PCB Production Tutorial

(c) Denford Ltd 2005

_____

Introduction

In response to requests to produce circuit boards with machine tools as opposed to acid etch, Denford have developed a file import for both Gerber files and NC drill files.

The principles of etching boards required the minimum amount of copper to be removed to save the etch solution becoming saturated and ineffective. As a result on most PCB's a ground plane would be created to fill the areas of the board that are not used.

When machining boards a boundary around the tracks and pads is cut to a depth greater than the layer on the copper clad board. Ideally the minimum amount of cutting will be done and as much clearance around the tracks as possible should be given. If there is an option then the "Area Fill or "Ground Plane" will be removed.

Machining PCB's has a disadvantage over etching in that the minimum track pad isolation gap has to be increased to allow the cutter to pass between the two.

As a result the tooling to be used for machining should be taken into account before designing the circuit.

In most cases when machining a PCB the profiling tool will cut a path about 0.4mm wide so the minimum isolation gap that should be designed is set to this value or preferably higher (0.5mm).

When saving the file as a Gerber format file the board boundary should be defined either on the screen, board boundary or with a copper track outline.

_____

PCB Production
Tutorial

(c) Denford Ltd 2005

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.
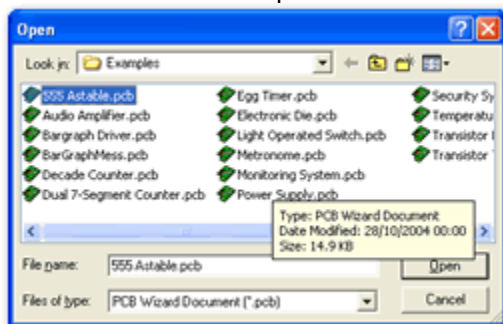We will use the first sample circuit contained in the software as an example.

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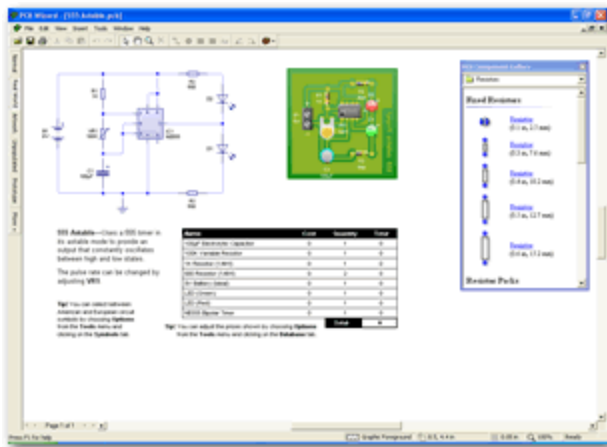Select Open Sample Circuits:


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" :
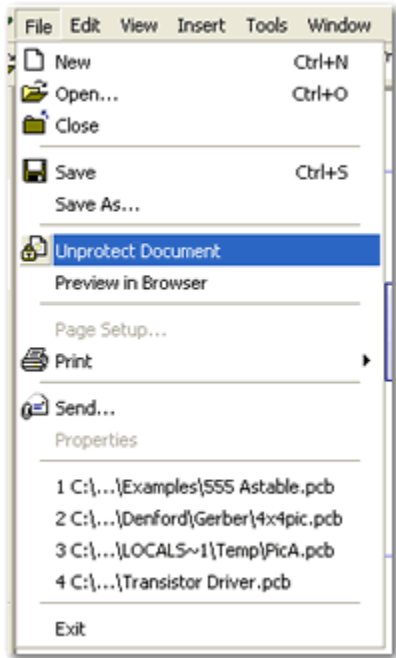
Now select the ground plain on the tracked PCB. Right Click the Mouse to get the function menu.
If all the options are greyed out then the document is protected.
Select "Delete".
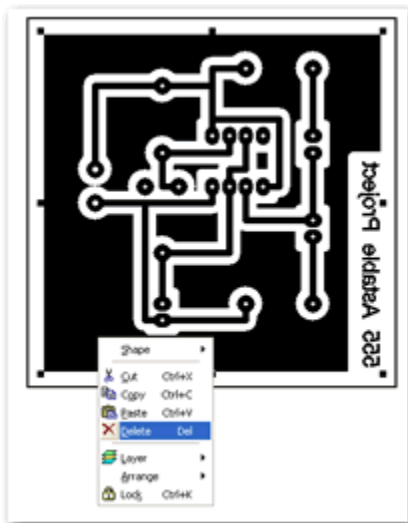
All that is now left is the track layout and identifying text label.
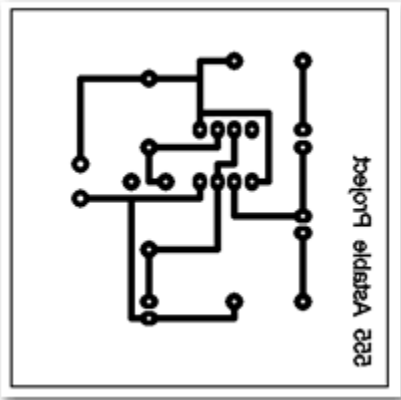
Machining around the Text will be a problem and will probably produce errors so you may wish to remove the text as well.

