DENFORD

CNC Milling/ Routing Training

www.denford.co.uk

Table of contents

Introduction

VR CNC Milling is a Windows based software package allowing full editing and control of CNC files, either offline (away from the CNC machine) or online (controlling the operation of a CNC machine).

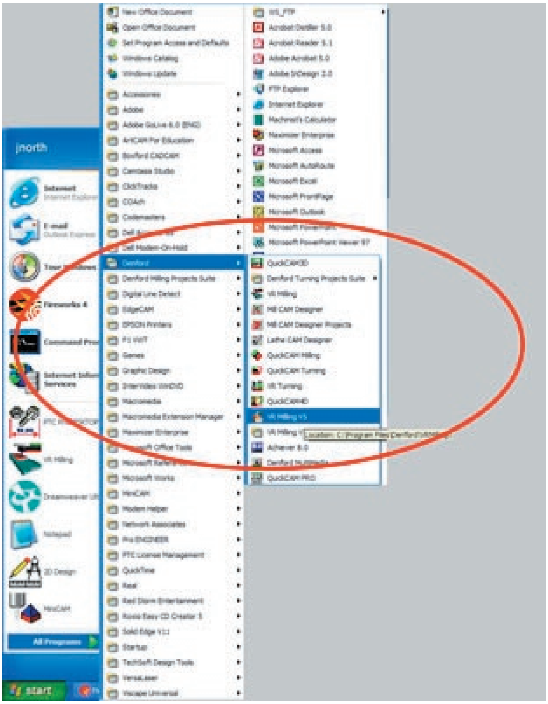
The VR Milling V5 software contains detailed help files including tutorials and animations. Access these by going to Help on the menu. See the appendix in this document for more details

As you move through the different areas of the software you will see this icon  if you need help about the area of software you are in, click this icon to see context sensitive help.

Step1 - Start the VR Milling V5 software

To start the VR Milling software double-click the VR Milling V5 shortcut icon (if available) on your desktop.

If the shortcut is not available, click “Start” on your “Windows” Start button followed by the “Programs” option, the program group “Denford” and finally the “VR Milling V5” icon.



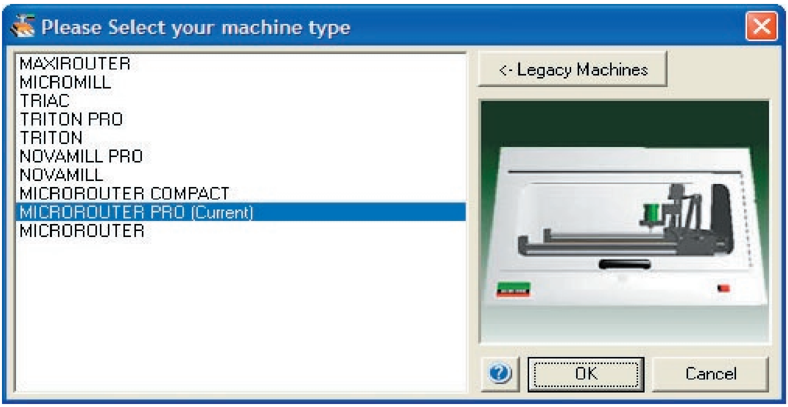
Step 2 - Configure the software for the machine

Ensure that the software is configured for the machine you are going to use.

The text at the end of the main title-bar indicates the type of Denford CNC machine that you are currently able to control with the software. In the example screenshot below, the “MICROROUTER PRO” text indicates that a Denford Microrouter Pro can be controlled by the software.

To change the name of the Denford CNC machine that can be controlled by the software:

1. Click the “Setup” menu and choose “Select Machine ...”
2. Highlight the name of the machine required and click [OK]
3. You may need to look at the CE identification panel on your Denford CNC machine to identify the name of your CNC machine

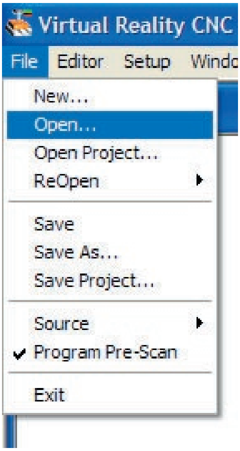


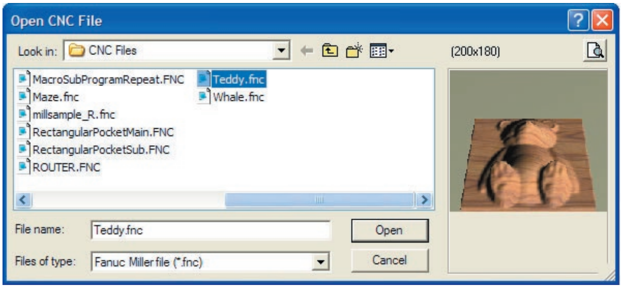
Legacy machines: These are older machines that are no longer in production but are still compatible with VR CNC Milling V5 software. Click the [Legacy Machines] button to list these types of machine.

Step 3 - Load your CNC file.

Click the “File” menu and select the “Open” option.

Browse to the drive and folder containing your CNC file – look for files with the extension letters “.fnc” then [Open] the file.

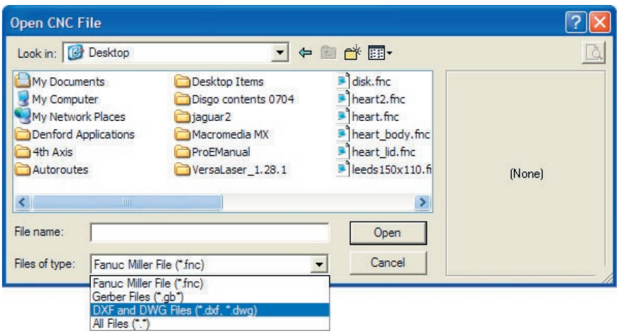




The contents of your CNC file will be displayed in the Editor window. As the name suggests, CNC files can be further edited here or you could even write one from scratch.

Loading alternative file types

By using the 'drop-down' menu labelled 'Files of Type:' two alternative file types can be opened, these are: Gerber files for creating PCB's and DXF/DWG files for creating programs from 2D line drawing data.



If either of these file types are opened a simple wizard guides the user through a series of steps to create a CNC file to create the component. See the help files within the software or the appendix in this document for more information.

Step 4 - Configure the tooling

Click the [Tool and Offset Editor Window] button and the window will open.

Click the "Tooling Data" tab.

Each tool used in your CNC program must be defined here, failing to do so will cause an error message when running a simulation.
The length and diameter of the tools shown in the VR, 3D and 2D simulations are taken from this table, for the simulations to be accurate the correct tool sizes need to be defined.

Adding a new tool to the list
A new tool can be added to the list by:
a) Selecting a blank tool in the list, then entering all the values for that tool in the right hand section of the window. Note: a new tool created here can be added to the 'Tool Library' by clicking the button pictured or by right clicking on the tool and selecting "Save tool to Library" from the pop up menu.
b) Selecting one of the pre-defined tools in the 'Tool Library'. This can be done by clicking the button pictured or by right clicking on a blank tool and selecting "Insert Library Tool" from the pop up menu.

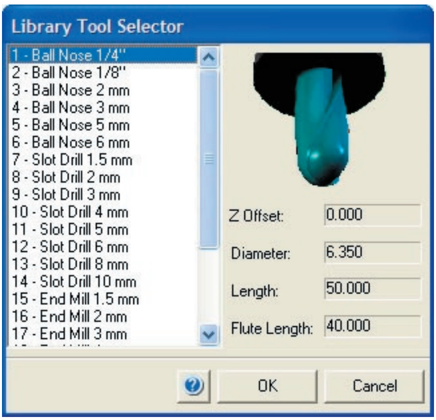
"Save tool to Library"

"Insert Library Tool"

In this example we are going to load a ¼" Ball Nose cutter, which is a versatile tool supplied with the Denford range of CNC Routers.

To add the ¼" Ball Nose cutter, highlight tool position 1, then click the "Insert Library Tool" button.

