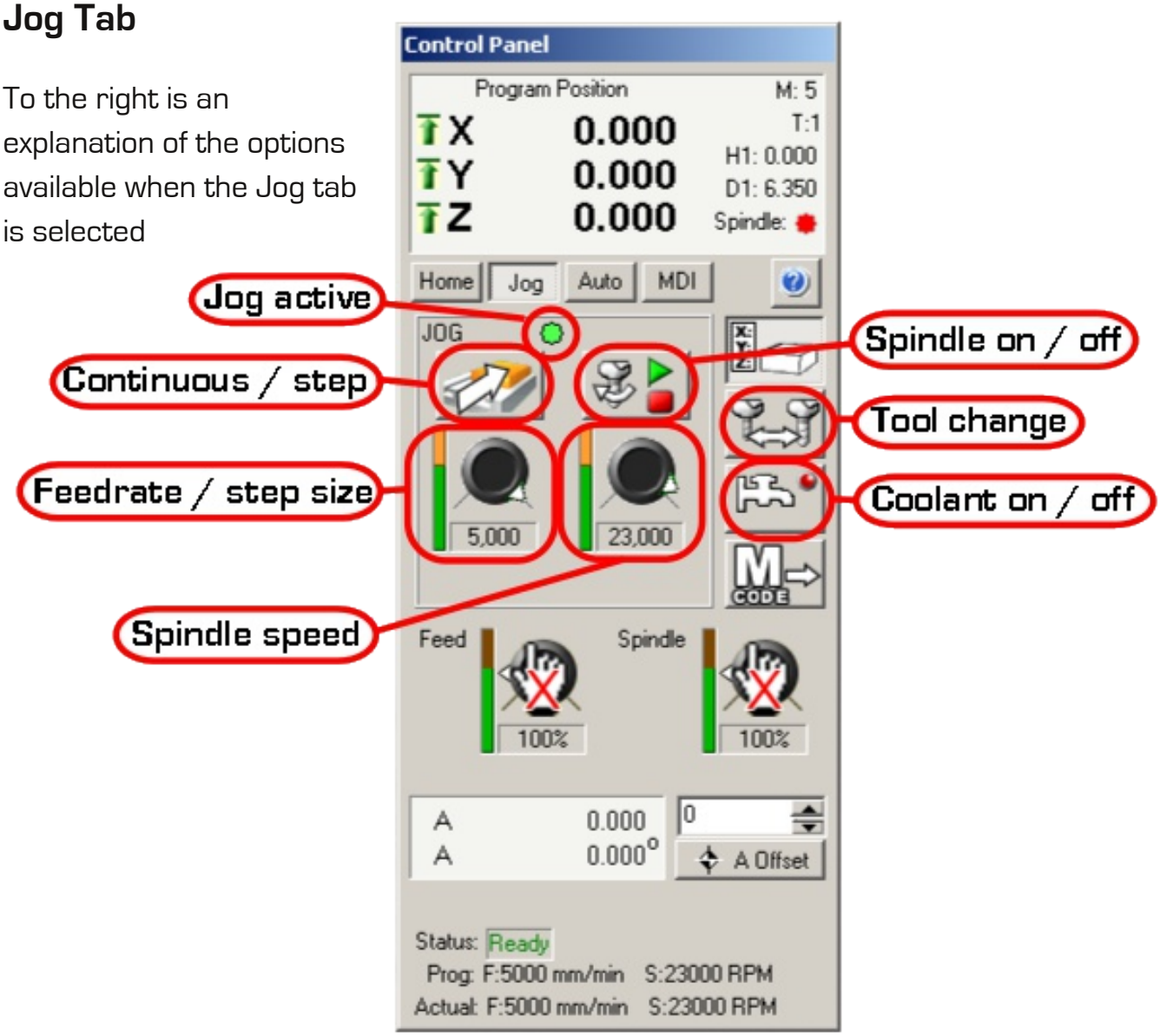
Home Tab

To the right is an image of the options available when the Home tab is selected

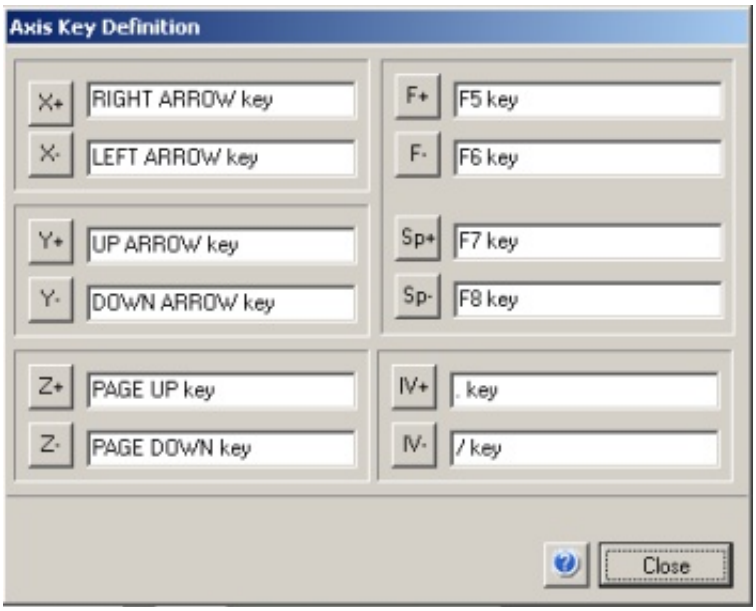


Jog Tab

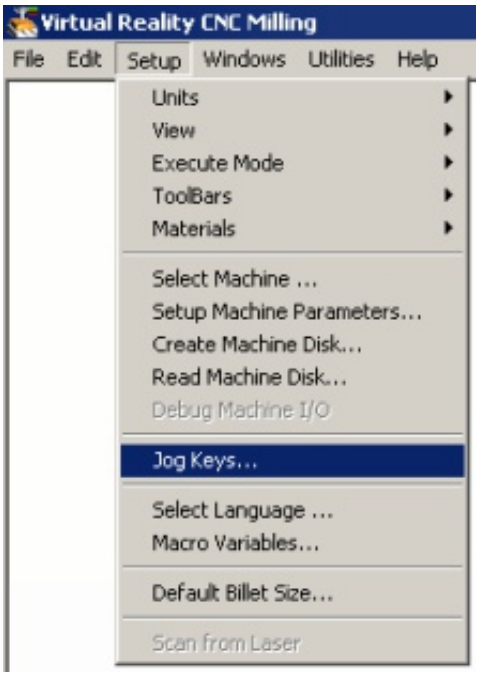
To the right is an explanation of the options available when the Jog tab is selected



The image below shows the keys used to jog the axis

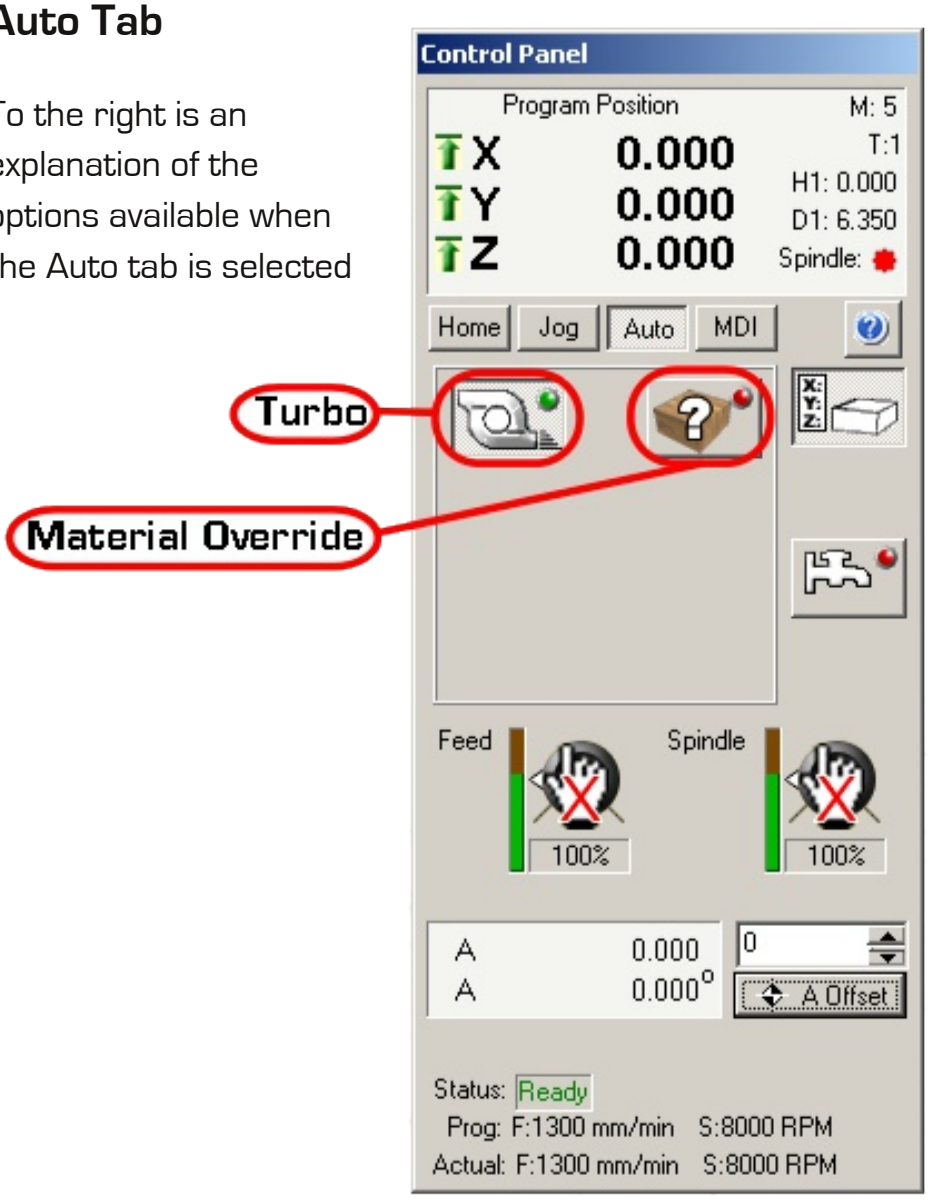


The menu on the left can be found by following the path shown below



Auto Tab

To the right is an explanation of the options available when the Auto tab is selected



Turbo Mode

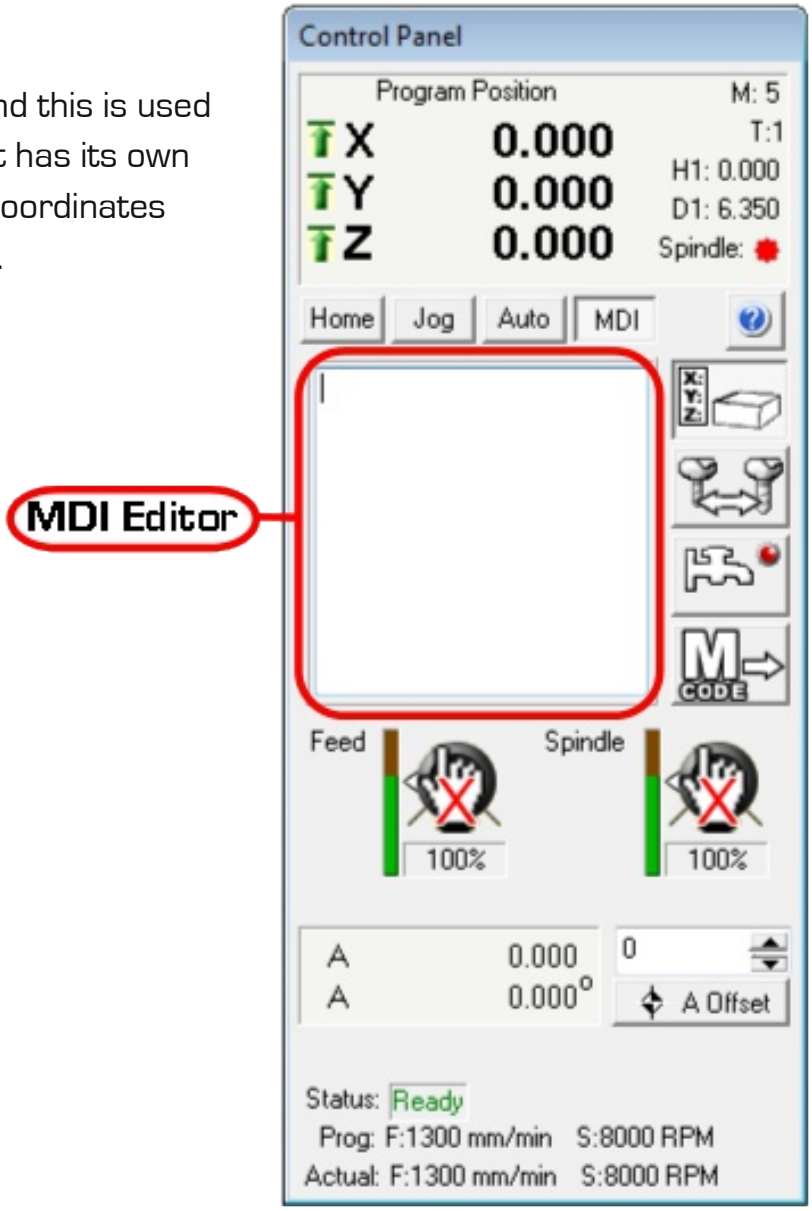
This will send a stream of information to the machine and will produce a smooth fluid movement of the tool resulting in a better surface finish. This is turned on by default.