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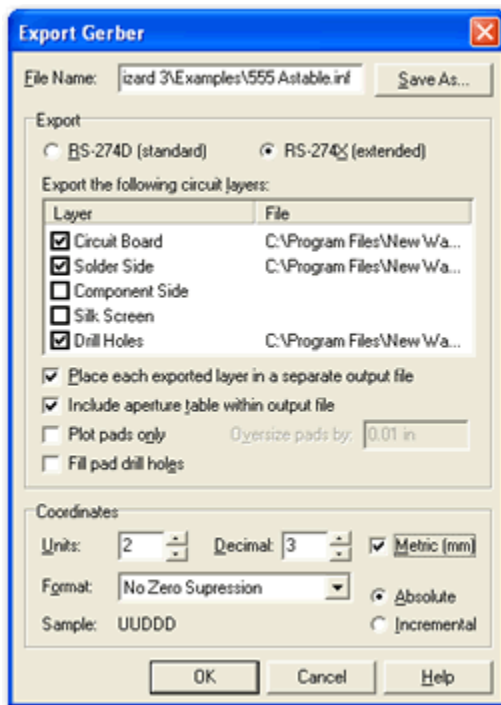Select the "Tools" menu then "CAD/CAM" and finally "Export Gerber" output:
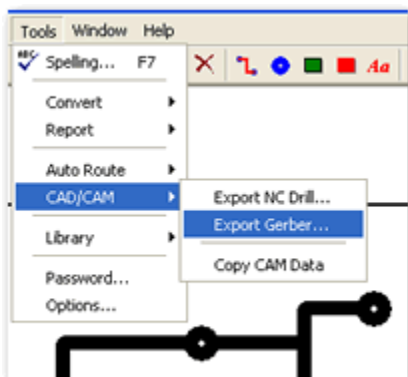
This will then open a sub menu where it is possible to define which layers and information you wish to output.
In this case we only have a Bottom Copper layer.
The Export Gerber menu is shown:

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 above image) should be made.

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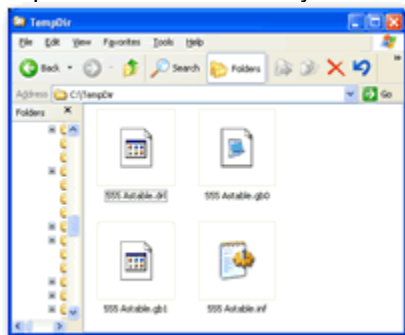Drill Holes
Include Aperture table

Uncheck the following:
Plot Pads Only
Fill Drill Holes

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

Explore to the folder and you can view the files as shown:

Right click on the Drill file and select the "Open" or "Open With" option. Select Notepad from a list and the drill file will be opened. This file shows a list of X, Y co-ordinates that correspond to the centre of each of the drill holes. It also shows that there are two tools used. T01 drills most the holes while T02 drills the last two:
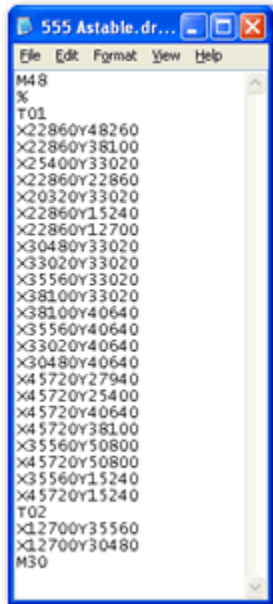
Next open the Astable.INF file. This is a report file and again can be opened in Wordpad. If you cannot see this file in the folder it may be the folder display properties on the computer you are using need to be changed. From other PCB packages this file may have a different file extension.
This file defines the pad sizes and drill sizes as well as information about the unit, date created etc:

The other two files in the folder are the gerber output files for the Circuit Board outline and the solder side tracking.
While PCB Wizard automatically names each layer GB0 to GB3 depending on the number of layers exported. Most packages add the layer to the reference file name.
The standard file format is .GBR for Example the output file would be: 555 Astable-bottom copper.gbr

Congratulations ! The file export has now been successfully completed.