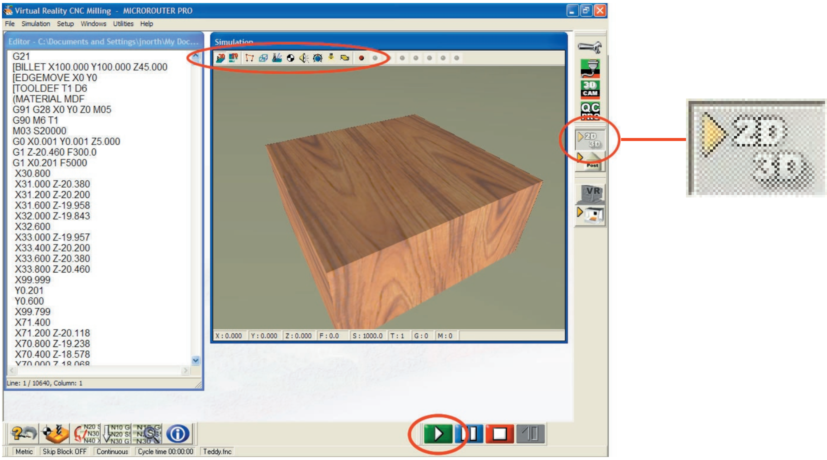
Choose the tool from the "Library Tool Selector", click [OK]. The ¼" Ball Nose cutter should appear in tool position number 1.



Click the [Tool and Offset Editor Window] button again to close the Window down.

Step 5 - Running a simulation.

Click the 2D/3D Simulation Button and the “Simulation” window will open. Click the Play button on the file control toolbar to begin the simulation.



Click the “Turbo” button to speed up the simulation, the graphics will be generated periodically, not “on the fly”.

Left click (hold) and move the mouse to rotate the view. Right click (Hold), and move the mouse to zoom the view. Hold both the left and right mouse buttons to move/pan the view.

Under the “Simulation” menu choose “Billet Materials …” to change the appearance of the simulation including the material with which the model is rendered.

Use the buttons at the top of the simulation window to toggle between a 2D and a 3D view, to display the toolpaths, to hide and show the billet, and to play machining sounds among other things.

Most CNC files will have the datum position (0,0,0) in the near, top, left corner of the billet, however with programs like Denford QuickCAM 3D and QuickCAM Pro it is possible to shift the datum position.

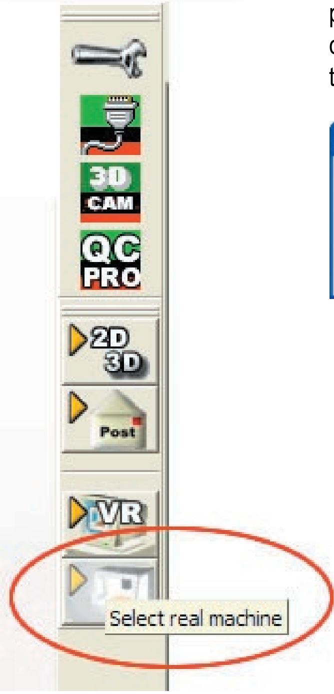
To get an accurate simulation the datum can also be shifted in the simulation.

This option can be found in the “Simulation | Datum Position...” menu.

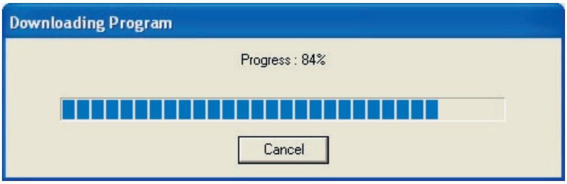
Select a position on the block to define the datum used in the CNC program.

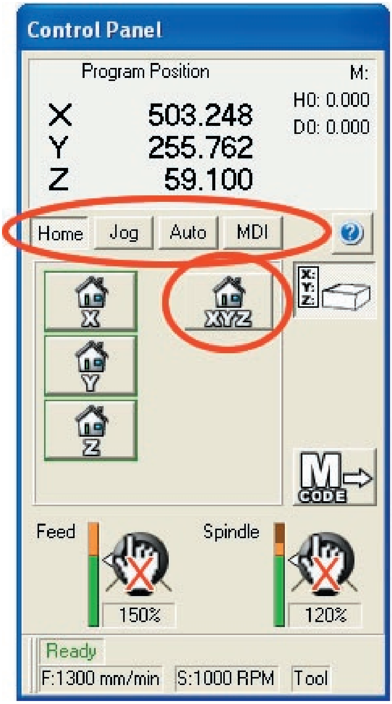
Step 6 – Connect to, and Home the CNC machine

At this point ensure that the cable is connected from the PC to the Machine (either RS232 or USB) and that the machine is switched on.



To connect to the CNC machine, left click the [Machine] button. Depending upon which machine you are using, a progress bar may appear, allow this to reach 100% and a connection will be established between the machine and the PC.





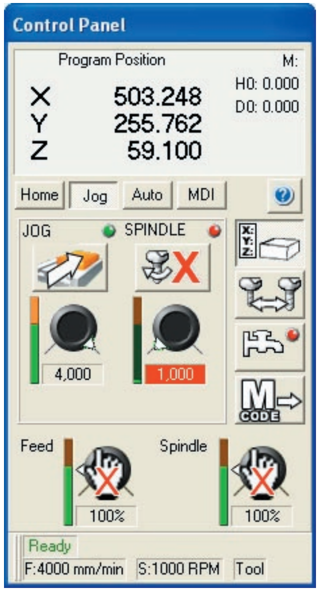
Once a connection has been successfully established, the machine "Control Panel" window will appear.
At the moment, only the "Home" tab is available. Click the [Home All] button to home all three machine axis.

After Homing, the "Jog, Auto and MDI" tabs become available, as shown right.

Step 7 - Move the machine head and fit the cutting tool.

The position of the machine head (the cutting tool) can be manually controlled using Jog mode. In the “Control Panel” window, click the “Jog” tab to select Jog mode.

To change the position of the machine head quickly, click the [Jog] button until a straight arrow is displayed, signifying ‘Jog Continuous’ mode.

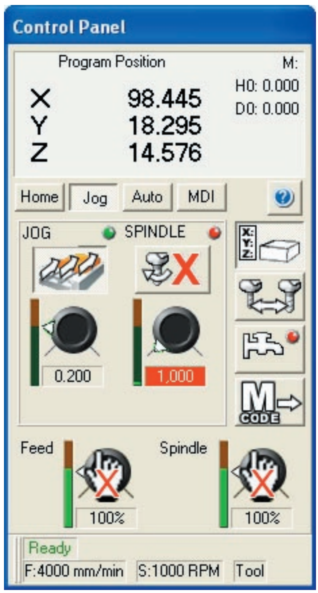


"Jog Continous"

Click and drag the Jog Feed control knob to the top of the scale. The feedrate value is shown in the readout below the control knob.

The four cursor (arrow) keys, and the [Page Up] and [Page Down] keys on the keyboard, are used to control the X, Y and Z axes. Press and hold the appropriate key to move the required axis.

To change the position of the machine head incrementally, click the Jog button until the image changes from a single arrow to three small, stepped arrows, signifying Jog Step Mode.



"Jog Step"

Click and drag the Jog Feed control knob to adjust the increment. When you press the cursor keys the cutter will move by the amount set.

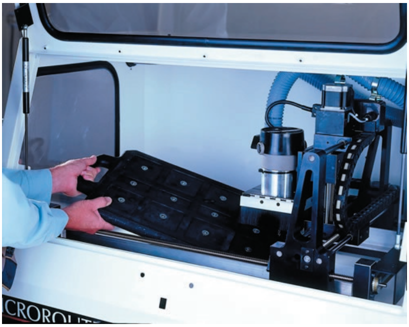
Jog the machine head to an appropriate position, then, fit the cutting tool. The procedure for this will vary depending on the machine type. See the machine manual for more detailed information on this procedure.

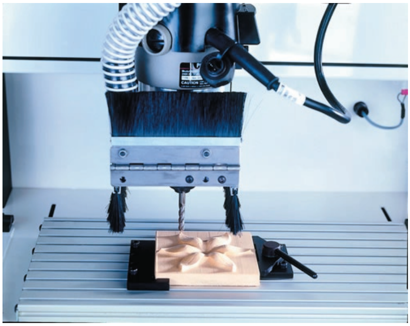
For the Denford Microuter, Microrouter Pro and Microrouter Compact you may find it easier to remove the motor to change the tool.

