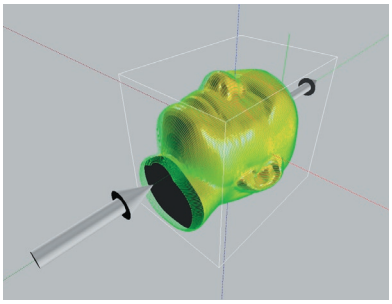
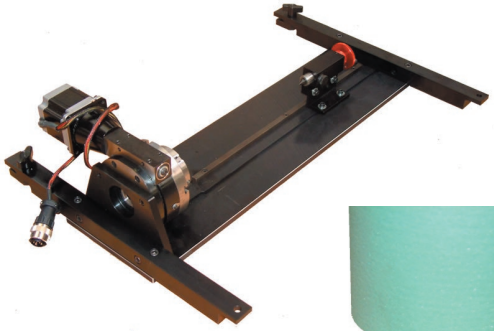
DENFORD

4th Axis Manufacturing Training - QuickCAM 4D and VR Milling V5 software with 4th Axis Programmable Rotary Fixture





www.denford.co.uk

Table of contents

Introduction

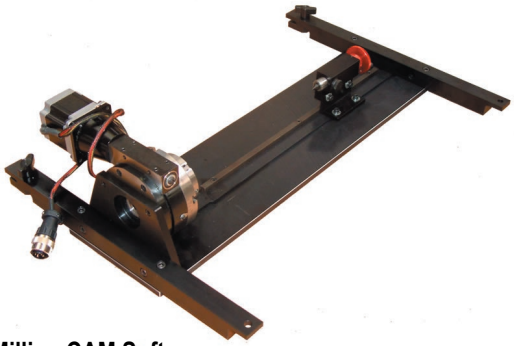
This tutorial guide leads you through the process of converting a 3D model into a CNC file using Denford QuickCAM 4D software and its manufacture with a 4th axis programmable rotary fixture on a Denford CNC miller / router.

These models can be created using the following CAD software :

Pro/DESKTOP v2000i2, Pro/DESKTOP v8, Solid Edge (or any software that can export a model as an STL file (*.stl).
The process of converting a 2D/3D program into a 4th axis rotary program is documented at the end of this guide.

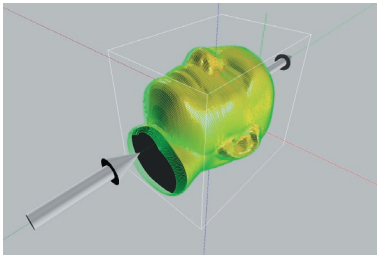
4th Axis Programmable Rotary Fixture

The Denford 4th axis fixture is designed for use with the Denford range of CNC Milling machines and Routers. It allows 360 degree rotation of material and comes complete with its own software - QuickCAM 4D Milling; an easy to use, wizard based CAM package that allows you to convert most 3D CAD files into 4th axis CNC program data.



QuickCAM 4D Milling CAM Software

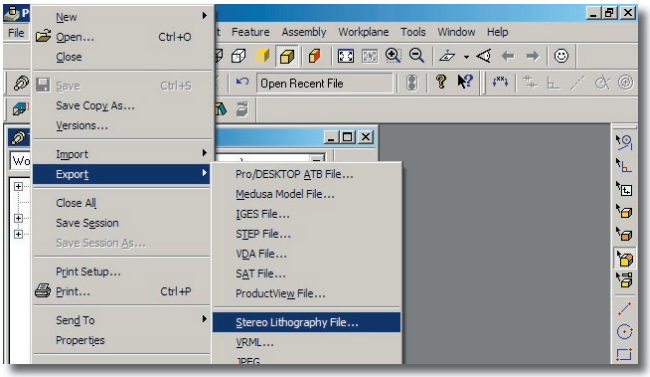
QuickCAM 4D is an easy to use, wizard based CAM package specifically designed for use with the Denford 4th axis programmable rotary fixture. QuickCAM 4D Milling imports 3D files from most 3D CAD packages and converts these into 4th axis CNC program data.