PCB Production
Tutorial

(c) Denford Ltd 2005

Importing a Gerber File into VR Milling v5

The Gerber import was introduced into VR Milling 5 with the release of Ver 5.7 (May 2005)
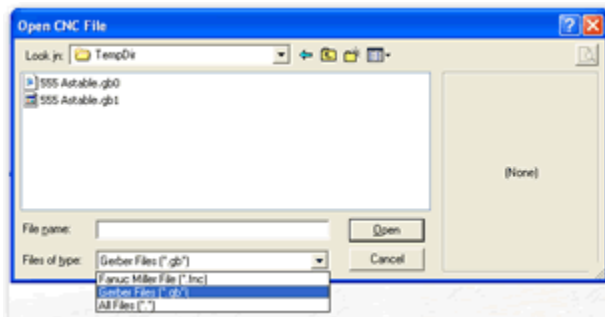To check your current version go to the Help Menu then select "About".
If you have a previous version of VR Milling 5 you can download an upgrade CD free of charge from the Denford Website.

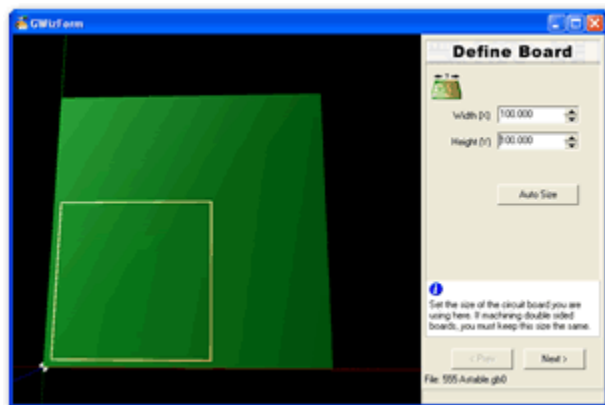Run the VR Milling 5 Software.
Select "File" "Open".
Select the file types dropdown and select "Gerber Files (*.gb*)"

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.
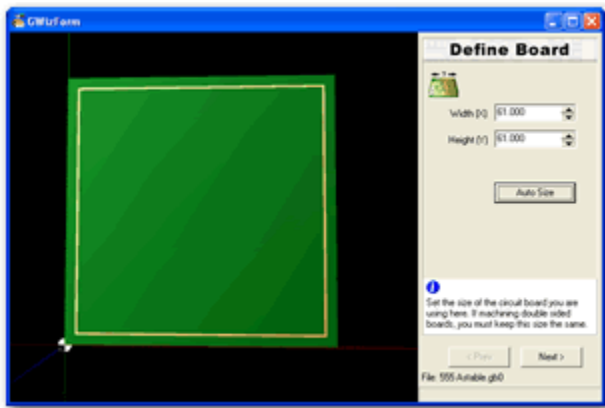
Close the Wizard (Cross at top right hand corner) and select "File" "Open" once again.
Select the "555Astable.gb1". This is the bottom copper file. Select Open.
The bottom copper artwork is opened in the Wizard as shown.
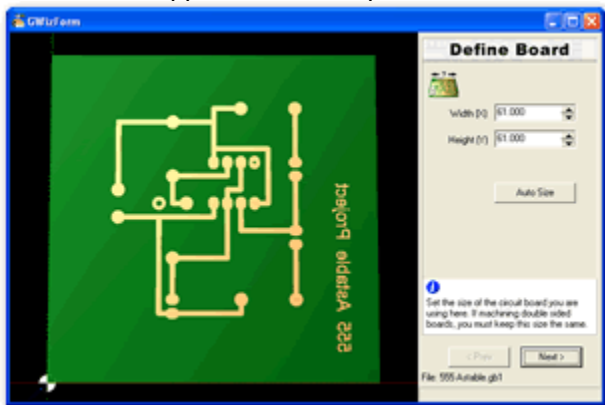
The Width and Height values should have been remembered from last time, setting the board size to: 61mm x 61mm.
Click "Next".
TIP: The image can be manipulated with the mouse. In the display window "Left click" the mouse and drag will rotate the image. Right Click and drag will zoom into the image. Left 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:
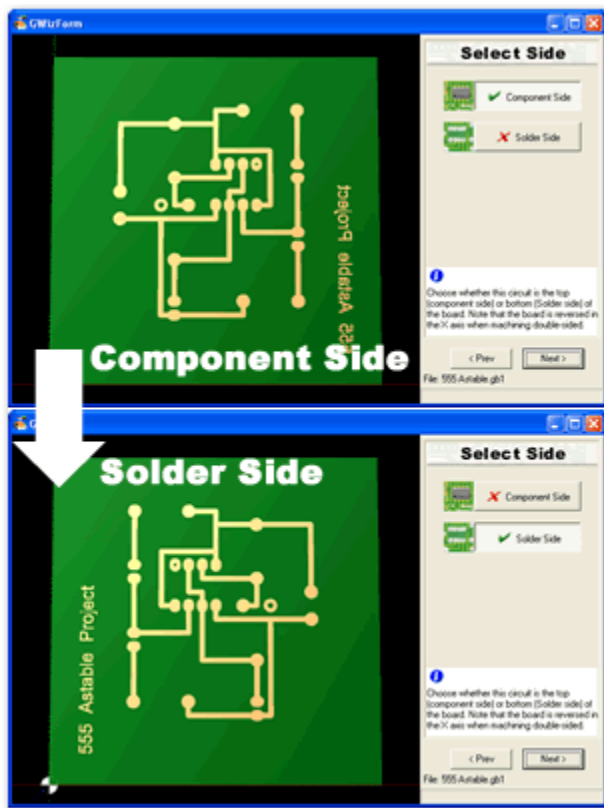
The view is now shown as it will look when machined.
Note: For double-sided boards the component side does not need to be mirrored.
Select "Next".