Material Override Mode

This will open a drop down box enabling you to choose from materials in the material library, this will override the feedrate and spindle speed in your program. Use caution when using this option as it will not override the step down, only the feedrate and spindle speed will be changed.

MDI Tab

The MDI tab is for manual data input and this is used to move the tool to a specific location. It has its own text editor, as shown on the right and coordinates can be entered here to control the tool.



File Control Buttons

The buttons shown below are the file control buttons and they control how the machine treats the program.

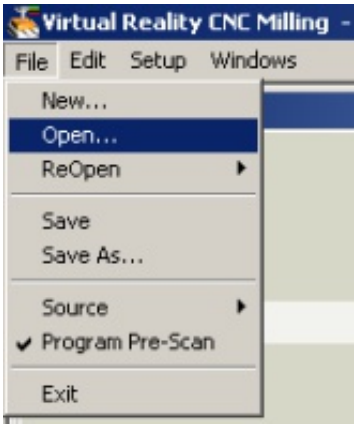
When in the Auto tab they control the Editor Window, and when in the MDI tab they control the MDI Editor.

It is good practise when running a program to rewind before playing as you would not want to start a program part way through.

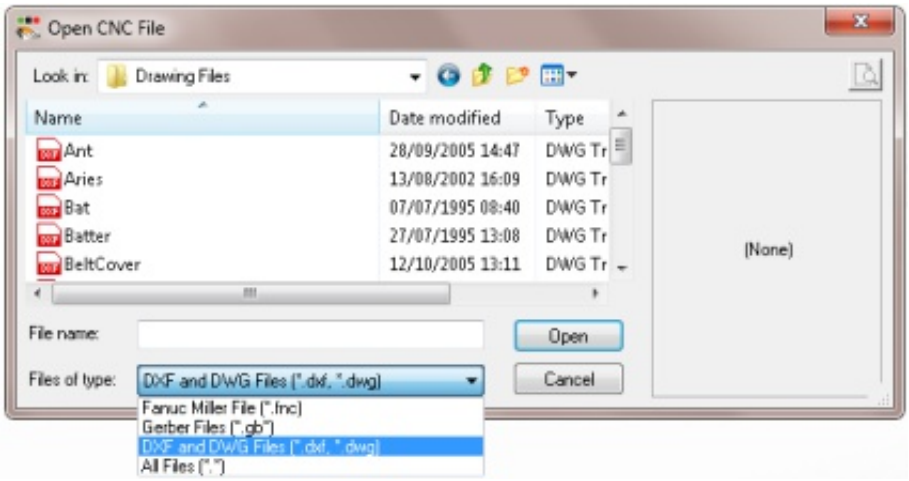
Opening a DXF

Many of our customers already have a 2D Vector drawing package that they are comfortable using and do not want to convert to a new piece of software.
In light of this we have added a DXF import function which takes your drawing and uses CAM to create a toolpath program that VR Milling can understand.

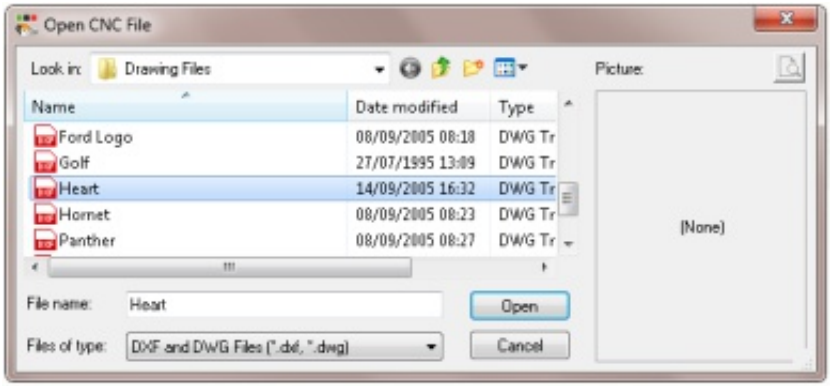
Select "File", then "Open" from the toolbar

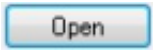


Under the files of type drop down, select "DXF and DWG files"

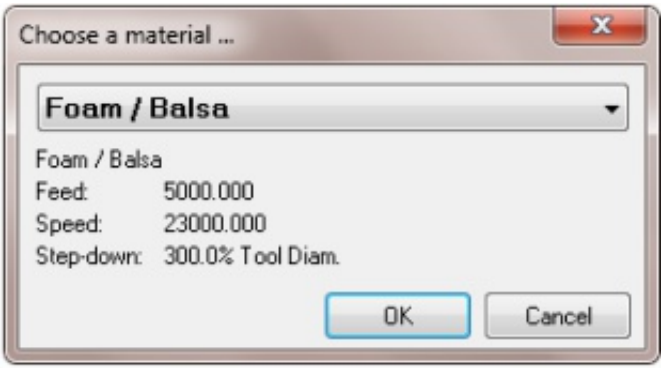


When VR Milling was installed a folder of DXF's was installed.
They can be found in "\My Documents\Denford\Drawing Files\"

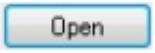


Select a file and "Left click" the Open button 

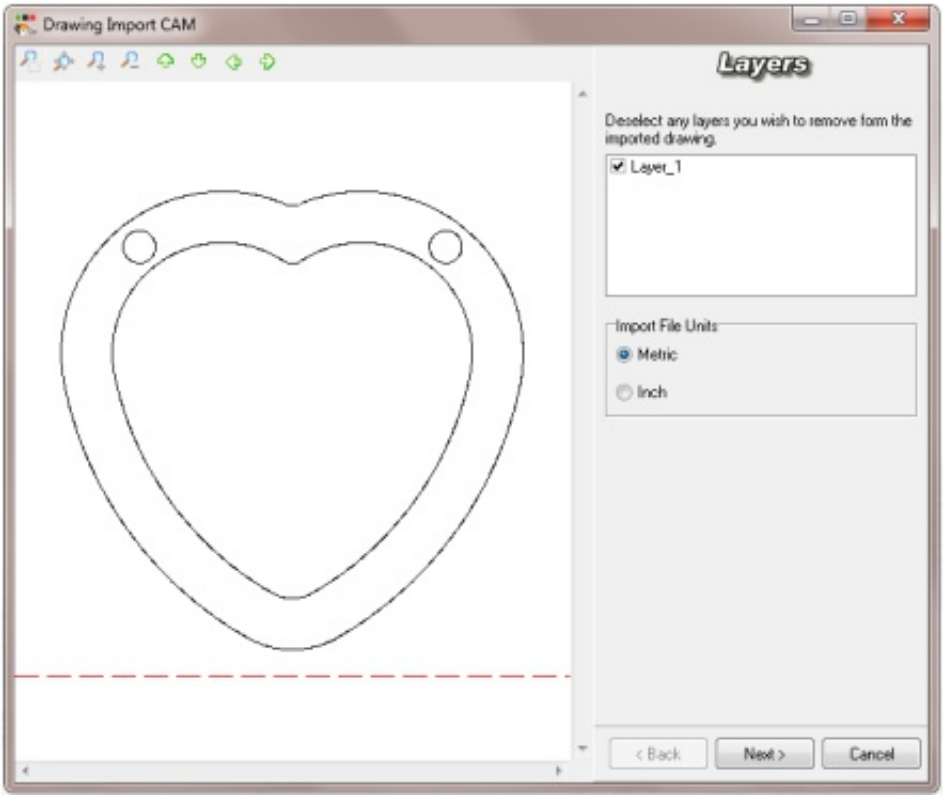
A window will appear, this window is to choose from one of the materials in your material library. This will set the feedrate, spindle speed, and step down for your program.



Select a material from the drop down.

"Left click" the Open button 

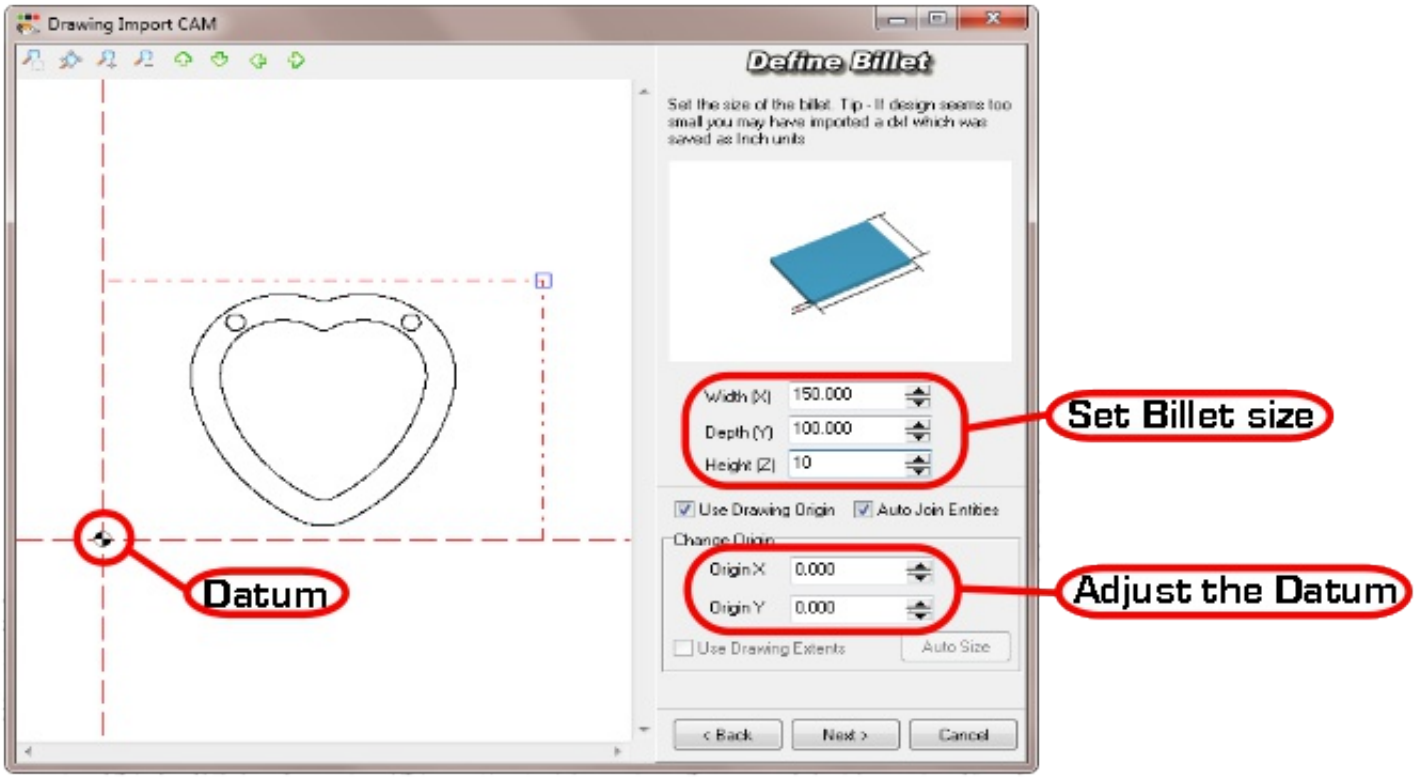
The Drawing Import CAM window will open and this wizard based program will guide you through the creation of a CNC program from your drawing.
The first screen as shown below is for layers.