Stage 1 - Exporting your Design

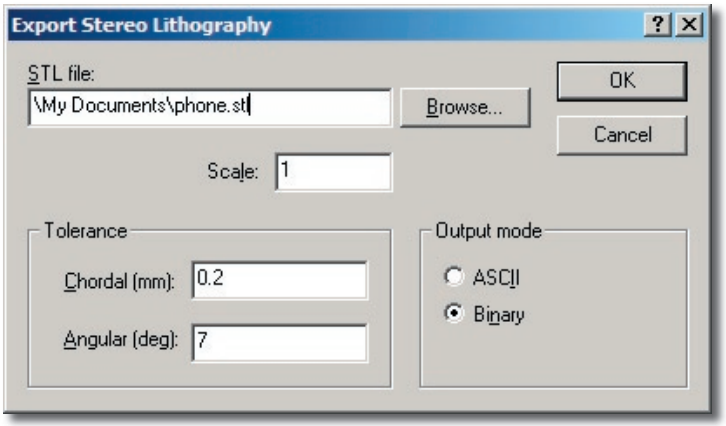
QuickCAM 4D will import STL files or 3DS (*.stl or *.3ds). Most 3D CAD modelling software can produce one of these file types, STL is the most common of the two. The process of producing the STL file will vary depending on the software used. The next section of this guide describes how to create an STL file from Pro/DESKTOP.

With your design loaded in Pro/DESKTOP, select 'File | Export | Stereo Lithography File...'.



A dialogue box will appear, you should click on browse so you can see which directory the file is being saved to. Remember the location where the file is saved.

Important : Choose the 'binary' option to keep the file size down.

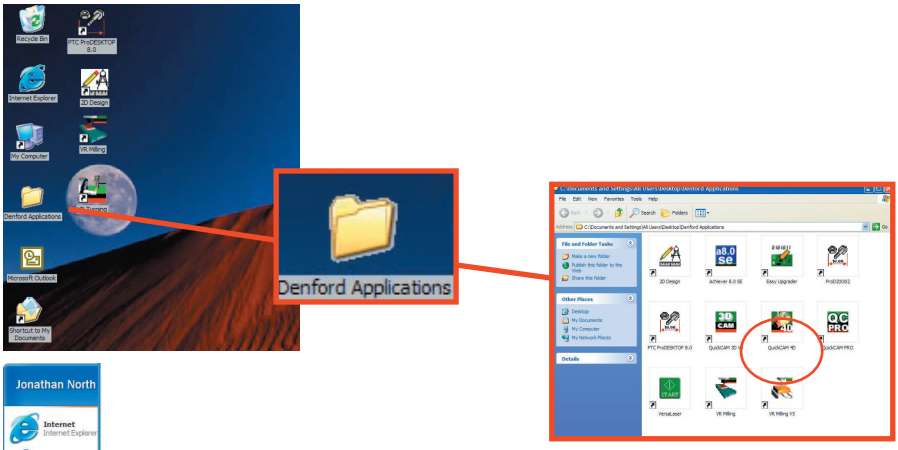


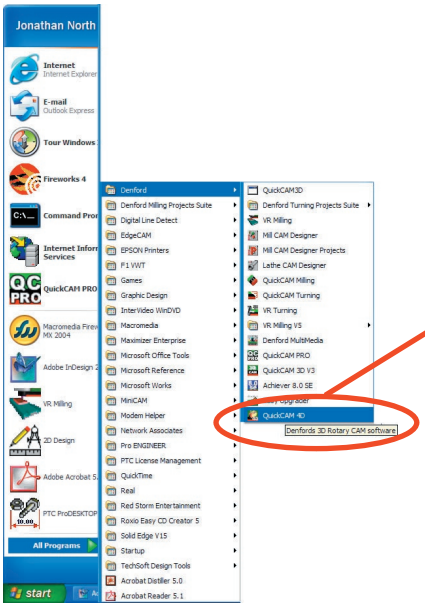
Stage 2 - Quickcam 4D software

The Denford QuickCAM 4D software is a simple to use, wizard based CAM package used to convert STL (*.stl) files into a CNC program that can be manufactured using a 4th axis enabled milling machine or router.

2.1 - Start the QuickCAM 4D software

To start the QuickCAM software open the 'Denford applications' folder on the desktop and double-click the “QuickCAM 4D” shortcut icon.