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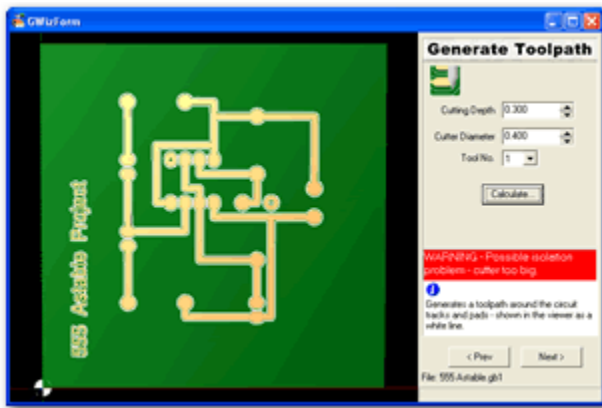
The cutter path around the tracks is calculated and displayed offset by half the diameter so in this case 0.2mm outside the tracks.
If the red warning banner appears then there is a problem somewhere in the design and the cutter cannot pass between all the tracks.

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.
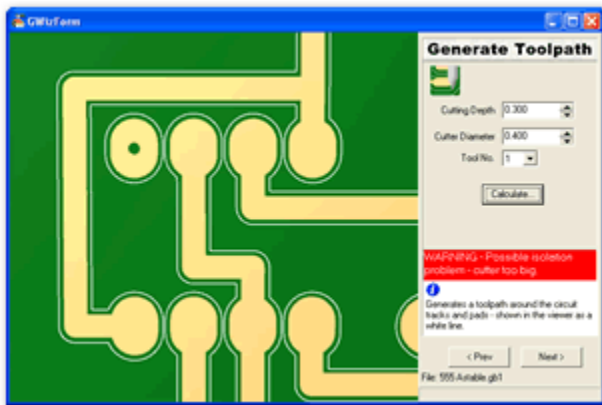

In this design there is no problem with the tracking but it is the Text that is the problem. As the text is treated as a copper track if the tool cannot pass between the letters it will show as a possible error.
As you can see in the picture the tool path is unable to profile round the text and as a result the file has warned there could be an error.
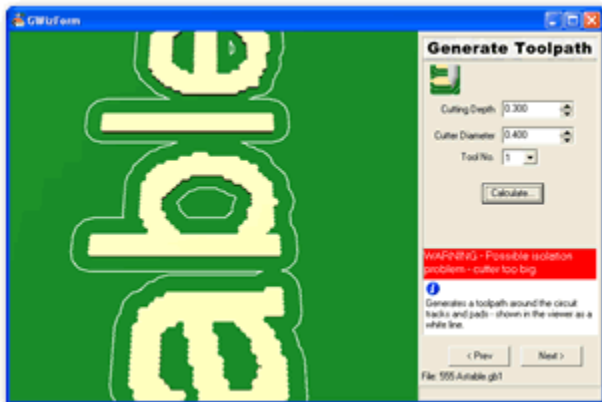