On this screen you can deselect any layers you wish to remove and select the units of the drawing you have imported.

"Left click" the Next button

The window shown below will appear, this is where you define the billet and adjust the origin position of your drawing.



Set your billet to the correct size

The red dashed line represents the perimeter of your billet

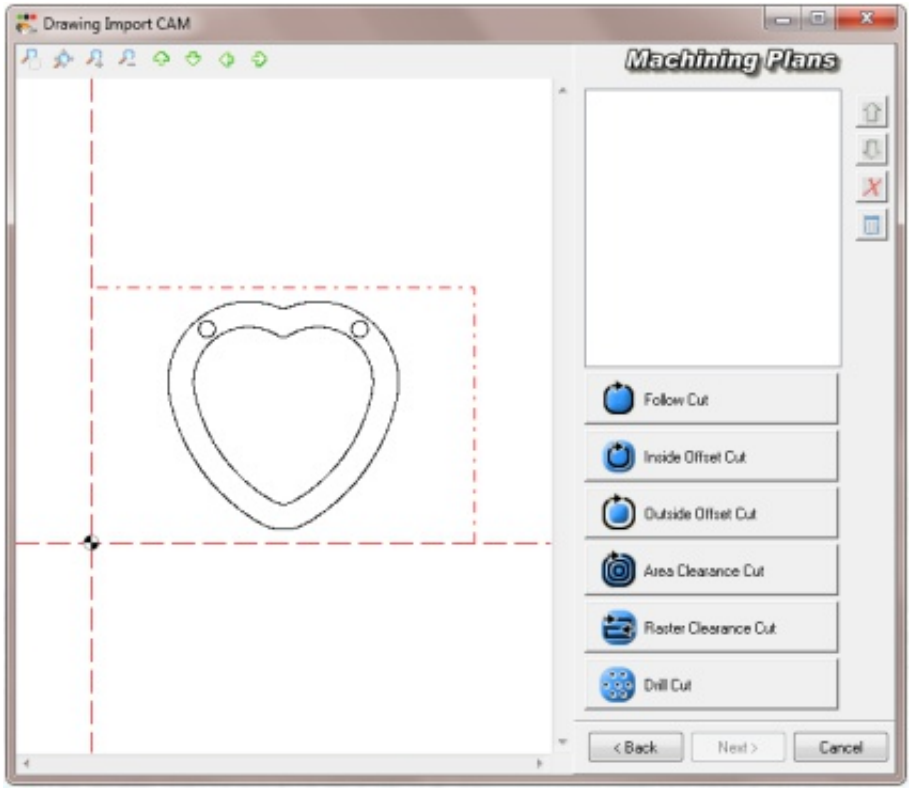
Move the origin if necessary

"Left click" the Next button

The window to the right will pop up.

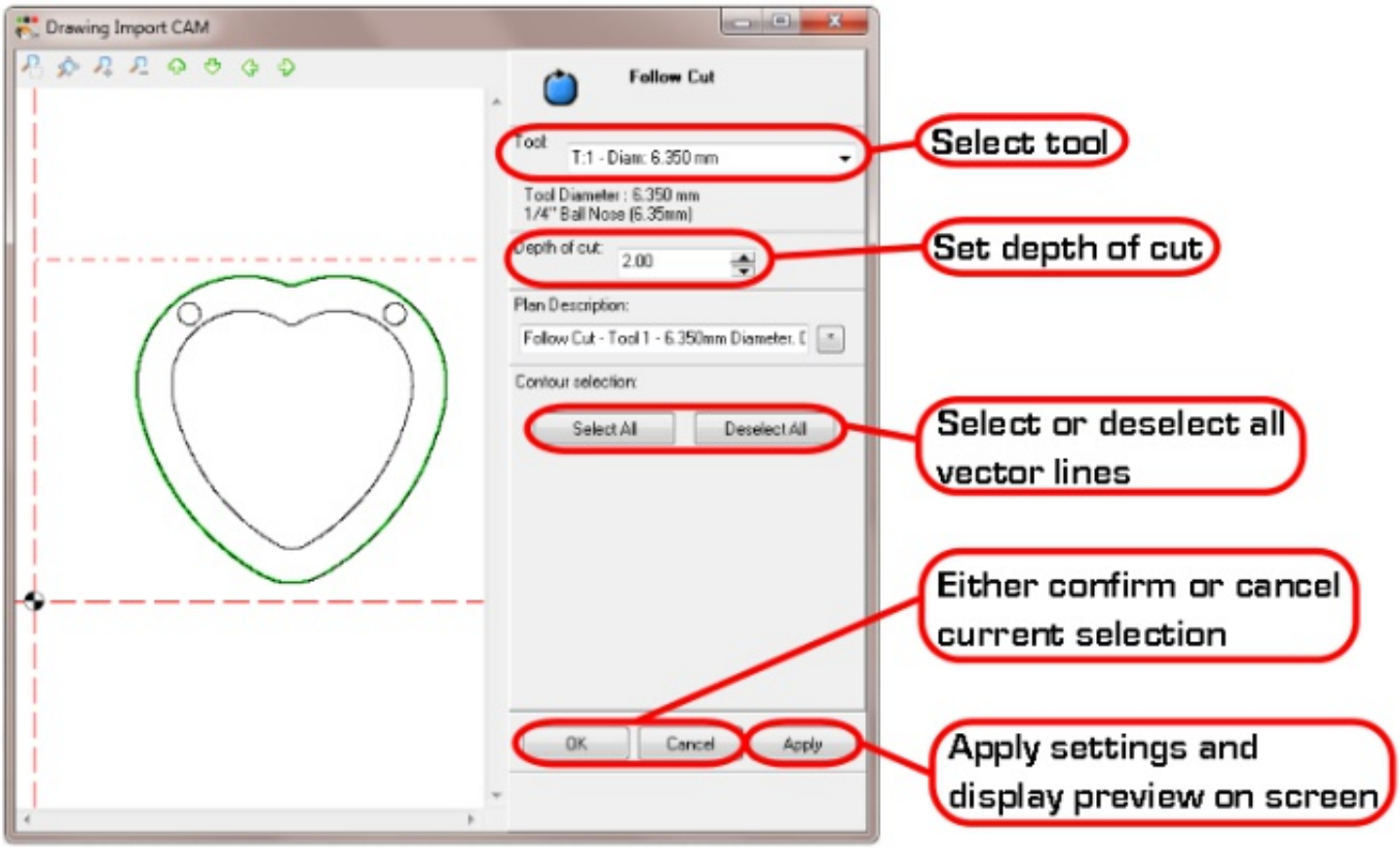
This is where you instruct the CAM software which machining plans to assign to your drawing.

The next few pages will describe each machining plan and how to use it



Follow Cut

"Left click" the Follow Cut button



Select the tool

Set the depth

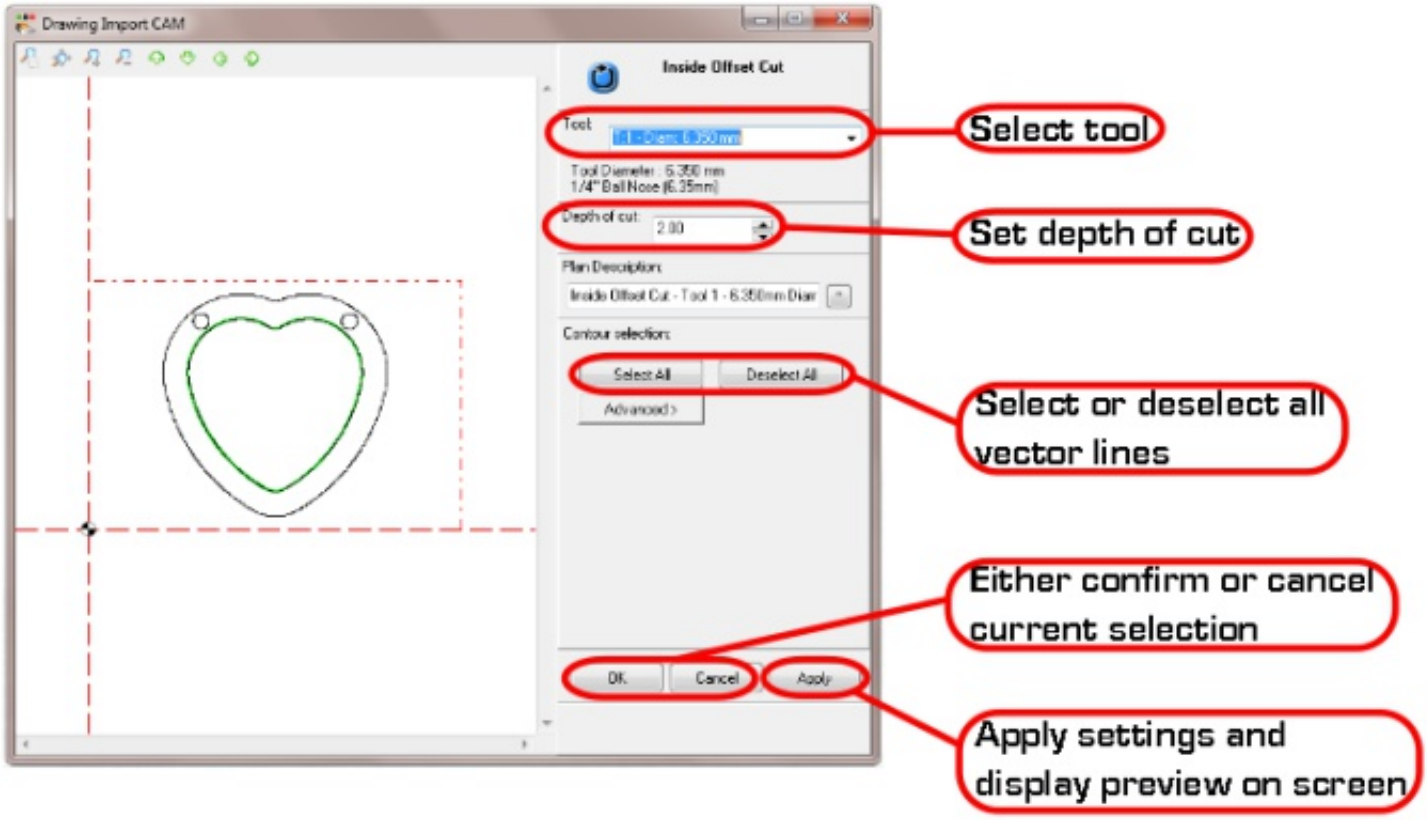
Select which part of your drawing to apply the cut to