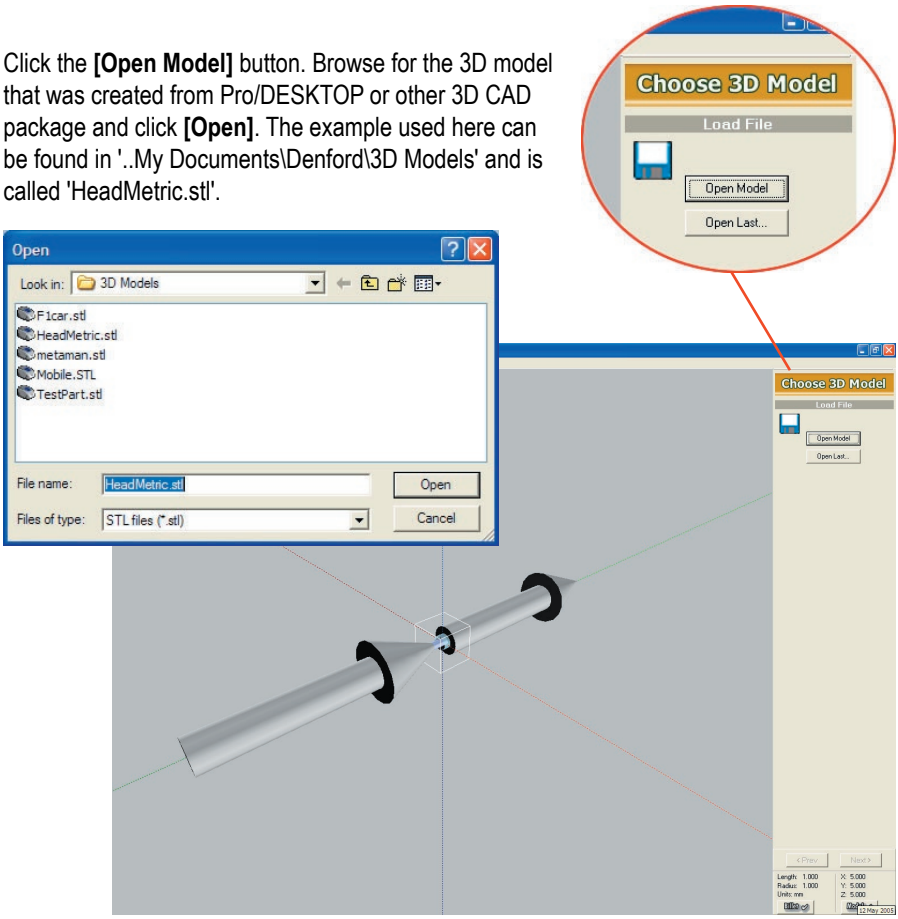


QuickCAM 4D menu icon

If the shortcut icon is not available, click “Start” on your “Windows” Startbar, followed by the “Programs” option, the program group “Denford” and finally the “QuickCAM 4D” icon.

2.2 - Load your STL file

Click the **[Open Model]** button. Browse for the 3D model that was created from Pro/DESKTOP or other 3D CAD package and click **[Open]**. The example used here can be found in '..My Documents\Denford\3D Models' and is called 'HeadMetric.stl'.



Manipulating the view

Click and hold the left mouse button whilst moving the mouse to rotate the view.