Removing the text from the PCB before exporting would prevent the error occurring.
In the example shown here the tool width has been set to 0.7mm and the new toolpath calculated.
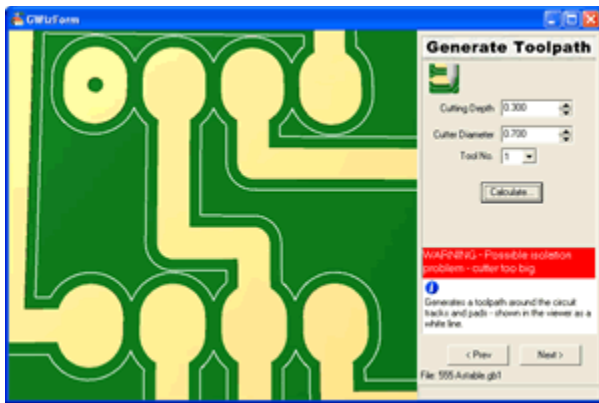
As the software will not allow the tool to pass through a gap that is too small as it would machine away copper that is required to make a connection the resulting tool path would leave all the pads on the IC Base connected together.

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, eg:
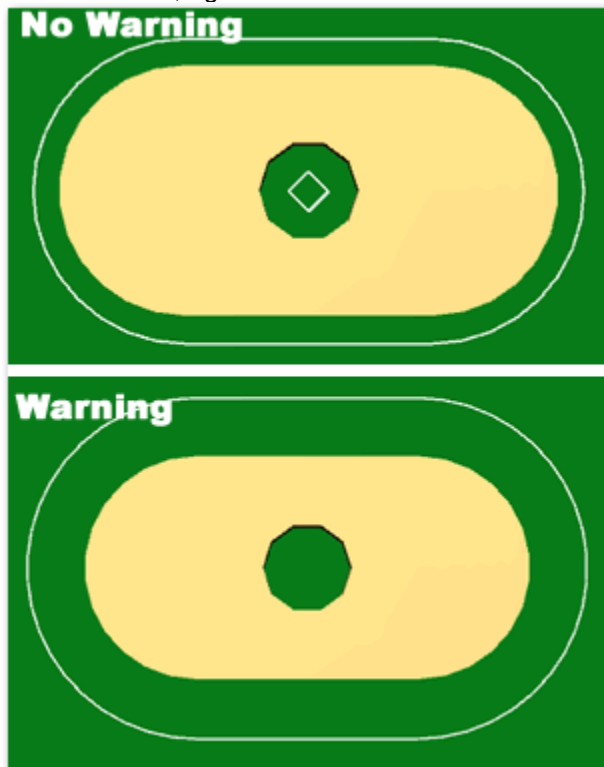

Once you are happy there is no problem machining the Tracks and Pads click "Next".
Tip: Care must be taken to ensure large enough isolation gaps were designed into the PCB before exporting as a gerber file.

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.
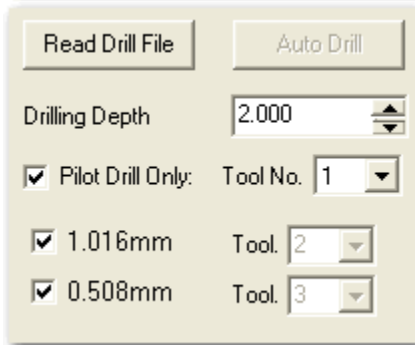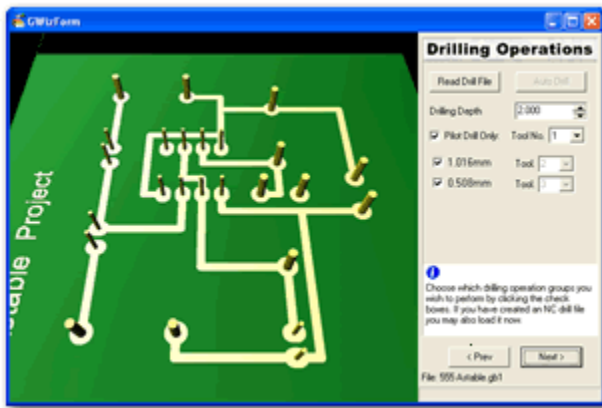
The drill depth can be set to any value but in this case it is set to 2mm so as to pass right through the board.
A normal PCB is only 1.6mm thick.
It is possible to stop holes being drilled by un-checking the box next to the drill diameter.