"Left click" the Apply button to preview

"Left click" the OK button to save

Inside Offset Cut

"Left click" the Inside Offset Cut button



Select the tool

Set the depth

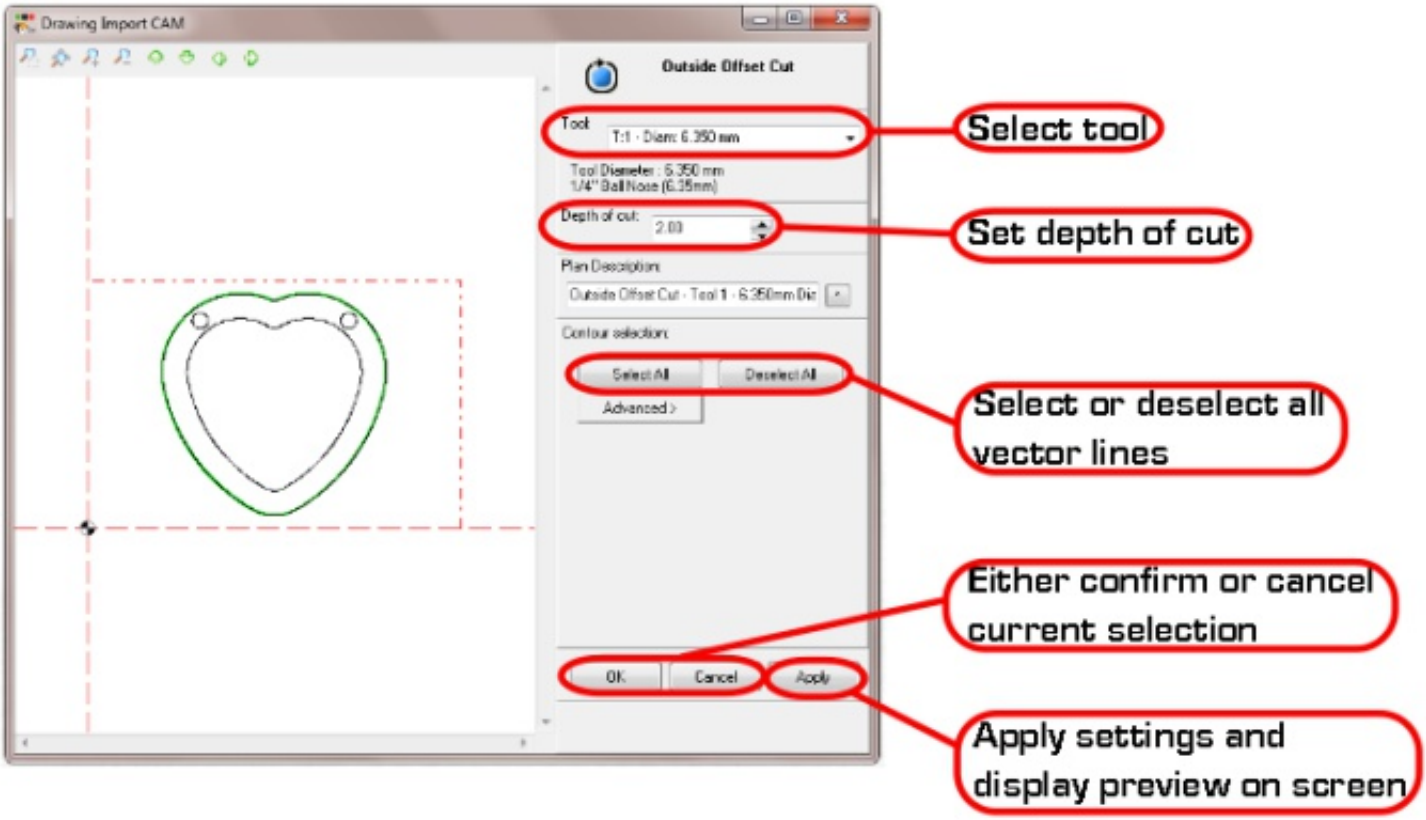
Select which part of your drawing to apply the cut to

"Left click" the Apply button to preview

"Left click" the OK button to save

Outside Offset Cut

"Left click" the Outside Offset Cut button



Select the tool

Set the depth

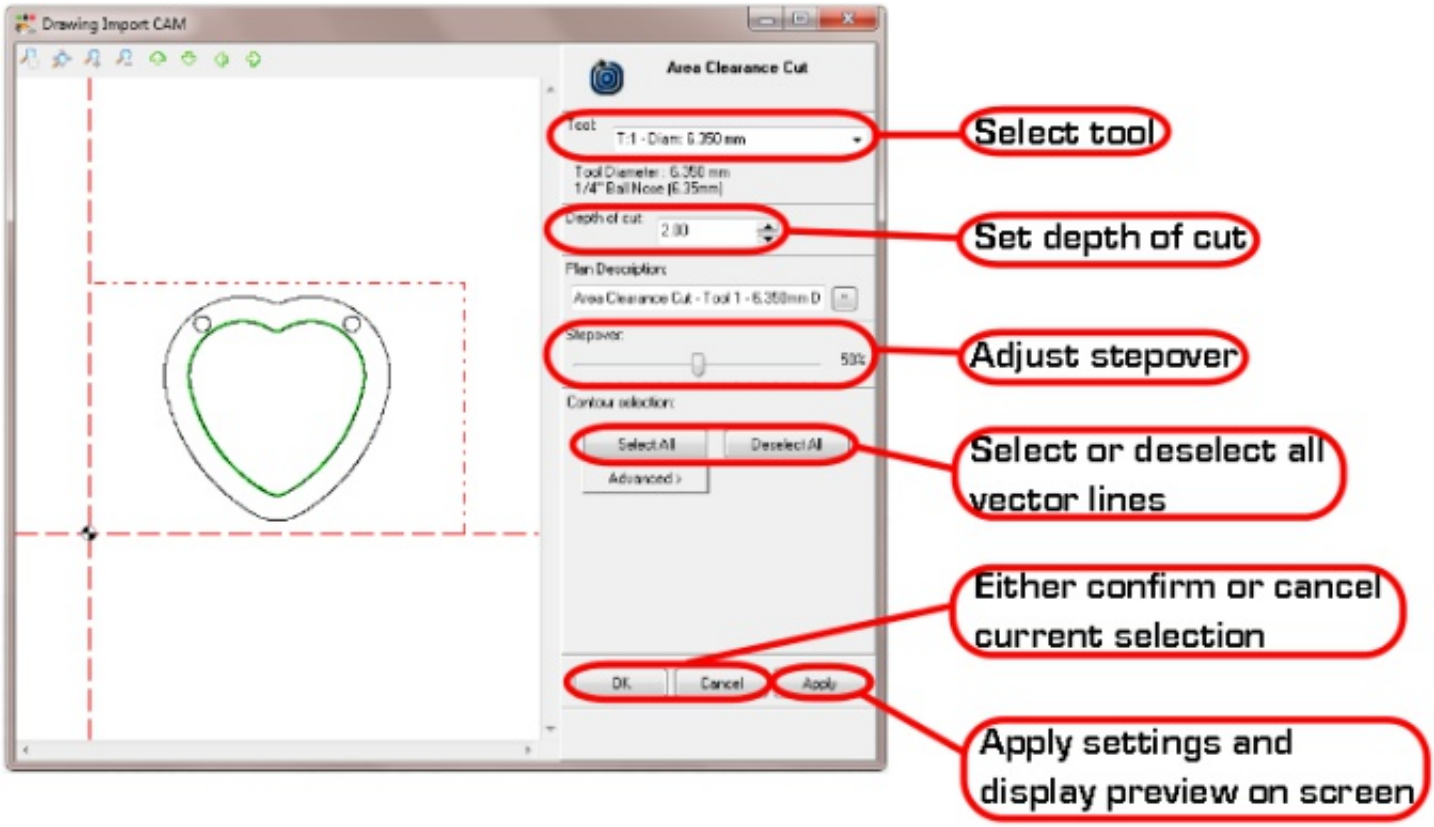
Select which part of your drawing to apply the cut to

"Left click" the Apply button to preview

"Left click" the OK button to save

Area Clearance Cut

"Left click" the Area Clearance Cut button



Select the tool

Set the depth

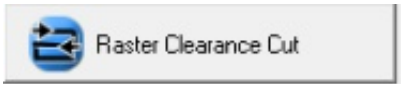
Set the step over

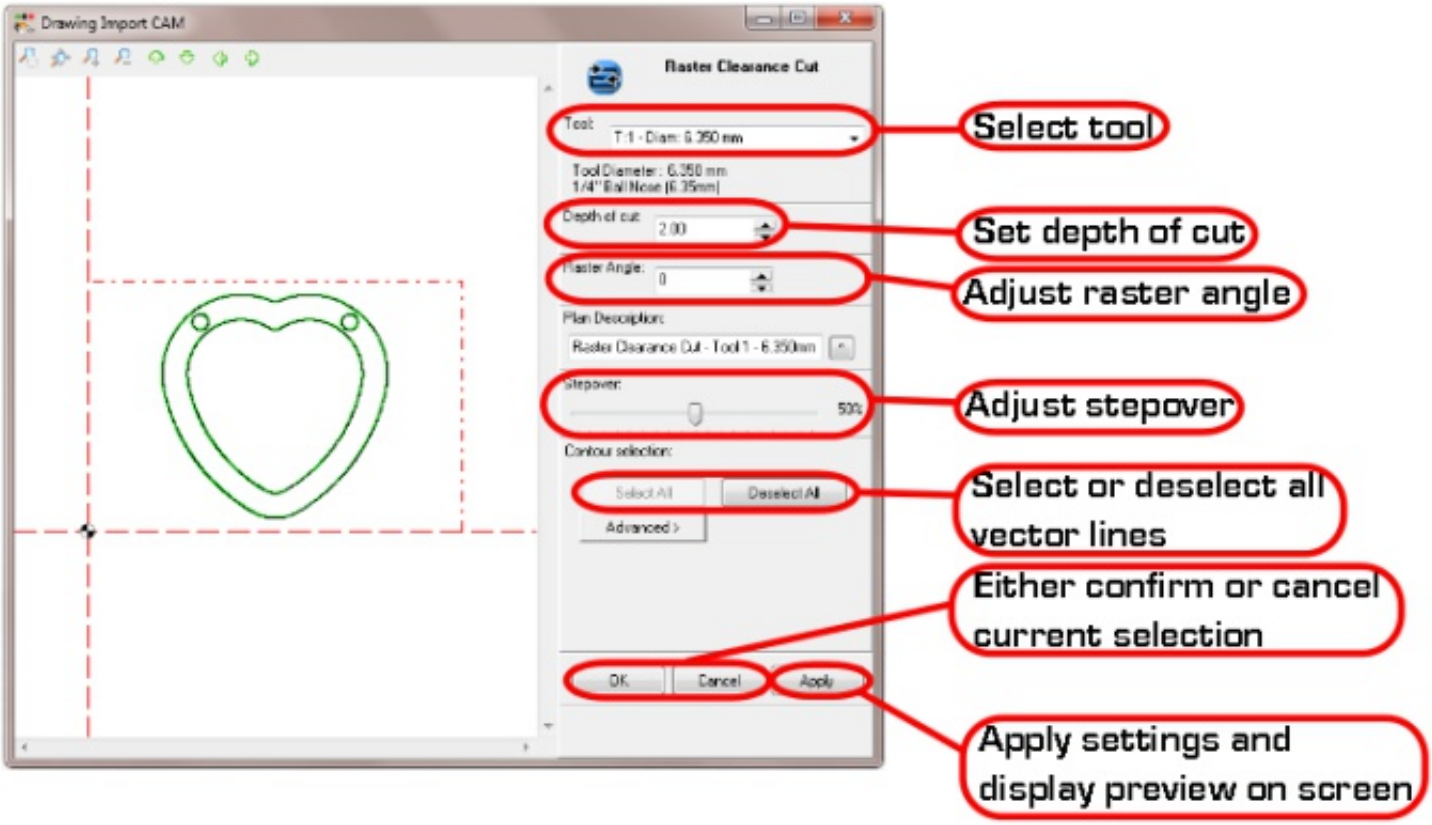
Select which part of your drawing to apply the cut to