Step 8 - Fit the material in the machine.

There are various methods for holding material in the CNC machines, the one you choose will depend on the type of job you are undertaking and the machine you are using.

On the Denford Microrouter/Microrouter Pro a vacuum pump is fitted to the machine and using a vacuum table, the material is literally “sucked” to the vacuum table. This is a very simple and effective method of securing the material. If you are cutting all the way through the material however you may want to protect the vacuum table from becoming damaged.

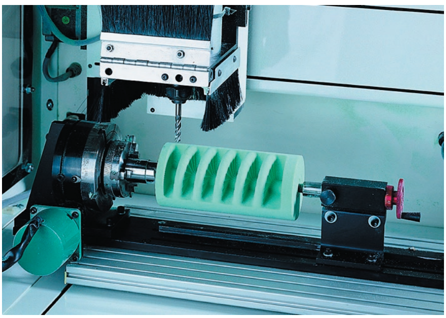


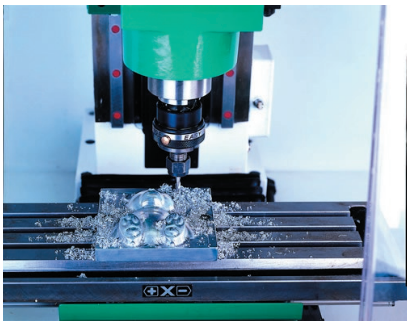


Parallel clamps are supplied with the Denford Microrouter/Microrouter Pro which enables a piece of material to be “sandwiched” between the clamps which is then elevated from the base of the machine. This would be a suitable method of securing material when it is to be cut all the way through.

The Microrouter Compact has a “T slot” table which allows for more traditional clamping arrangements. The Microrouter Compact is supplied with an easy to use, lever-operated, clamping kit.

Milling machines such as the Denford Novamill, Triton and Triac have options including a “Mitee-Bite” clamping kit which is quick and easy to use.





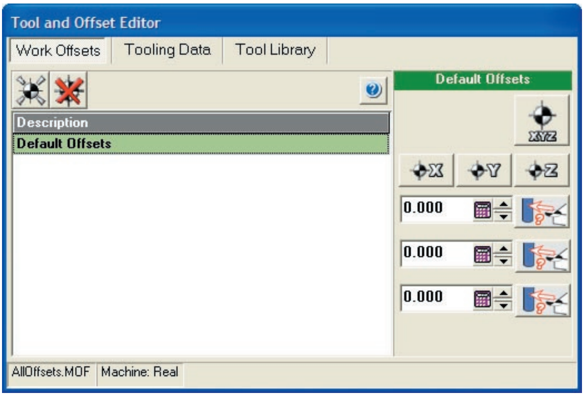
Special jigs or fixtures are available for the “F1 team in schools” competition and there is a “4th axis” fixture available, which rotates the material as it is cut, enabling production of fully 3D models in a single operation.

Step 9 – Set the work offsets

What are offsets?

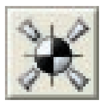
Offsets are the distances the cutter needs to travel, from its 'Home' position to the point from which the program starts in X Y & Z.
Usually the point at which the program starts (the datum) is the upper, front, left corner of the material although some software including Denford QuickCAM 3D and QuickCAM Pro allow the user to specify a different datum position.

Click the button to show the “Tool and Offset Editor”

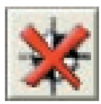


Create a new work offset

It is possible to store a number of offsets and swap between them for different jobs. Use this facility to create a new offset and add it to the list.
•Click the "New work offset" button.
•Click on the ‘blank’ offset that has been added to the list to select it.
•Type in a description for your new offset.



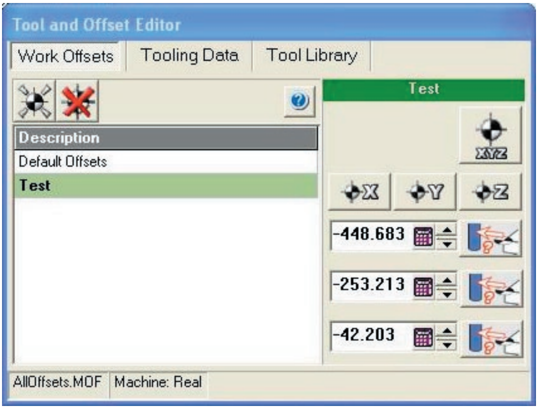
"New work offsets"



"Delete current selection"

To delete an offset you can either:
•Select the offset and click the “Delete current selection” button
•Right click on the offset description and select “Delete Offset” from the pop-up menu.

Activate the offset

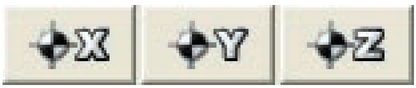


Make sure the Offset you have just created is Active. It is possible to have a number of Offsets stored and change from one to another. The machine will use the Offset which is currently active. To activate an Offset, select it in the list and either, click the Activate button or Right Click on the Offset in the list and choose Activate from the menu. The Active Offset is highlighted in green.

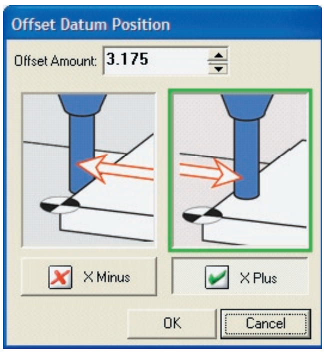
Using the techniques described in Step 7, Jog the cutter until you are touching the material in the datum position. For most jobs it is sufficient to visually sight the cutter over the X and Y axis and then incrementally jog the cutter until it is touching the material surface in Z.

When the cutter is in position click the  to set all 3 axes at once.

1. The axes can be set individually by moving the cutter to the correct positions and setting the offsets one at a time.



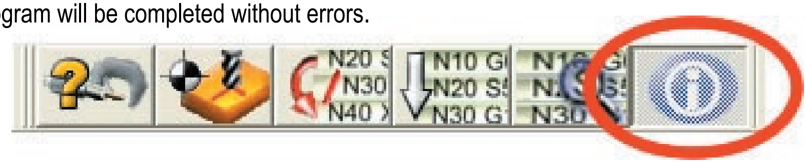
2. To set the offsets more accurately, touch the cutter against the side of the material and use the “datum offset button” at the right of each offset value to offset the datum value. This is usually used by touching the cutter up to the side of the material, and using the cutter radius, offset the datum value so that the center of the cutter is exactly over the edge of the material. It is the radius of the currently active tool which appears by default as the value in the “Offset Datum Position” window, this value can be changed. 



When the offsets have been set there will be three values assigned to the “Test” offset. This is the distance from the machines “Home position” to the position the cutter is currently in.

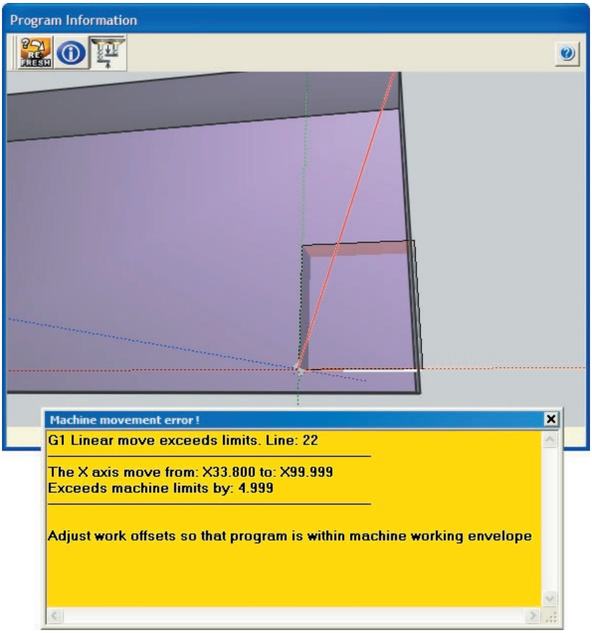
Step 10 – Verify the Work Offsets and the Program.