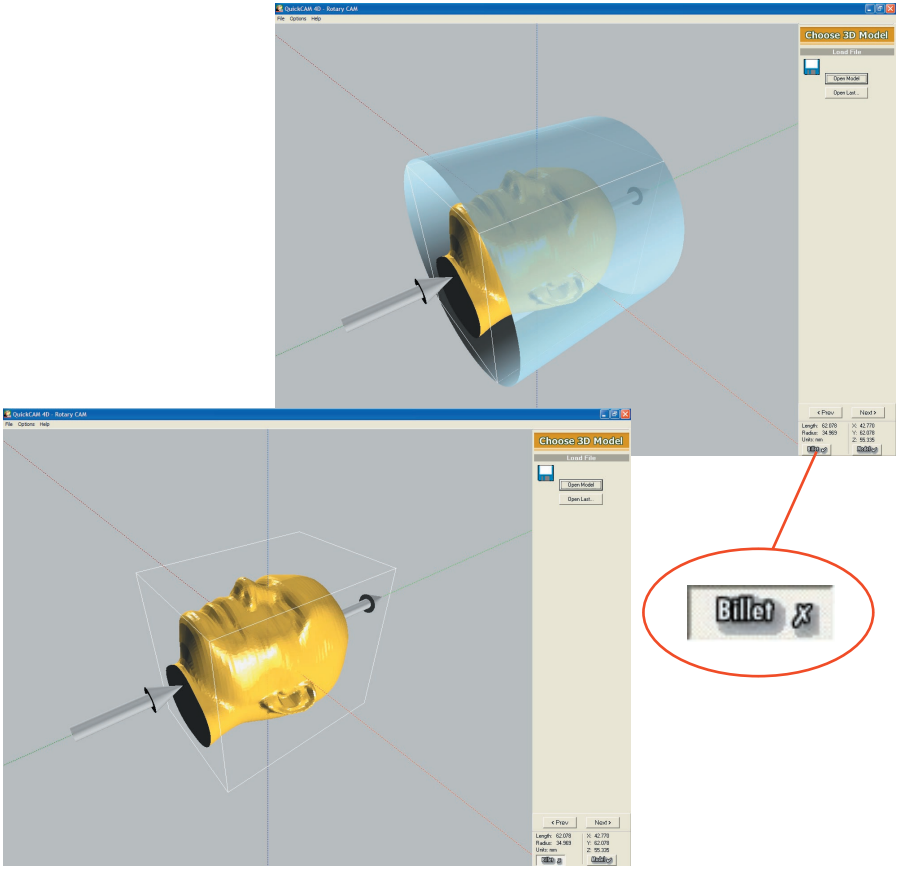
Click and hold the right mouse button whilst moving the mouse to zoom in or out.

If you have a wheel mouse, you can scroll the wheel to zoom in or out.

Hold both left and right mouse buttons and move the mouse to pan the view. (Move the object around the screen).

2.2 - Load your STL file

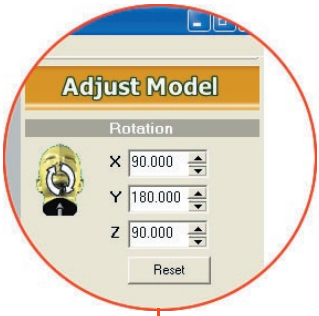
Once the model has been loaded into the software, in the bottom, right hand corner of the screen you will see the dimensions of the model displayed. The dimensions of a billet are also displayed. The billet has been sized automatically to fit the model. Both of these sizes can be altered further in the software.



Both the billet and the model can be hidden/unhidden to make viewing easier. This is done by clicking on the buttons in the bottom, right hand corner of the screen.

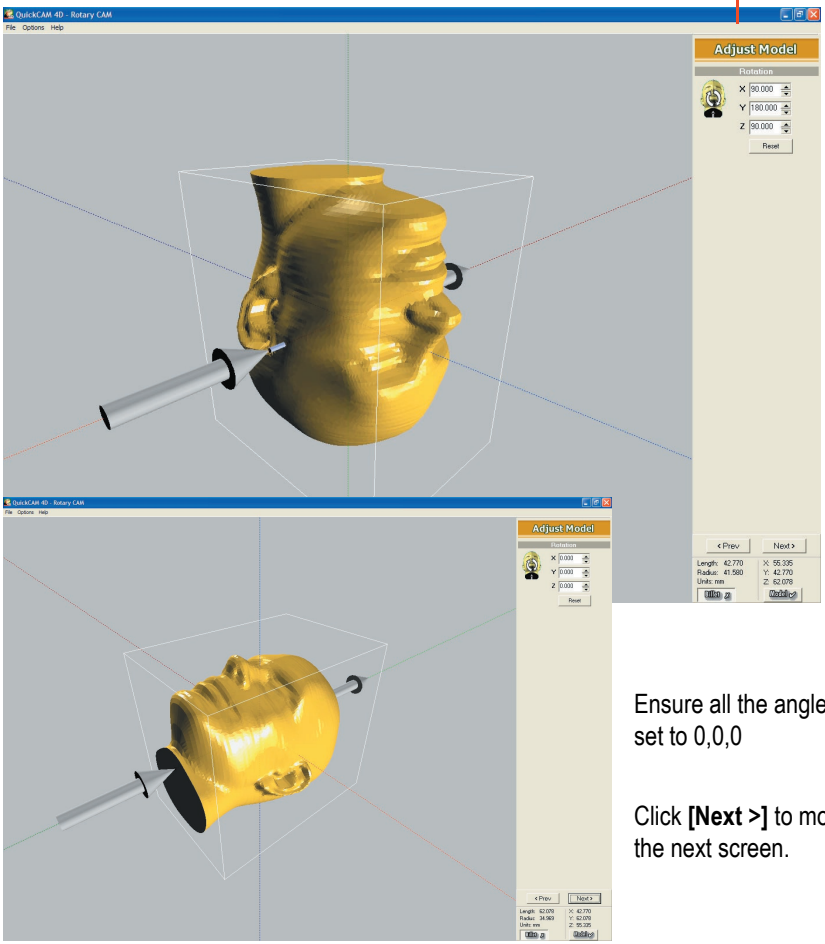
Click **[Next >]** to move to the next screen