"Left click" the Apply button to preview

"Left click" the OK button to save

Raster Clearance Cut

"Left click" the Raster Clearance Cut button 



Select the tool

Set the depth

Set the raster angle

Set the step over

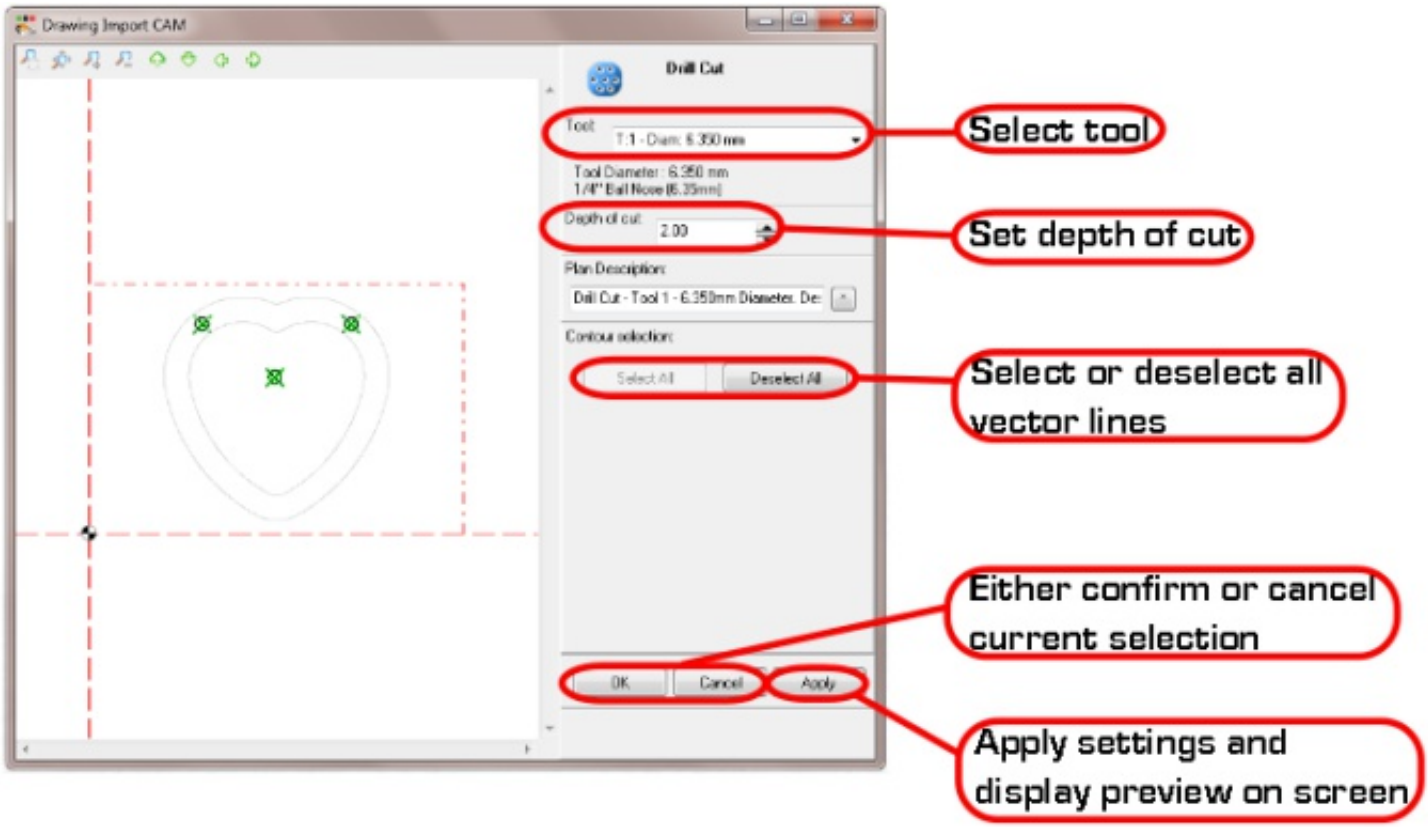
Select which part of your drawing to apply the cut to

"Left click" the Apply button to preview

"Left click" the OK button to save

Drill Cut

"Left click" the Drill Cut button



Select the tool

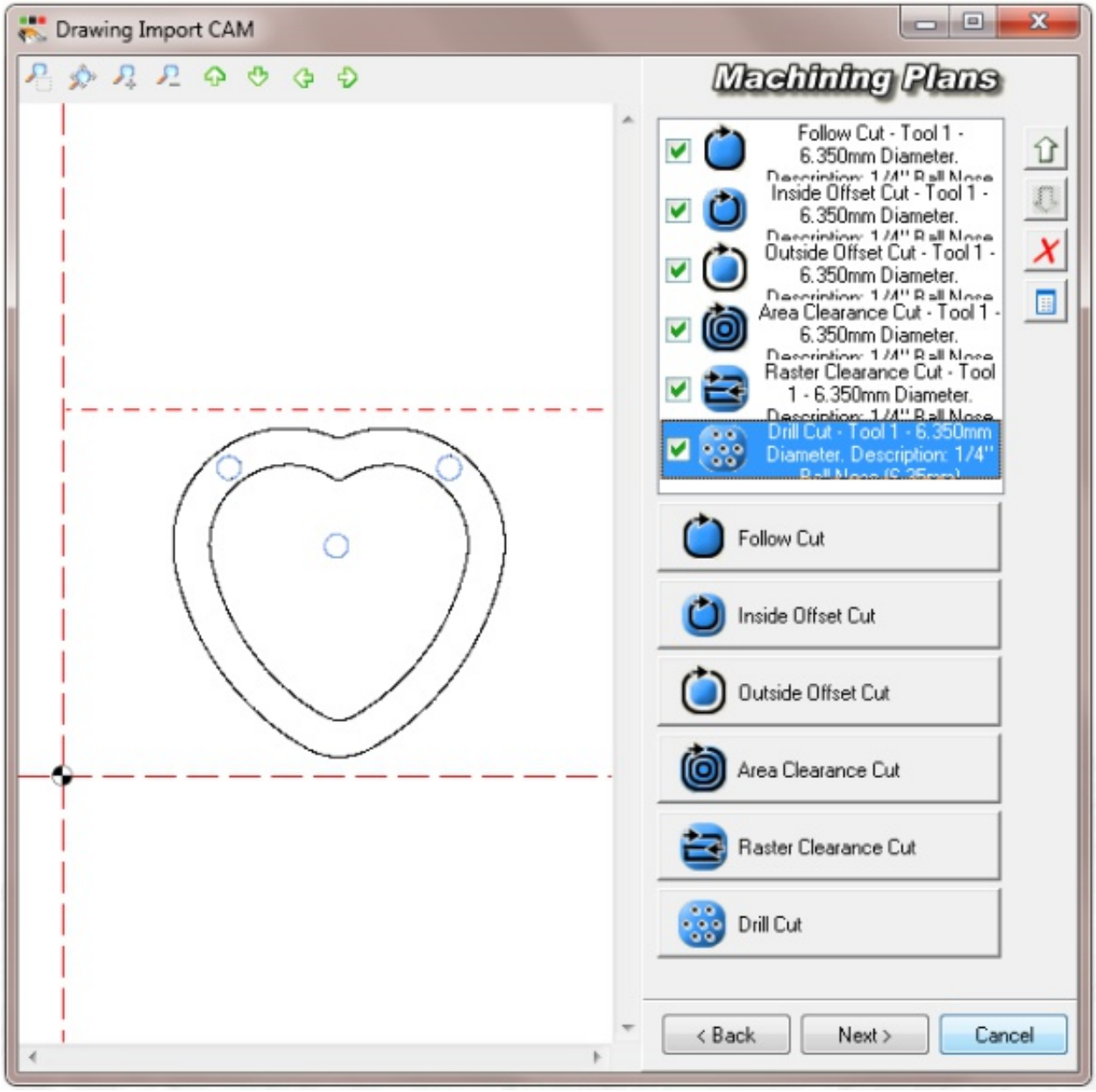
Set the depth

Select which part of your drawing to drill (only the centre of circles can be selected)

"Left click" the Apply button to preview

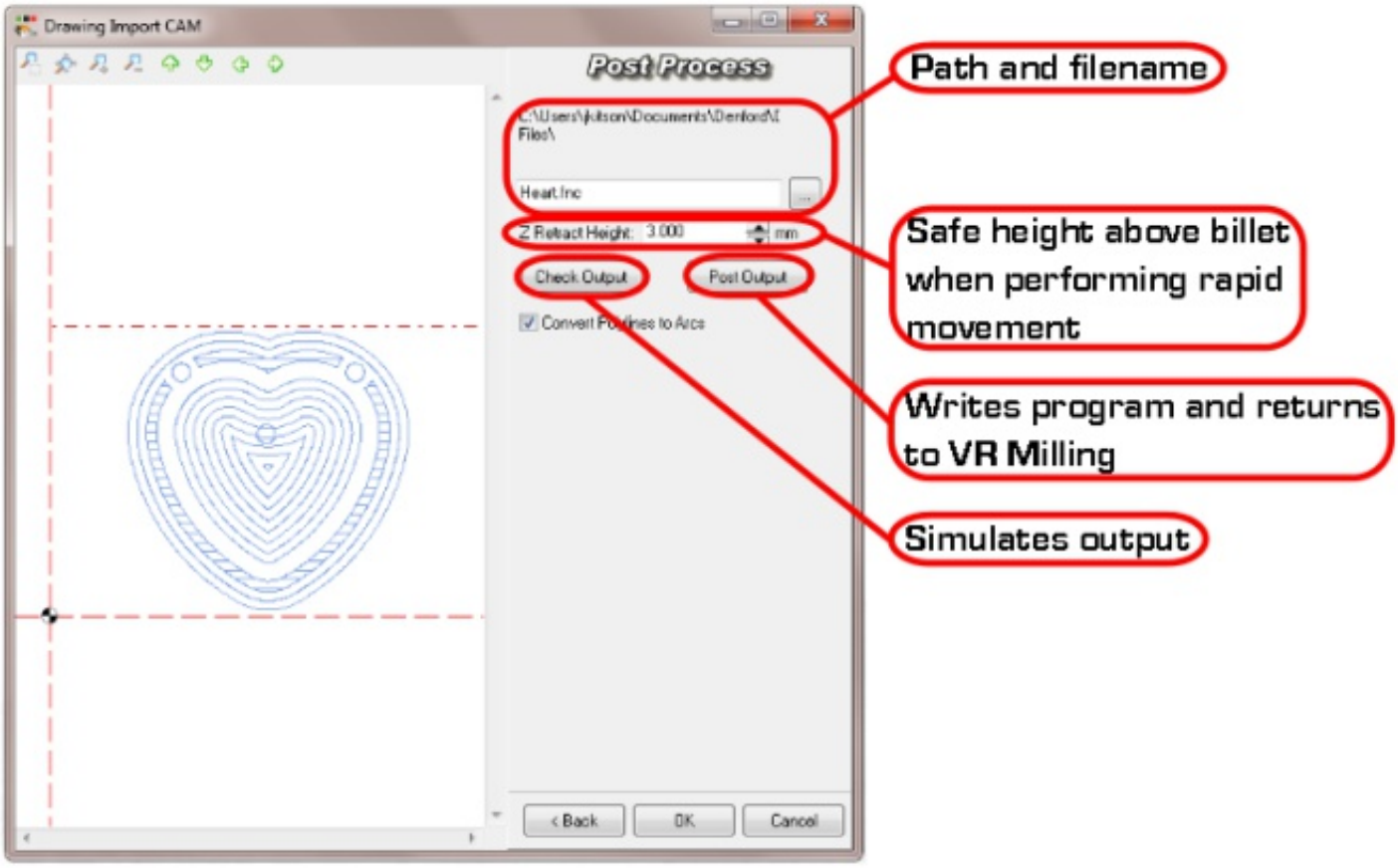
"Left click" the OK button to save

When all machining plans for your drawing have been created you are ready for the next step.



"Left click" the Next button

The screen below is the Post Process screen where the final options are set before the program is written



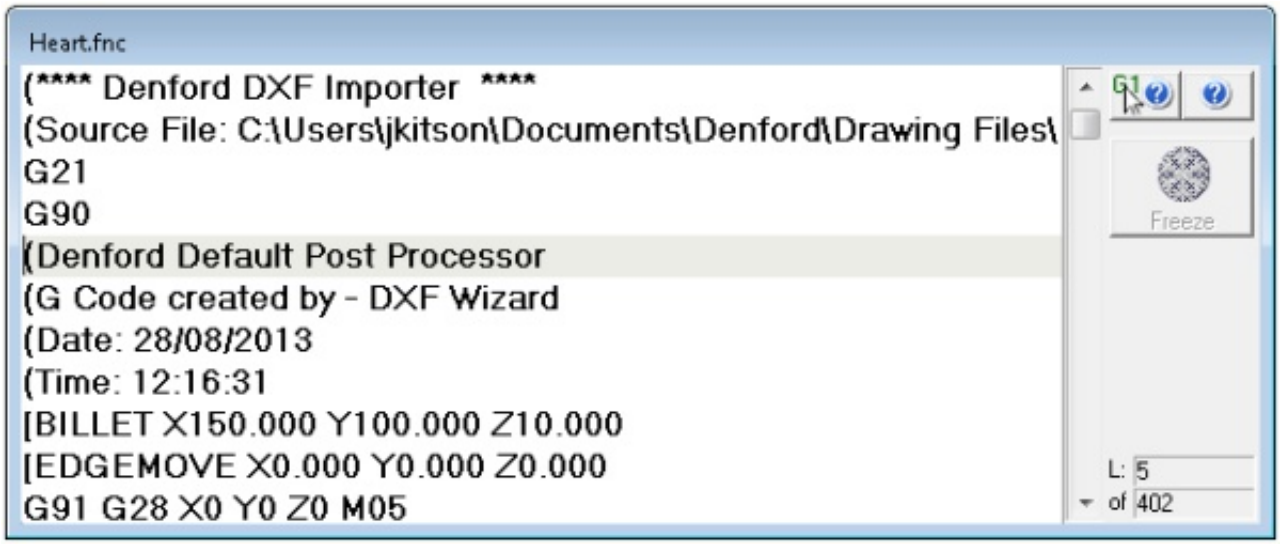
Give the file a name

Set a safe height (to avoid hitting any parts of the fixture which may be above the billet)

Check the Output

Post Output

The image below shows the finished program in the Editor window in VR Milling



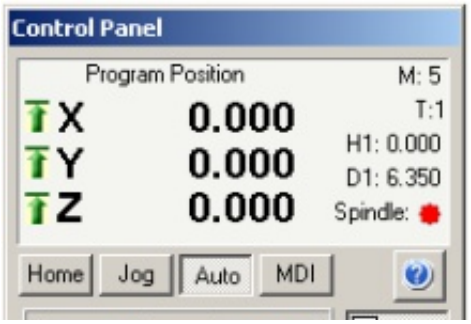
Running a Program

Having created your program you will now want to machine it. Follow the instructions below

Place a billet in the machine and secure it

Set the Offsets, using the Tool and Offset Editor

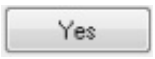
Ensure the Control Panel is in Auto Mode



"Left click" Stop, Rewind, then Play

The warning shown below will appear.