Before we run the program we can use the “Program Information” function to verify that the program will be completed without errors.



A common error is when a program “Exceeds Limits”, this error occurs when part of the toolpath is outside of the CNC machines working envelope.

If an error is detected a window appears on the screen similar to the one shown below.



This message appears if the CNC program is attempting to move to a position outside the cutting area of the machine. The window displays the first line in the CNC program that contains the ‘over travel’ movement and by how much.

There are basically two reasons why this could happen:

a) The workpiece and or tool length offsets are inappropriate for the program.

b) The overall size of the component you are trying to machine is too large for the CNC machine working envelope

There is a problem with the above example because the billet is positioned too far over to the right. We need to re-position the billet nearer to the left hand corner of the machine table and redefine the offsets.

The X offset value must be changed in the tools and offsets editor.

To close the window down click the “Program Information” button again.

Step 11 – Run the Program.

The program is now ready to be run. To run the machine you must click on the 'Auto' tab.

The program must be run from the beginning, to ensure this is the case click the Stop button, followed by Rewind and finally click the Start button.

The program will begin to run. A message may appear asking you to change to Tool Number *. Check you have the correct tool and click [OK]. The spindle will start and the program will begin.

At the bottom of the Auto tab are the Feed rate Override and Spindle speed Override controls. If the machine you are using is fitted with Potentiometers it is these which are used to override the Feed rate and Spindle speed. If not you can affect these using the mouse.

Tip: to gain more control, the feed rate can be reduced to gradually feed in the cutter until you are happy and then increased.

Spindle and Feedrate potentiometers

Turbo Mode

Click the [Turbo Mode] button to switch Turbo Mode on. This can be done at any time, even when the program is running. The 'turbo mode' feature has been developed to reduce the machining times of large 3D programs and complex 2D programs. For larger programs E.G. more than 100 lines, turbo mode on will usually make the machine perform with a smoother motion. It is recommended that programs produced from 3D CAD/CAM software are run with turbo mode on.

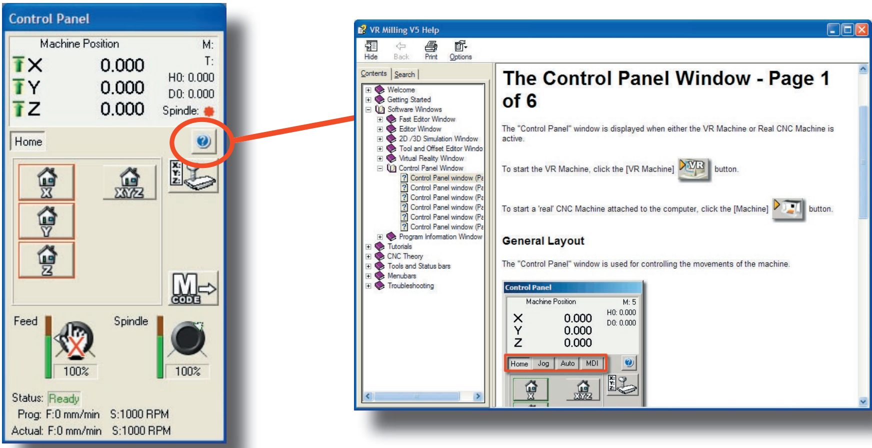
Notes:

Appendix:

Help Files and tutorials

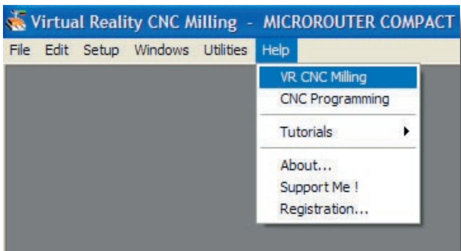
The VR Milling V5 software contains detailed help files including tutorials and animations. Access these by going to **Help** on the menu.

As you move through the different areas of the software you will see this icon:  if you need help about the area of software you are in, click this icon to see context sensitive help.



The full VR Milling V5 help files can be accessed by choosing **Help > VR Milling** from the main menu at the top of the screen.

There is also an option to view help files on CNC programming.



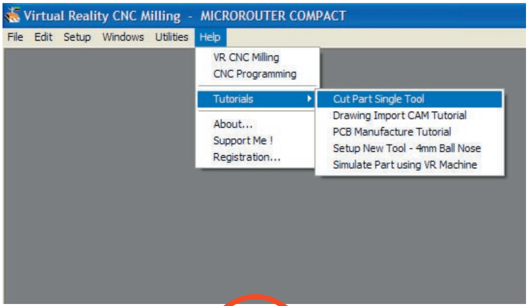
This help file covers the stages involved in producing the coded instructions, used by the CNC machine to make the component. These coded instructions are called the part program.

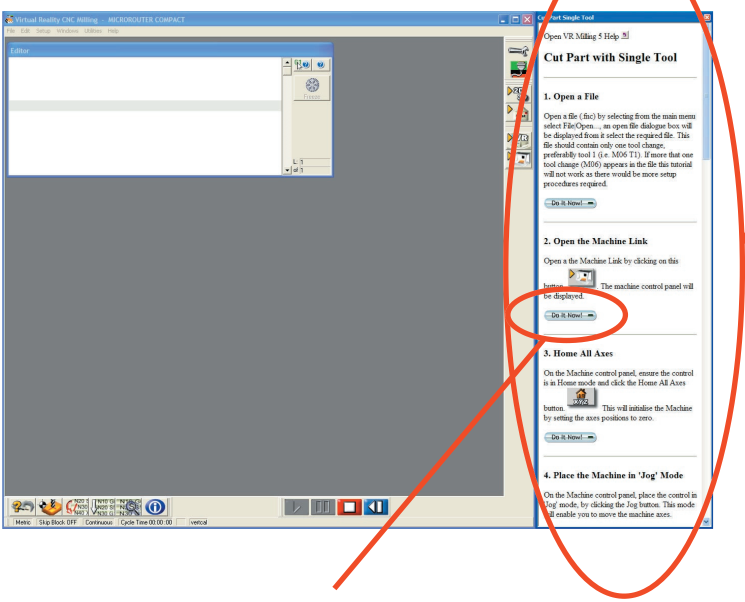
Each part program contains a number of different codes, the most important being the collection of G and M codes.

The helpfile is divided into three main sections, each with their own list of topics:

1) Basic Programming
2) G Codes
3) M Codes

Also found under the **Help** menu are various tutorials. The tutorials launch in a window next to the main VR Milling V5 window and guide the user step by step through the chosen topic. Topics include: Cutting a part with a single tool, Drawing (DXF/DWG) Import CAM tutorial, PCB manufacture, Setting up a New Tool and Simulating a Part using the VR Machine.





Tutorial window open next to the main VR Milling window

Many of the tutorials have **[Do it now!]** buttons which will automatically carry out the operation. The content of this guide is covered by the tutorials if you are having problems following these instructions or do not have access to this guide the tutorials should help you.

Dxf/Dwg file import

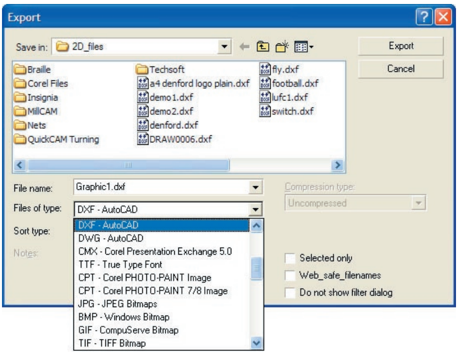
VR CNC Milling has the facility to open 2D vector drawing data in dxf and dwg formats (version 5.10 onwards). When dxf or dwg files are opened in VR CNC Milling, a wizard is launched which guides the user through some simple steps in order to manufacture the file. The wizard uses central machine tooling and material libraries within VR Milling to ensure only the tools you have available can be used and that suitable speeds and feeds are automatically generated.

The following tutorial gives an explanation of the steps involved in manufacturing a dxf or dwg drawing using Denford VR CNC Milling.