Here the 0.508mm drill is unchecked and you can see the drill indicators are removed over the IC base:

If the box is left unchecked then the CNC file will not include the drilling detail for that tool.

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".
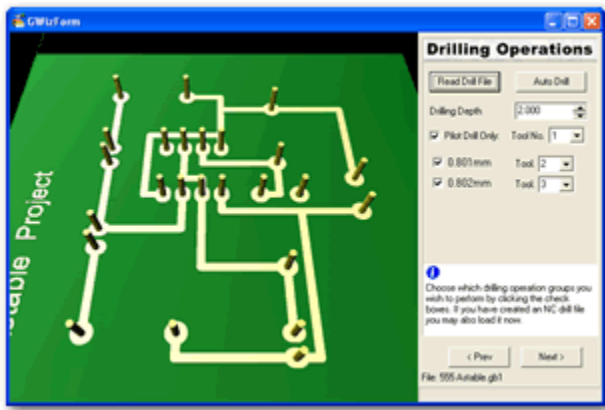
This time the drill centres are shown with a default drill diameter. Again the two different sizes of drill are shown but the correct diameters are not given.

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.
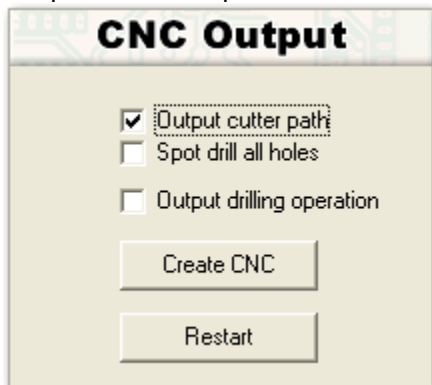
Select "Next".

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 latter on a pillar drill, if so uncheck the "Output drilling operation" as shown.
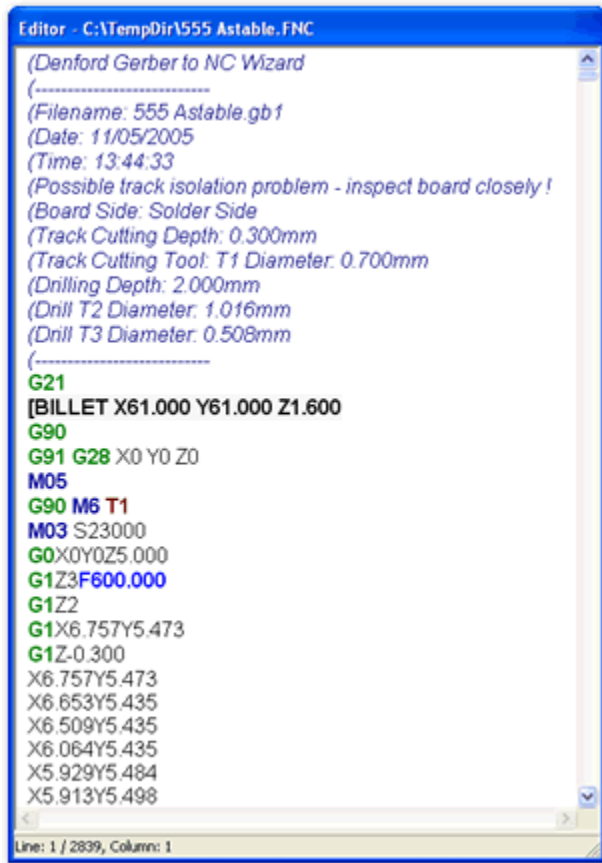If you want the drill output recheck the box.

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

editor.

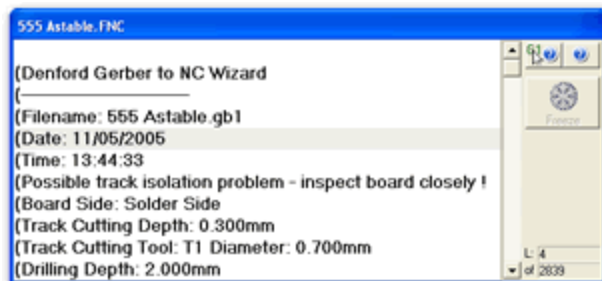The example shown shows the file loaded into the full editor with colour formatting on:

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.