"Left click" the Yes button



The machine will now run your program

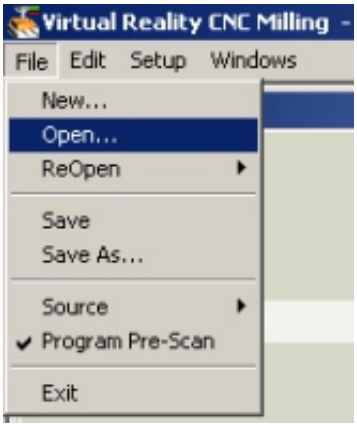
You may want to keep your hand over the Feedrate Override potentiometer on the front of the machine to adjust the feedrate if necessary.

Opening a Gerber

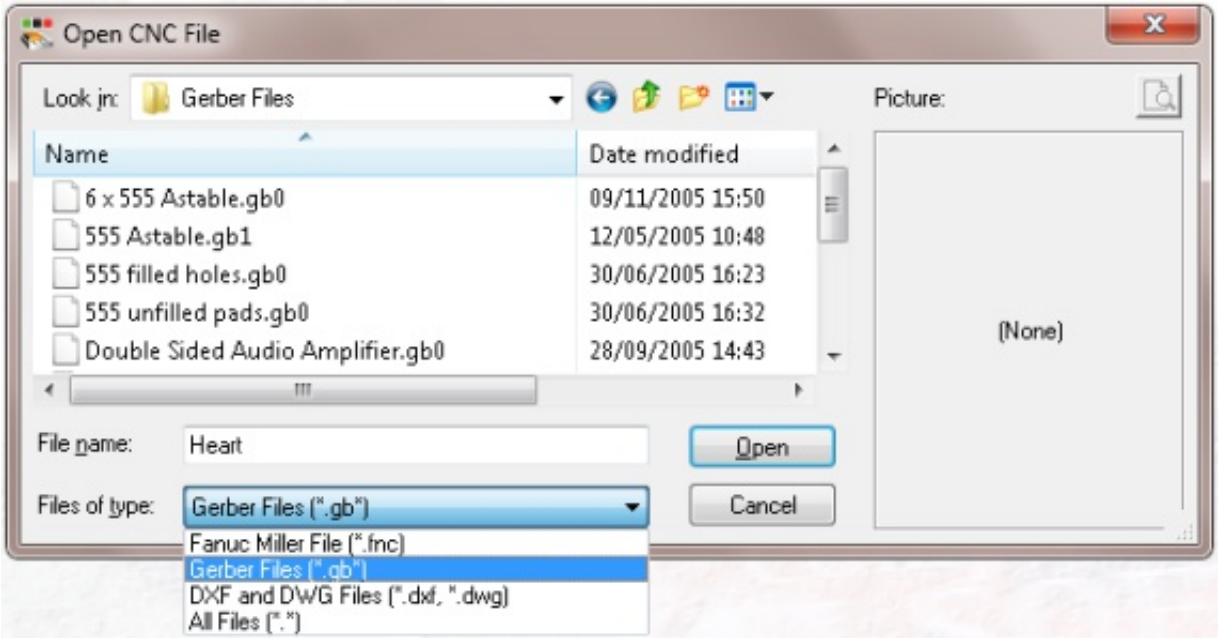
Denford Routers and Mills can be used to import Gerber files for machining circuit boards. Ideally you will require the Floating Head attachment which is Product Code FH1/0100P, but smaller boards can be machined without the floating head.

You can perform tool changes and fit different tools for cutting tracks in the PCB and drilling holes of different diameters.
For the purpose of this tutorial we will use the same tool and holes will be spot drilled only.
The tool used in this example is the PCB Engraving Cutter which is Product Code BI00811Z

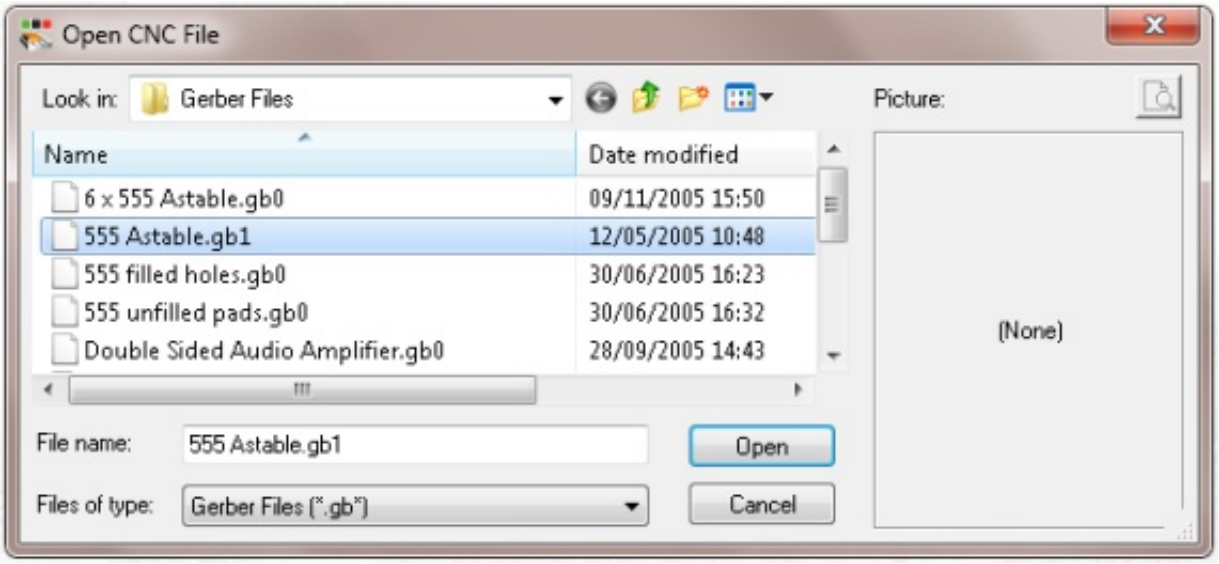
Select "File", then "Open" from the toolbar

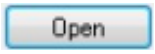


Under the files of type drop down, select "Gerber files"

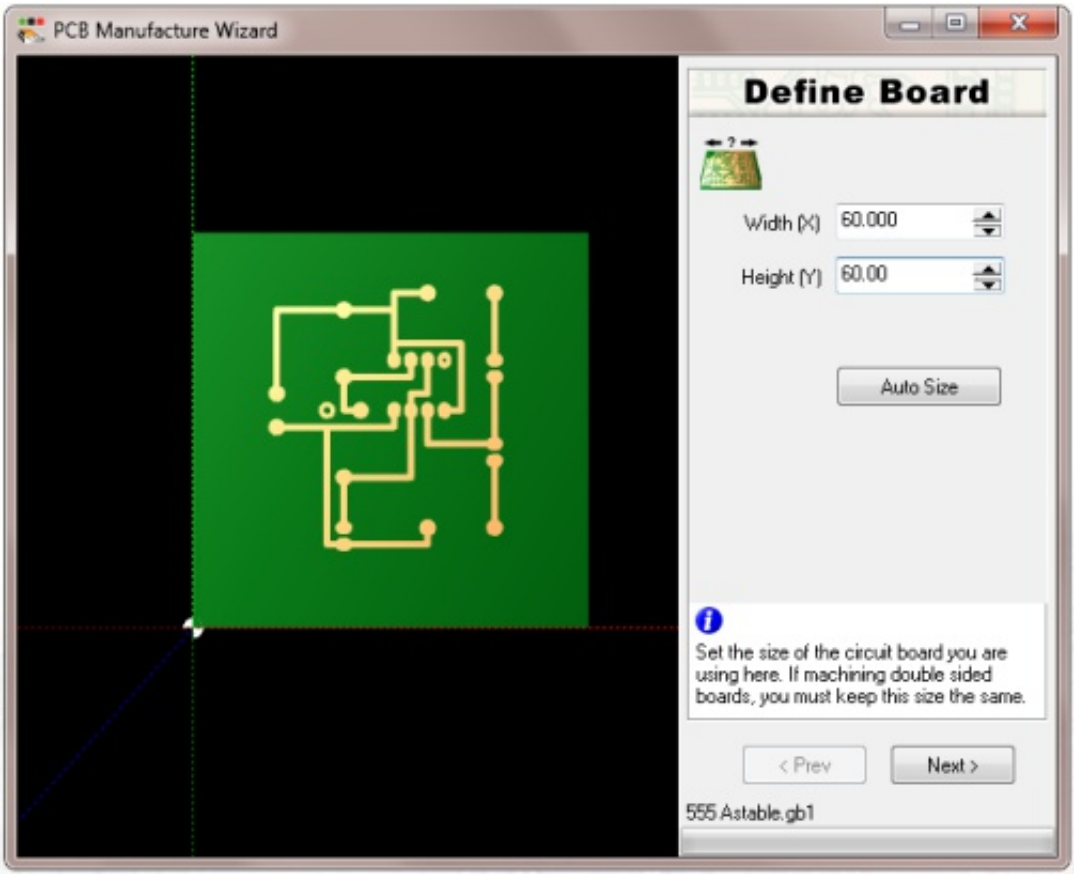


When VR Milling was installed a folder of Gerber files was installed.
They can be found in "\My Documents\Denford\Gerber Files\"