Creating a Dxf/Dwg file

Most CAD packages will create a dxf file and many good graphics packages will also create them. AutoCAD, ProDesktop, Corel Draw and Techsoft Design Tools are among a few of the packages capable of creating the dxf or dwg file format. The process of creating the dxf or dwg file will vary from package to package but is usually found under the **File | Export** menu option.

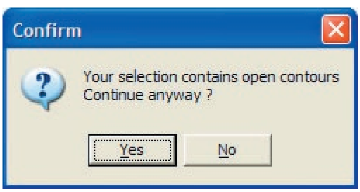
It will usually be necessary to select the type of file you want to export from a drop-down menu. This is shown in the image, right, taken from the export menu in Corel Draw. See the help files in your CAD software for more information.



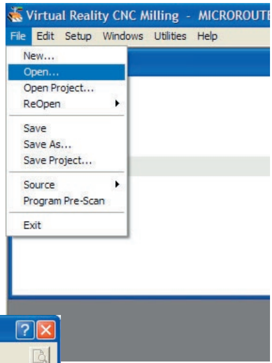
Preparing the DXF/DWG file

The Dxf/Dwg file import wizard within VR CNC Milling does not enable the user to edit the file in any way. The file cannot be scaled within the wizard. The user must therefore edit and size the file correctly in the design software before exporting the finished Dxf/Dwg file.

To apply some of the machining plans available in the wizard the vectors must be closed i.e: there should be no gaps in the shapes within the design. If you want to remove all the material within an area, the boundary around the area must be closed. If not the following message will be displayed



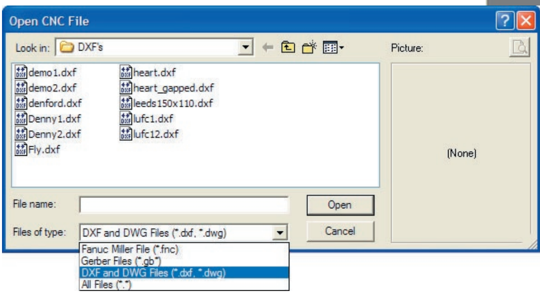
Load the Dxf/Dwg file



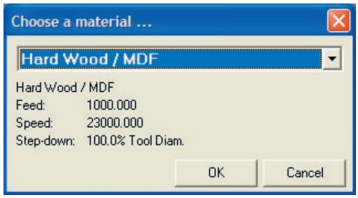
In the VR Milling V5 software click the “File” menu and select the “Open” option.

Browse to the drive and folder containing the Dxf/Dwg file that has been exported/saved from your CAD package.

You will need to click the drop-down menu labelled "Files of type" and choose **"DXF and DWG Files (*.dxf. *.dwg)"**



Material Selection



When you have located the DXF or DWG file you would like to manufacture click **[Open].**

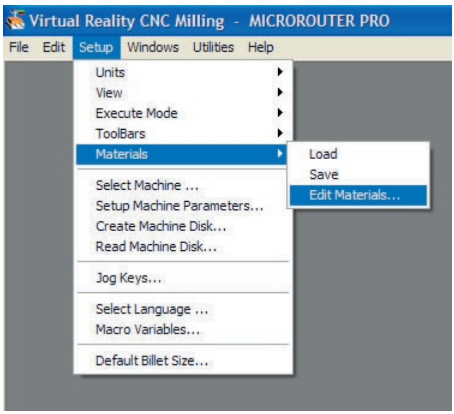
You will then be prompted to choose a material to manufacture your design from.

Once you have chosen a material and clicked **[OK]** the main CAM wizard will be launched.

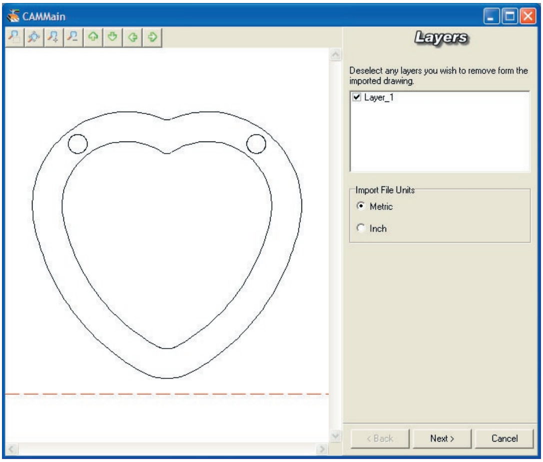
By choosing a material from the list the spindle speed, feedrate and depth of cut will be predetermined. You can add, edit and save material types within VR Milling V5.

To edit materials go to "**Setup | Materials | Edit Materials...**"

You will be prompted for a password, by default this is "denny".

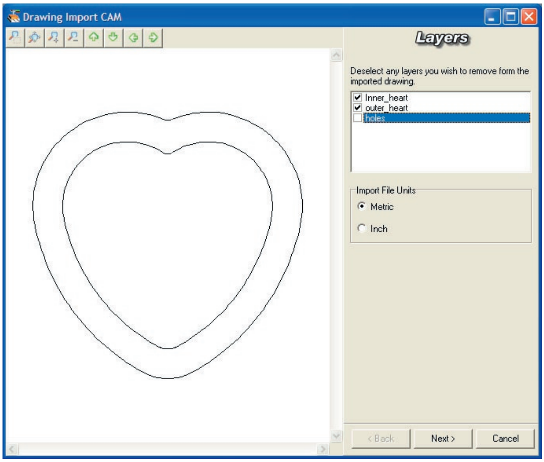


Once a material has been chosen the Main CAM (Computer Aided Manufacturing) Wizard is launched and your Dxf file will be dispayed. The first screen is called 'Layers'. DXF files can be exported with Layer information. Layers can be deselected so that only the required information can be seen.



In the example left, the heart has been drawn on one layer. This means that the layers cannot be turned on and off. A typical use for layers is to put dimensioning on a separate layer, so that if further work is needed on the drawing, the dimensioning layer may be turned off to remove the "clutter" of the dimensioning. In a similar way, if a Dxf drawing is to be cut, machined, or engraved, the dimensioning/ notes/border, or any unwanted information, may be "turned off" first

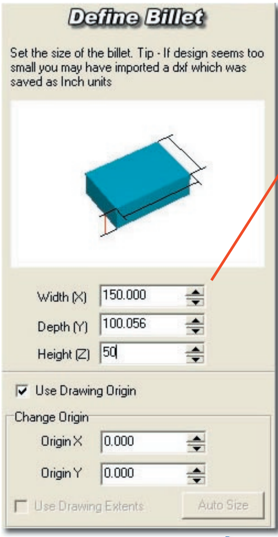
In the example right, the different elements of the heart have been drawn on seperate layers and can be switched on or off. The holes have been drawn on their own layer and have been turned off.



Input File Units

Files can be exported from the CAD software in either Metric or Imperial units so an option is available to select the correct units. DXF and DWG files cannot be scaled or edited within the CAM Wizard, if the drawing needs to be scaled this must be done in the CAD software. However, if you find that the drawing is smaller than you expect it to be, it may have been exported in imperial units, try clicking the 'Inch' radio button to see if this resolves the problem.

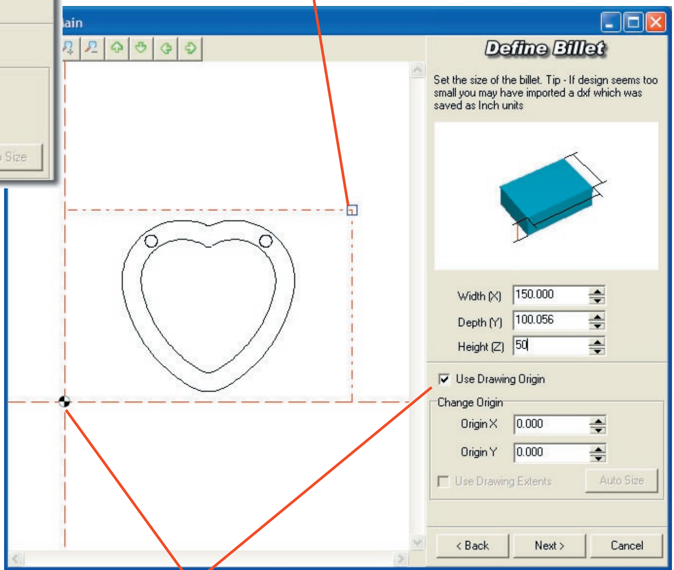
Define Billet



The Define Billet screen allows the size of the billet (material) to be defined and the position of the drawing within the material to be determined.