Select a file and "Left click" the Open button 

The window shown below will appear and ask you to define the board



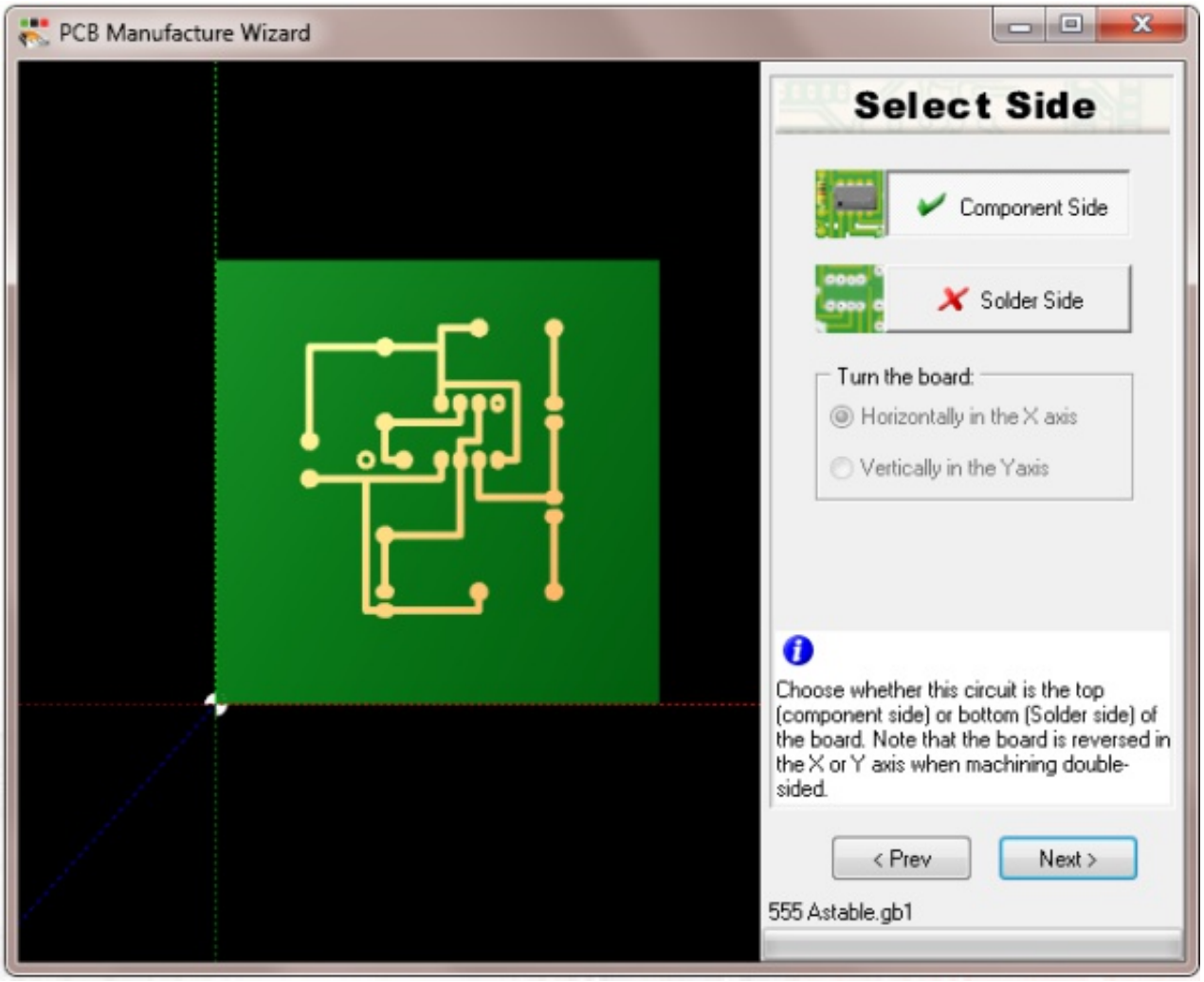
Enter the Width of the board

Enter the height of the board

The datum point is fixed at the bottom left corner

"Left click" the Next button

The window shown below will appear



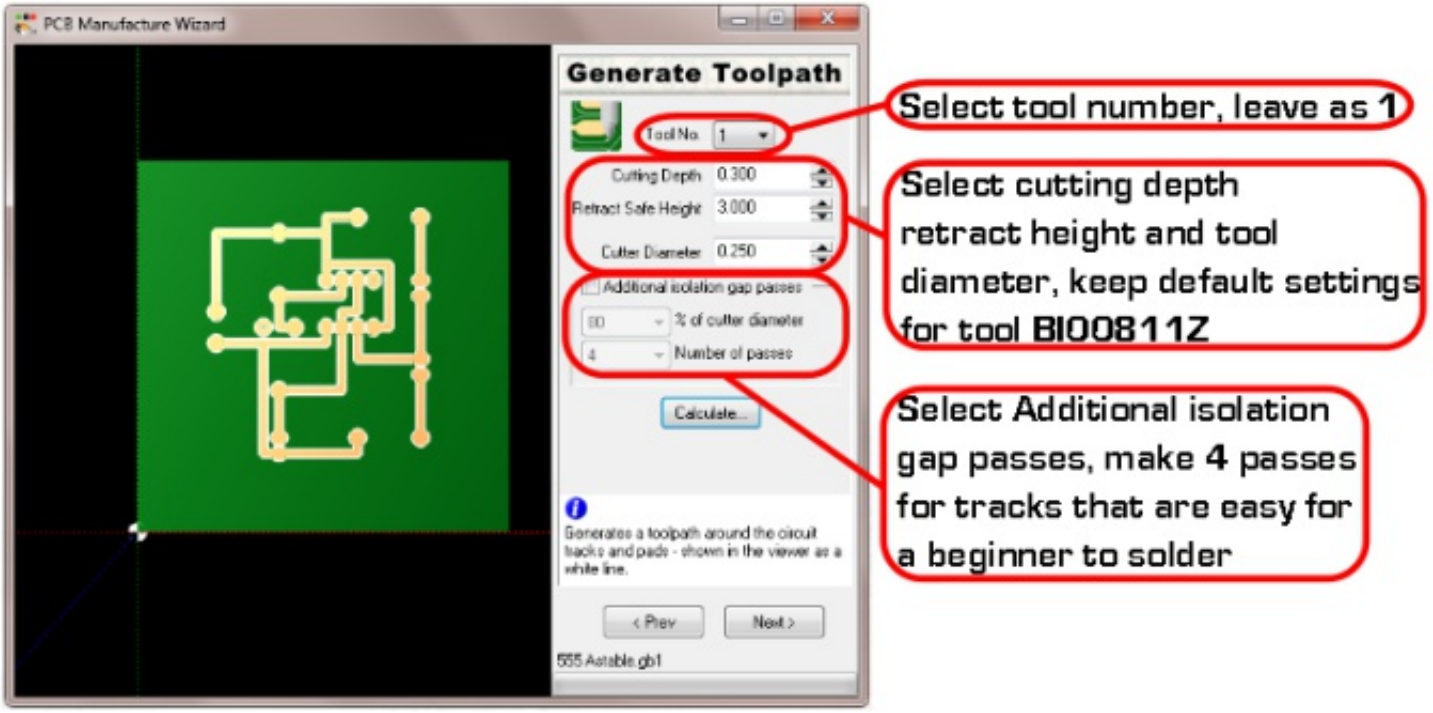
Select which side of the board this is, Component or Solder side

If solder side is selected

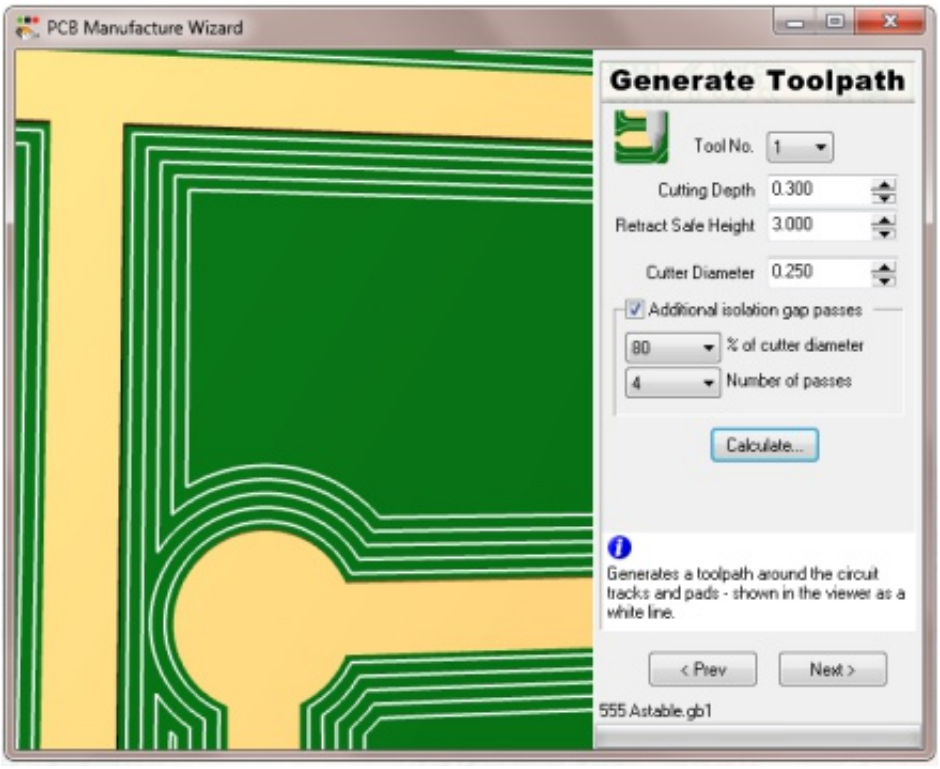
Select how the board is turned

"Left click" the Next button

The window shown below is where you specify the tool to be used and the number of passes the tool will make.
A single pass will be a very narrow track and will make it difficult for someone inexperienced at soldering to solder without bridging the tracks.
4 Additional isolation gap passes makes soldering the board much easier



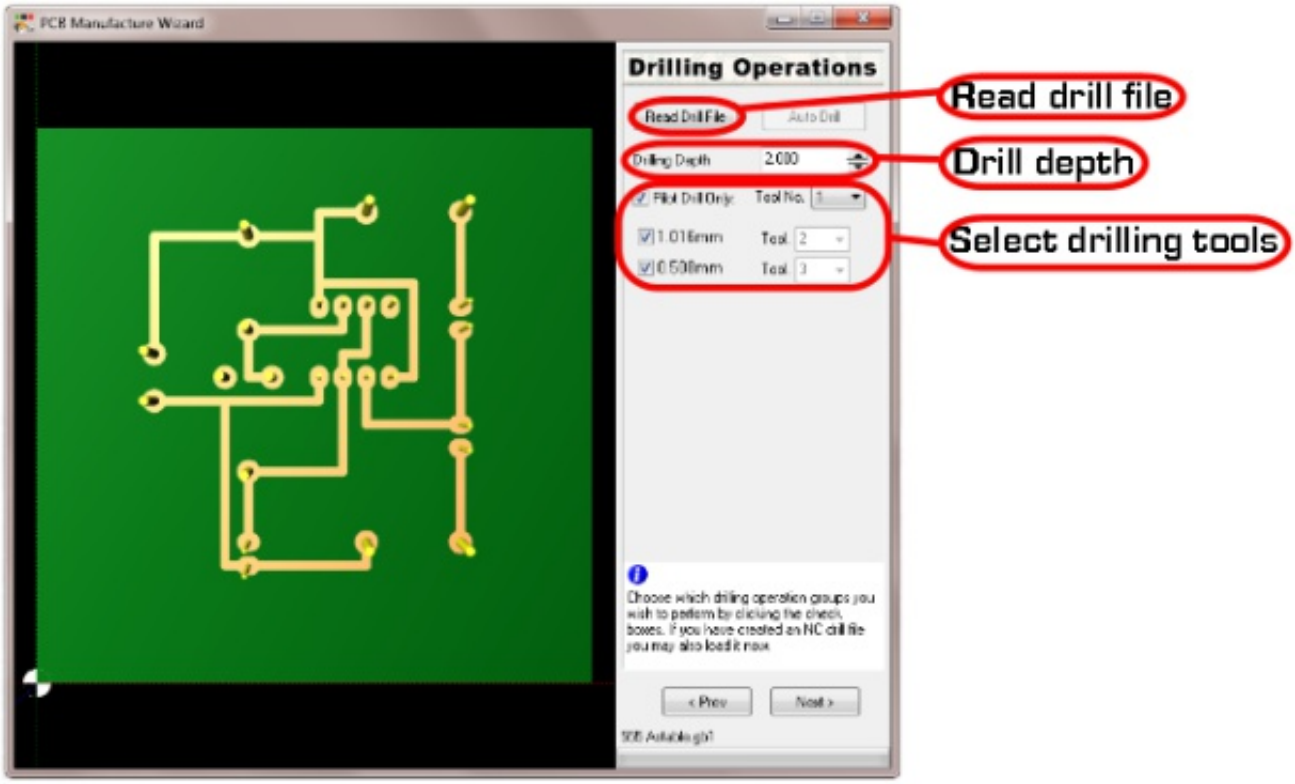
"Left click" the Calculate button



The image on the left shows the 4 additional isolation gap passes around the circuit.
The step over is set at 80% of the cutter diameter, this should not need changing.