The width, depth and height of the billet can be entered in the dialogue boxes.

Alternatively, the blue control handle can be dragged to determine the size of the billet.



With the 'Use Drawing Origin' box checked, the datum position is taken from the datum position in the CAD software used to create the drawing.

If the box is unchecked the datum symbol on the screen can be dragged to replace the drawing origin (this effectively moves the material).

When the "Drawing Extents" check box is ticked, the [AutoSize] button will set the billet size to the extents of the drawing with no border around it. Clicking the [AutoSize] button with the drawing extents box unchecked will set the billet size with an equally spaced border around the drawing. The spacing amount is determined by the distance from the drawing origin to the nearest vector.

Machining Plans

The machining plans screen is used to compile a list of strategies for machining the component. There are 6 different 'Plans' that can be applied, they are:

• **Follow Cut** - the centre of the cutter will follow the line. This tool is used normally for engraving.

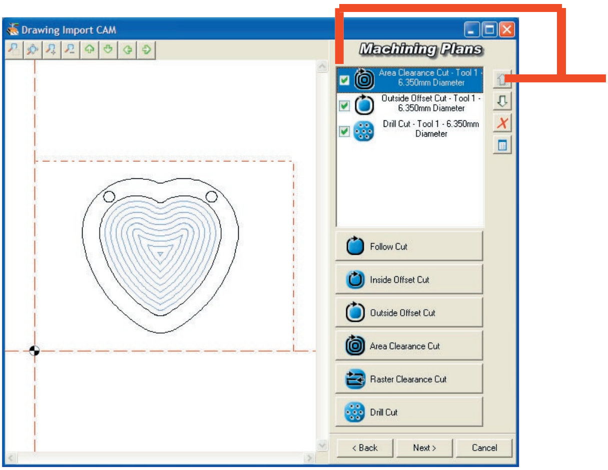
• **Inside Offset** - This tool will create a tool path inside a Vector offset by the radius of the tool diameter. (a closed vector is required)

• **Outside Offset** - This tool will create a tool path outside a Vector offset by the radius of the tool diameter (a closed vector is required)

• **Area Clearance** - This plan removes the area of material enclosed by a vector but leaves any other selected vector islands intact. If the tool cannot fit between two vectors it will remove only what is possible. The tool path created follows the outline of the vector shape (a closed vector is required)

• **Raster Clearance -** This plan does much the same as the Area Clearance plan but instead of following the contour of the shape the cutter path rasters in linear moves. The Raster toolpath is cut first then an inside offset toolpath is generated to clean up all the edges of the shape. (a closed vector is required)

• **Drill Cut -** This plan will find the centre of any circle included in the design. If different holes are to be drilled different depths then create multiple plans.



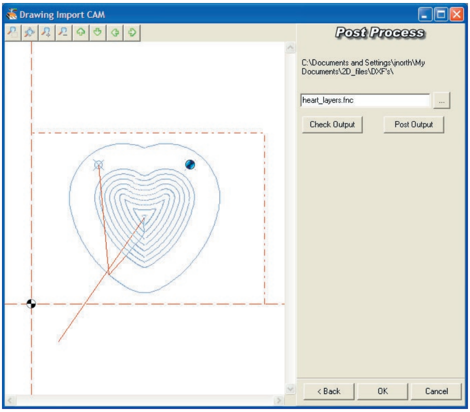
A list of plans can be compiled, plans can be reordered, turned on and off, edited and deleted using the buttons next to the list of plans.

The same plan can be used multiple times to create cuts of different depths

See the Help files in VR Milling V5.10 and above for more details on each machining plan. These can be found by accessing **Help > Tutorials > Drawing Import CAM tutorial.**

Click **[Next >]** to move to the next screen.

Check Output and Post Process



Select “Check Output” and the tool-paths are simulated in order.

This allows you to check if the order of machining is correct before creating the code.

The tool diameter is shown.

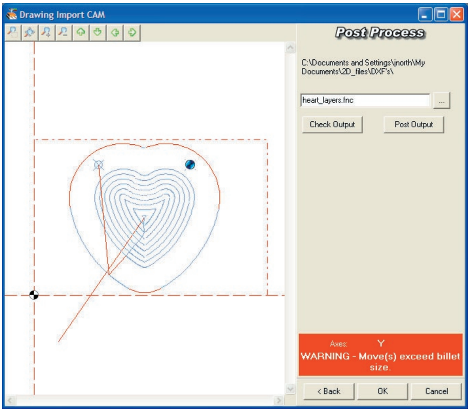
Rapid moves are shown in Red while the cut path is shown in Blue. If the depth requires multiple passes these will also be shown.

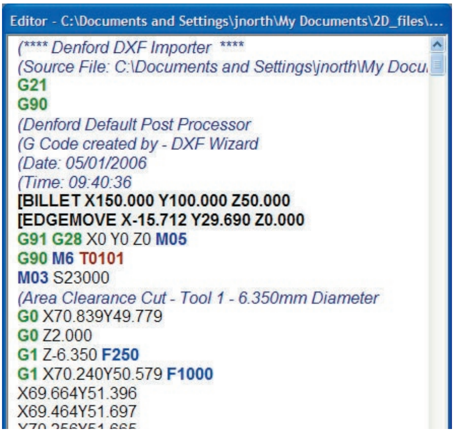
In this example the outline of the heart shaped box was cut with the “Outside Offset Cut”

As the centreline of the cutter is close to the edge of the block the actual cutter diameter will break through the side of the block.

While this may be OK the cutter could hit a clamp so a Red warning message is displayed.

This will not prevent the file being Post Processed but is displayed as a warning.





Select “Post Process”

The G&M code will be created and will open in the machine editor.

The screen shown is has the program opened in the full editor with colour formatting on.

The file can now be manufactured in the normal way.

PCB manufacturing

Gerber format (RS-274X) is an industrial standard transfer file for defining circuit board tracks and pads.
VR CNC Milling has the facility to open Gerber data (version 5.10 onwards). When Gerber files are opened in VR CNC Milling, a wizard is launched which guides the user through some simple steps in order to manufacture the PCB.
The following tutorial gives an explanation of the steps involved in manufacturing a Gerber file using Denford VR CNC Milling.

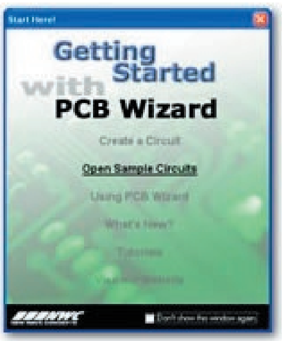
Creating a Gerber file

Many PCB design softwares will create a Gerber file. The process of creating the Gerber file will vary between PCB design software, see the help of the individual software for more details. 'PCB Wizard' and 'Crocodile Clips - Real PCB' will both create Gerber files.
This tutorial shows the process of creating a Gerber file from PCB Wizard.

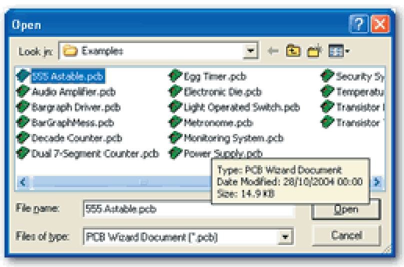
Exporting a file from PCB Wizard

In this section we will cover the settings required to export a Gerber file from PCB Wizard to make it suitable for manufacture on a CNC machine.
We will use the first sample circuit contained in the software as an example.