"Left click" the Next button

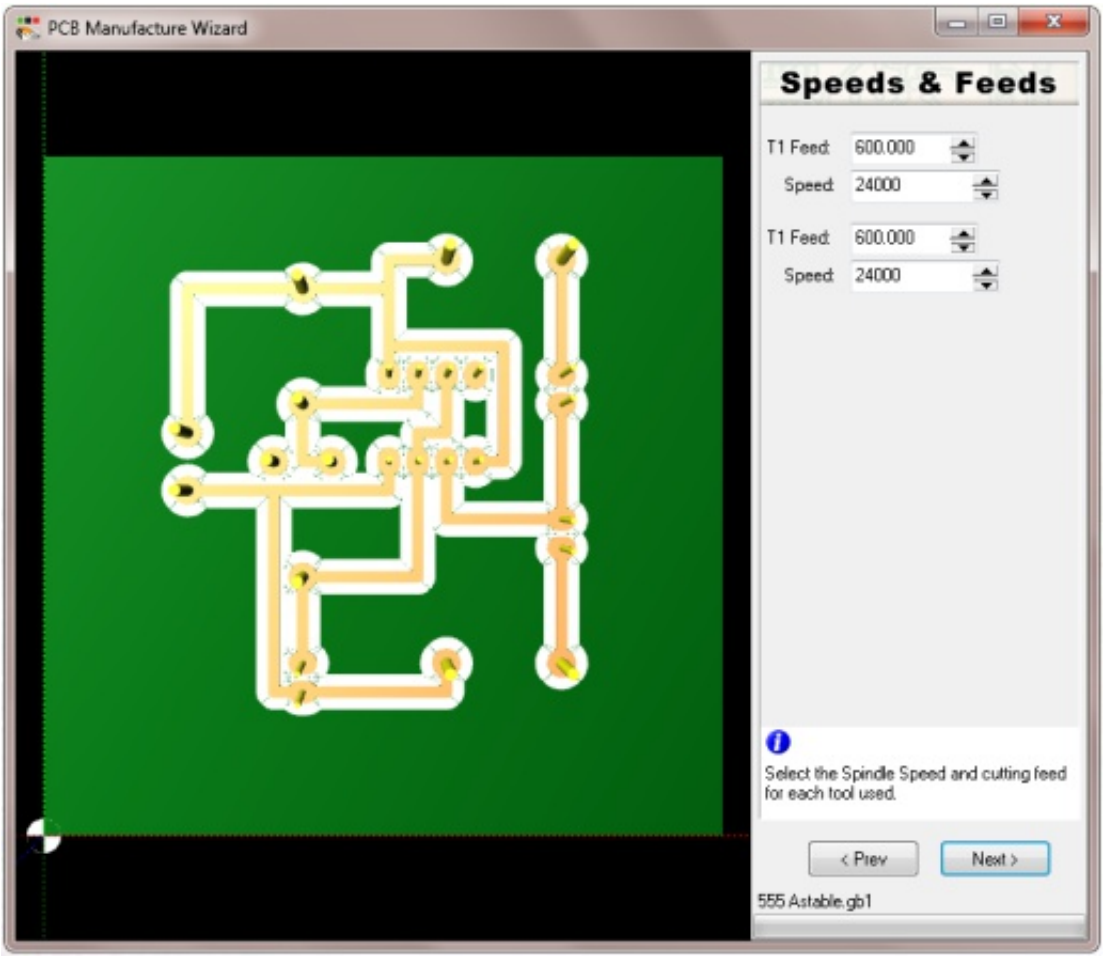
The window shown below is where you set up the parameters for drilling.
If you have a separate drill file for your PCB you can load it here, by default the centre of each pad is selected for drilling.
Here I have selected Pilot Drill Only, with this selected the tool selection for tools 2 and 3 are greyed out and the depth will be the same as the tracks.



Select Pilot Drill Only

"Left click" the Next button

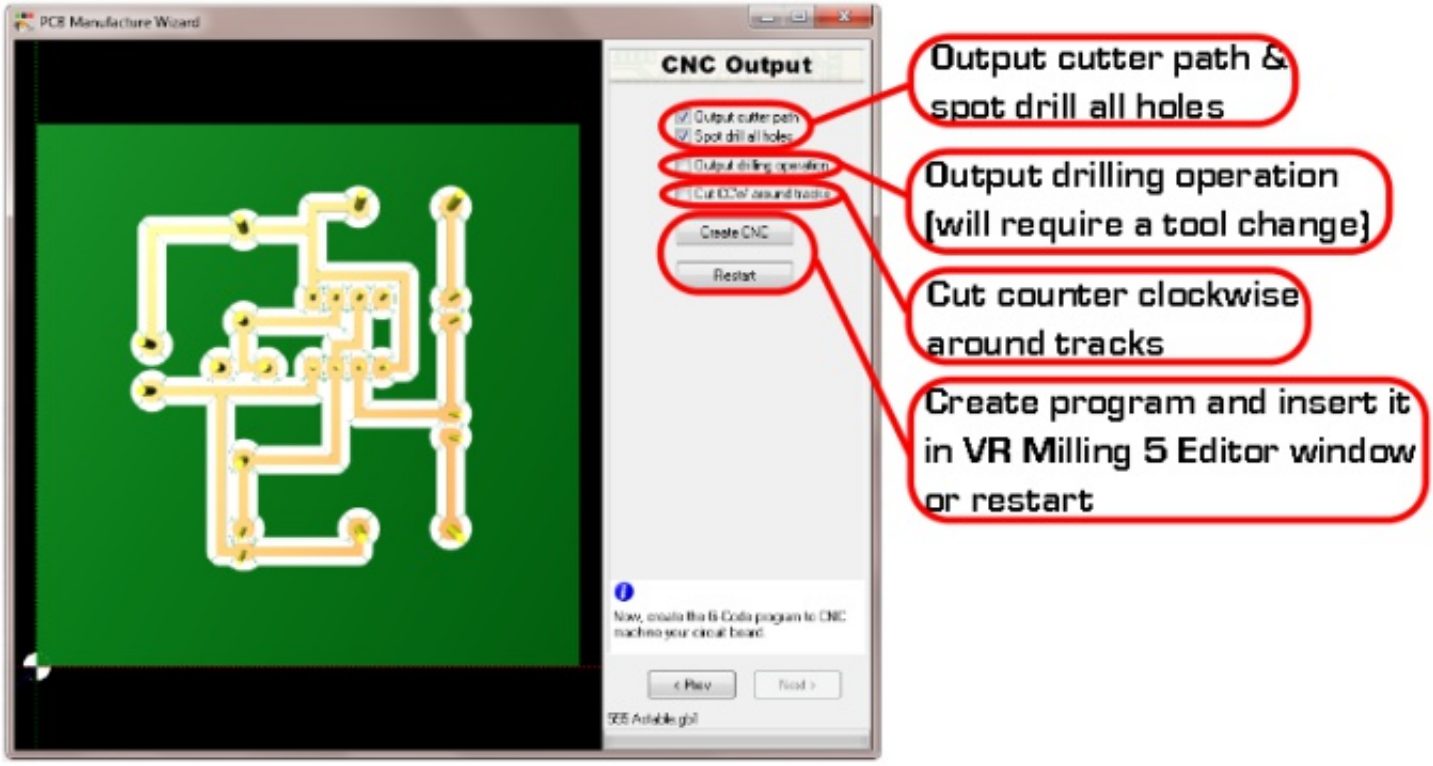
The window shown below is where you set the feedrate and spindle speeds.
The default speeds are for the Copper Clad board supplied by Denford Product Code 4x40079 and the PCB Engraving Cutter Product Code BI00811Z
If using a different type of board and/or cutter these may need adjusting



"Left click" the Next button

The window shown below is the final window and it is here that you select how the CNC program is generated.

As we are spot drilling the holes we have not selected "Output Drilling Operation"



To create the program and return to VR Milling follow the instruction below

"Left click" the Create CNC button

The program is now in the Editor window and is ready to be run

DENFORD

4th Axis Programmable Rotary Fixture

COMPLETE WITH QUICKCAM 4D MILLING SOFTWARE

4TH AXIS PROGRAMMABLE ROTARY FIXTURE

for use with
MRC 40
Compact 1000 Pro
Router 2600/Pro/ATC
Router 6600/Pro/ATC
Vertical Router
(also available for VMC1300/Pro with the exception of flood coolant models).

QUICKCAM 4D MILLING SOFTWARE

(Supplied FREE with the Denford 4th Axis Programmable Rotary Fixture.)

An easy to use, wizard based CAM package specifically designed for use with the Denford 4th Axis Programmable Rotary Fixture. QuickCAM 4D Milling imports 3D files from most 3D CAD packages and converts these into 4th axis CNC program data for output to the range of Denford CNC Routers. Users are guided through a series of simple steps, defining billet size, model orientation, machining strategy and axis of rotation before generating the appropriate CNC output file.

QUICKCAM 4D MILLING FEATURES

True 3 Dimensional model-making capabilities. Seamless integration with VR CNC Milling software. Circular, spiral and linear machining strategies. User definable limits allow for workholding avoidance. Supports both roughing and finishing paths. Resize, orientate and centre the model. Autoscale of model to fit the workpiece.

SUPPORTED OUTPUT FORMATS

CNC controllers for Denford CNC Routers.

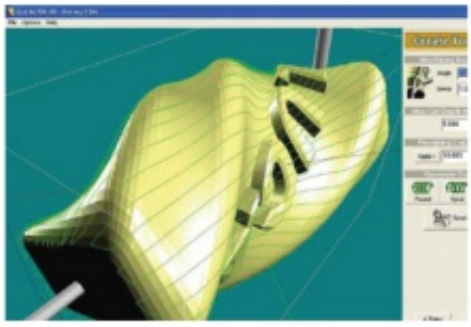
SUPPORTED INPUT FORMATS

3D Stereo Lithography *(STL) files

*Used by 3D design packages such as Autodesk Inventor, Solid Works, Pro/DESKTOP and ArtCAM.

SYSTEM REQUIREMENTS

IBM or 100% Compatible PC,
Pentium III, 1Ghz, 512MB RAM,
200MB Free Hard Disk Space,
Microsoft Windows XP; NT; 2000; Vista; Windows 7, 8
CD-ROM Drive,
OpenGL 3D Accelerator Graphics Card with 128MB RAM supporting at least 1024 x 768 screen resolution.
CNC Machines require USB Connection.
EasySCAN requires 1 USB Connection



DENFORD

Denford reserves the right to alter machines and software specifications without prior notice. All Denford products are subject to copyright. All brands and products are trademarks or registered trademarks of their respective companies.

Get in touch...
T: +44(0)1484 728000
denford.co.uk

denfordata.com/bb/

On-Line Technical Forum

TECHNICAL SUPPORT AVAILABLE 24 HOURS A DAY, 7 DAYS A WEEK

Denford's Technical Forum is a free of charge on-line technical support service that is available to Denford customers 24 hours a day, 7 days a week.

"The technical forum has provided a wealth of information and support for our 20-year-old Denford CNC machine, in fact just as good as the support we receive for our brand new CNC Router!"

Denford's On-Line Technical Forum is a free of charge service that can be accessed 24 hours a day, 7 days a week.

The On-Line Technical Forum is available to Denford customers, old and new, and it couldn't be easier to use. Just visit http://www.denfordata.com/bb/ and register on line.....it's that simple.

Denford's On-Line Technical Forum opens up the traditional communication channels that can restrict customer and technical support, due to availability of staff, teaching commitments or different time zones.

A multitude of topics relating to Denford machines and software (both new and old) are covered within the forum, which is simple to search, and easy to use.

Denford's Technical Team and Denford customers from around the world regularly log on to the forum to offer support and advice and, most importantly, post a solution for all to see.

As well as offering comprehensive technical support, Denford's On-Line Technical Forum enables customers to share ideas and projects with other users. Media such as teaching material, project work, PDF's, images, drawings and text documents are easily attached to messages for all users to view and comment on.

You can also read the latest Denford news before anyone else, and keep track of machine and software upgrades, some of which can be downloaded direct from the Technical Forum web site.

The On-Line Technical Forum has proved to be hugely popular with customers. One recent user posted a note to inform us that the Technical Forum has "provided a wealth of information and support for our 20-year-old Denford CNC machine, in fact just as good as the support we receive for our brand new CNC Router!"

Of course the traditional methods of phone and email are still available, but try out this new service by simply logging on to www.denfordata.com/bb/ and register.

Scan this to view our website

denford.co.uk ISO9001 Compliant

DENFORD
T: +44(0)1484 728000
F: +44(0)1484 728100 Email: info@denford.co.uk
Denford Limited, Armytage Road, Brighouse, West Yorkshire HD6 1QF, England