Open PCB Wizard.
The getting started Wizard will appear:
Select Open Sample Circuits:

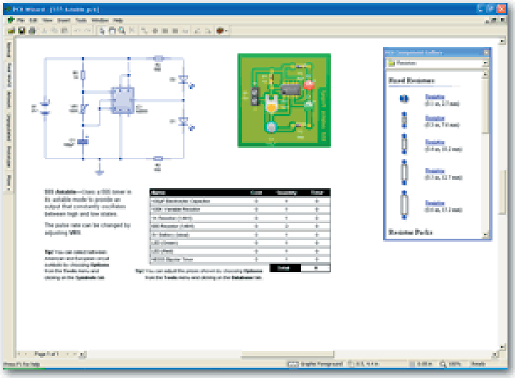


Select the 555 Astable.pcb file:

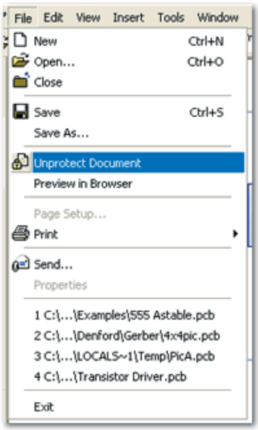


Click **“Open”**

The standard PCB Wizard screen below is displayed.



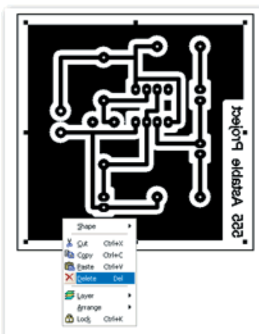
The PCB tracking shown has a large ground plain or flood fill area. This is inserted automatically by PCB Wizard and has to be removed before exporting the gerber file to be manufactured.



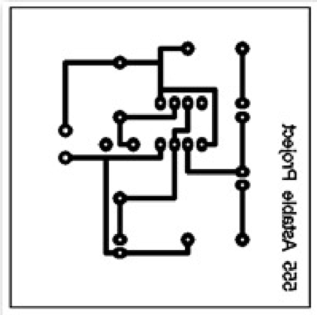
Tip - choose to view “Artwork” by selecting the button down the left hand side of the application

In the example files in PCB wizard the documents are locked and cannot be edited. To be able to remove the ground plain it is necessary to unprotect the document. To do this, select the “File” menu. Then select “Unprotect Document”

Now select the ground plain on the tracked PCB. Right Click the Mouse to get the function menu.
If all the options are greyed out then the document is protected.
Select “Delete”.

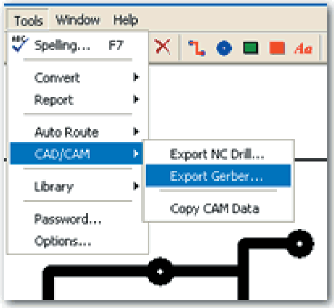


All that is now left is the track layout and identifying text label.



Machining around the text will be a problem and will probably produce errors so you may wish to remove the text as well.
If you want to remove the Text simply select it then Right click the mouse then Select “Delete”.

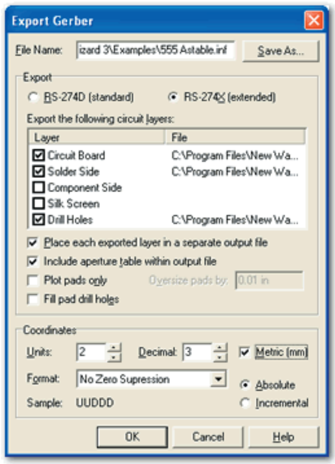
The circuit is now in a format where it can be exported as a Gerber file for manufacture.
Gerber format (RS-274X) is an industrial standard transfer file for defining circuit board tracks and pads.
Select the “Tools” menu then “CAD/CAM” and finally “Export Gerber” output:



This will then open a sub menu where it is possible to define which layers and information you wish to output. In this case we only have a Bottom Copper layer. The Export Gerber menu is shown:

Select the ”Save As” option after File Name. This allows you to define where the gerber files will be saved. Create a folder and note where it is located. Name the file and click “Save”.

The following selections (as shown in image, right) should be made.



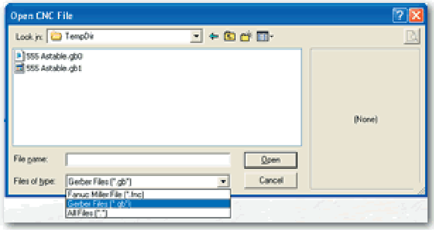
Check the following:
Circuit Board
Solder Side
Drill Holes
Place each layer in a separate output file
Drill Holes
Include Aperture table

Uncheck the following:
Plot Pads Only
Fill Drill Holes

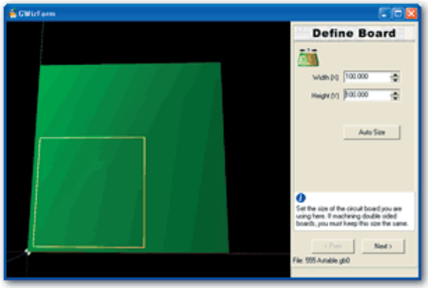
Select “OK”.
The files are automatically created and placed into the folder specified.

Importing a Gerber File into VR Milling V5

Run the VR Milling V5 Software.
Select "File" "Open".
Select the file types dropdown and select "Gerber Files (*.gb*)"
Select the "555 Astable.gb0" file
Select "Open"
The Gerber Import Wizard opens as shown:

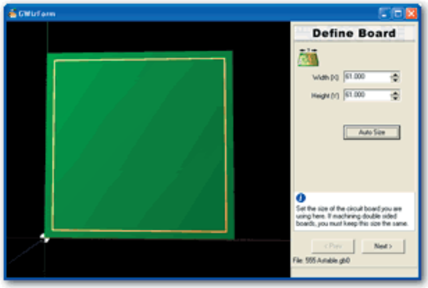


Define Board



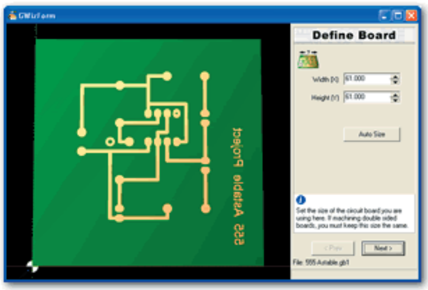
The origin is drawn in the bottom left corner of the PCB but the top right extreme is defined by the last used PCB size.
Press "Auto Size" to set the board size to match the extents of the PCB tracks.

If you know the size the board was designed at type the values into the width and height boxes.
In this case the board is defined as 61mm x 61mm in size.



As you change the values the board shown will also change in size.
The first layer imported is useful as it defines the board size but we do not want to machine this.

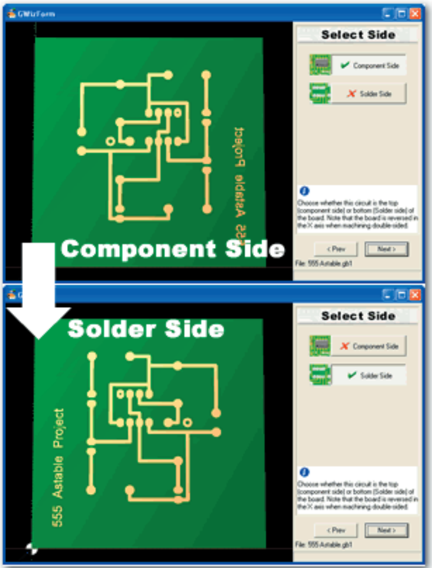
Close the Wizard (Cross at top right hand corner) and select "File" "Open" once again.
Select the "555Astable.gb1". This is the bottom copper file. Select Open.
The bottom copper artwork is opened in the Wizard as shown.



The Width and Height values should have been remembered from last time, setting the board size to: 61mm x 61mm.
Click "Next".

Tip: The image can be manipulated with the mouse. In the display window "Left click" the mouse and drag will rotate the image. Right Click and drag will zoom into the image. Left and Right together will 'pan' the image.

Select Side



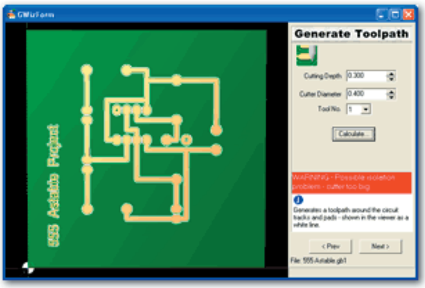
With most PCB design packages the view of the board is from the component side looking downward. By default the gerber file is also done this way. The bottom side or Solder side will be a mirror image of this when the board is turned over.

Select “Solder Side” and the image will mirror from left to right as shown:

The view is now shown as it will look when machined.
Note: For double-sided boards the component side does not need to be mirrored.
Select “Next”.

Generate Toolpath

To be able to generate the toolpath you need to define the depth you want to cut to (making sure you cut right through the copper) and the diameter of the cutter you are going to use (to ensure it can get between the pads and tracks).
A typical depth to use would be 0.3mm as this will machine through the copper and into the board behind. It is also deep enough to take out some error if the board was not flat while machining.

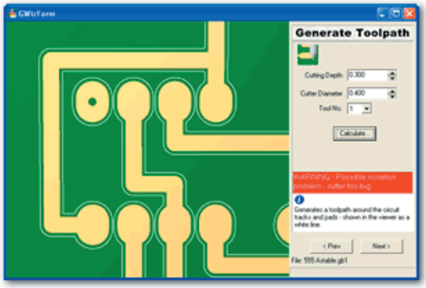


The standard engraving cutter Denford recommend is a 0.25mm tip with a 30-degree angle. Cutting at 0.3mm deep will leave a groove about 0.4mm wide.

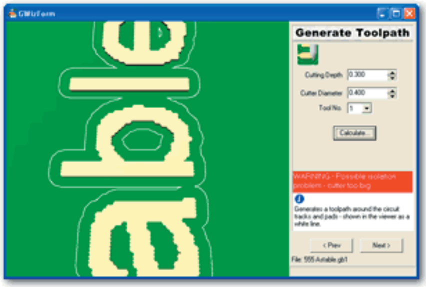
Set the Cutting depth the 0.3mm
Set the Cutter Diameter to 0.4mm
Select “Calculate”

The cutter path around the tracks is calculated and displayed offset by half the diameter so in this case 0.2mm outside the tracks.
If the red warning banner appears then there is a problem somewhere in the design and the cutter cannot pass between all the tracks.

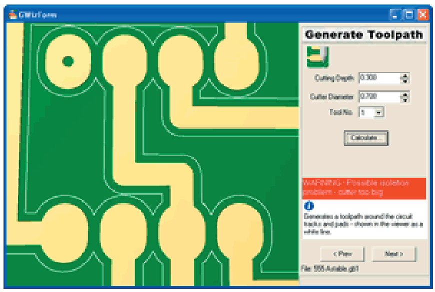
Using the Zoom and Pan tool (via mouse buttons) you can inspect the tool cutter path and search for the error. If an error is found due to the tracks being too close together then you will need to modify your PCB design or find a narrower cutter.



In this design there are no problems with the tracks but it is the text that is the problem. As the text is treated as a copper track if the tool cannot pass between the letters it will show as a possible error.
As you can see in the picture the tool path is unable to profile round the text and as a result the file has warned there could be an error.Removing the text from the PCB before exporting would prevent the error occurring.

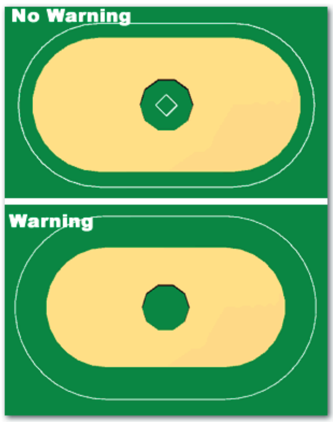


In the example shown here the tool width has been set to 0.7mm and the new toolpath calculated. The software will not allow the tool to pass through a gap that is too small as it would machine away copper that is required to make a connection. The resulting tool path would leave all the pads on the IC Base connected together.



Note also, that a pad with a hole in the center will produce this same error, if the tool diameter is too big to cut inside the hole, see image right.

Once you are happy there is no problem machining the Tracks and Pads click “Next”.

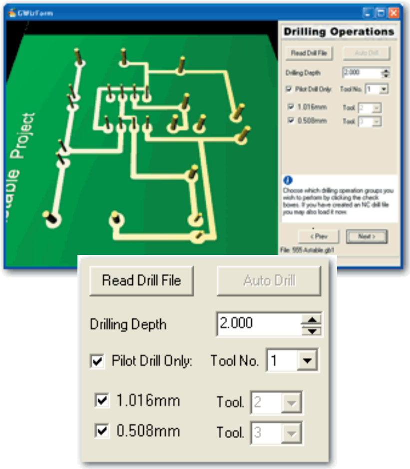


Tip: Care must be taken to ensure large enough isolation gaps were designed into the PCB before exporting as a gerber file.

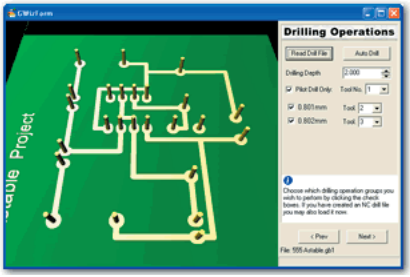
Drilling Operations

There are two options available to you when drilling a circuit board. Provided the Check Box: “Fill Drill Holes” was unchecked when the files were exported the pad information includes the internal hole sizes and the external pad diameter. As a result the Wizard can automatically calculate the number of different sized holes in the board and allocate a drill of that correct diameter to each hole.

Here the Auto Drill setting is selected and the Wizard has selected two drill sizes to be used.



The second method of drilling a PCB is to import the drill file.
To do this click on the “Read Drill File” button.
Locate the file 555 Astable.drl and “Open”.

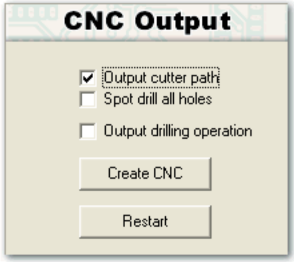


This time the drill centres are shown with a default drill diameter. Again the two different sizes of drill are shown but the correct diameters are not given.
In this instance you have to read the “ini or report” file to find the diameters of the drills required and ensure they are fitted when the tool number is requested.
All drilling operations that are grouped by tool size, can be overridden and all drilled with the same tool. To do this, check the box “Pilot Drill Only” and select the drilling tool number. Doing this will cause all holes (that are turned on) to be drilled with the one tool, in effect pilot dilling them for opening out later.
Select “Next”.

Speeds and Feeds

A default spindle speed and tool feedrate is allocated to each tool used in the program. These feeds are the suggested values Denford recommend but each one can be customised as required.
Once the values are approved, select “Next”.

CNC Output



It is possible to output the CNC code to cut the following combinations:
The tracks only
Drilling the holes only
Spot drilling only (to the same depth as the track cut depth, using the track cutter tool)
or, any combination of the above.

It may be that you want to just output the track outline and then drill the board later on a pillar drill, if so uncheck the “Output drilling operation” as shown.
If you want the drill output recheck the box.

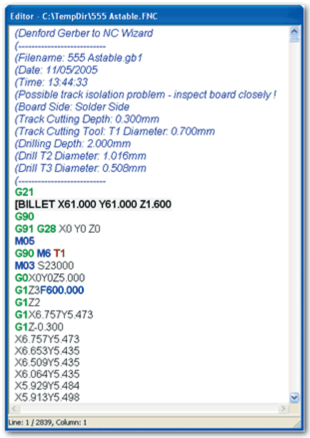
Click **[Create CNC]**

The Wizard software will close and re-launch VR Milling 5. The newly created toolpath is loaded into the current editor.

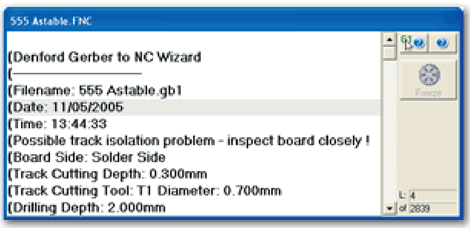
The example shown shows the file loaded into the full editor with colour formatting on:

The section at the start of the program (program header) has comments within brackets.

These comments tell you about the program, how it was created. They also includes information on the tools to be used, track side to be machined and a warning if there were isolation problems while processing.



The fast editor should be used for manufacture and is shown below:



The program is now ready to be manufactured in the normal way.