2.3 - Adjust Model



The “Adjust Model” screen allows you to change the orientation of the 3D model around the axis of the rotary fixture. The model can be rotated around the axis by any angle. The best angle to set will depend on the model you are manufacturing.

In the image below the Head has been rotated by X = 90, Y = 180 and Z = 90. For this particular model this would not be the best angle. Set all the angles to 0,0,0



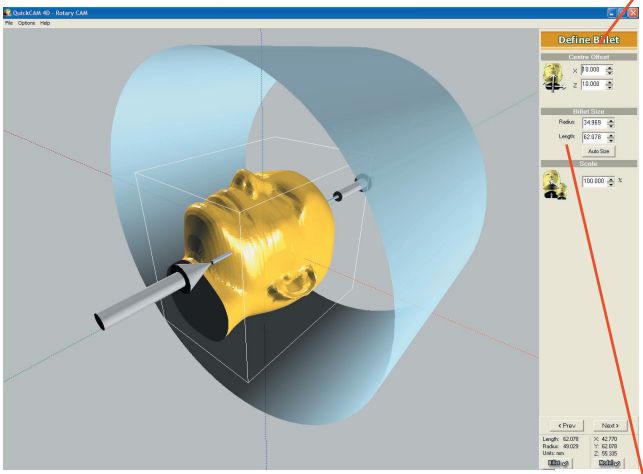
Ensure all the angles are set to 0,0,0

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

2.4 - Define Billet

The “Define Billet” stage allows you to scale the 3D model, define the billet size and to shift the centre offset.

Shifting the 'Centre Offset' allows you to change the point where the axis intersects the 3D model. Doing this will increase the radius of billet required, or require the model to be scaled down

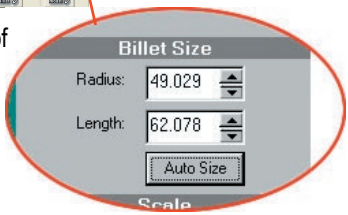


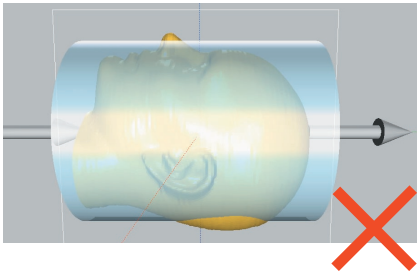
Click the [Auto Size] button in the 'Billet Size' section to create the billet size required to manufacture the part.

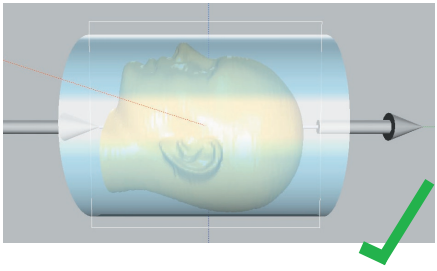
Alternatively, type in the dimensions of your actual billet and use the scale box to adjust the model to fit within your billet.

If adjusting the billet size or model scale ensure that all of the model will fit inside the billet. In the image below the nose and the back of the head would not be produced if manufactured.

It is not critical to have the cube which surrounds the 3D model within the billet, just the model itself.







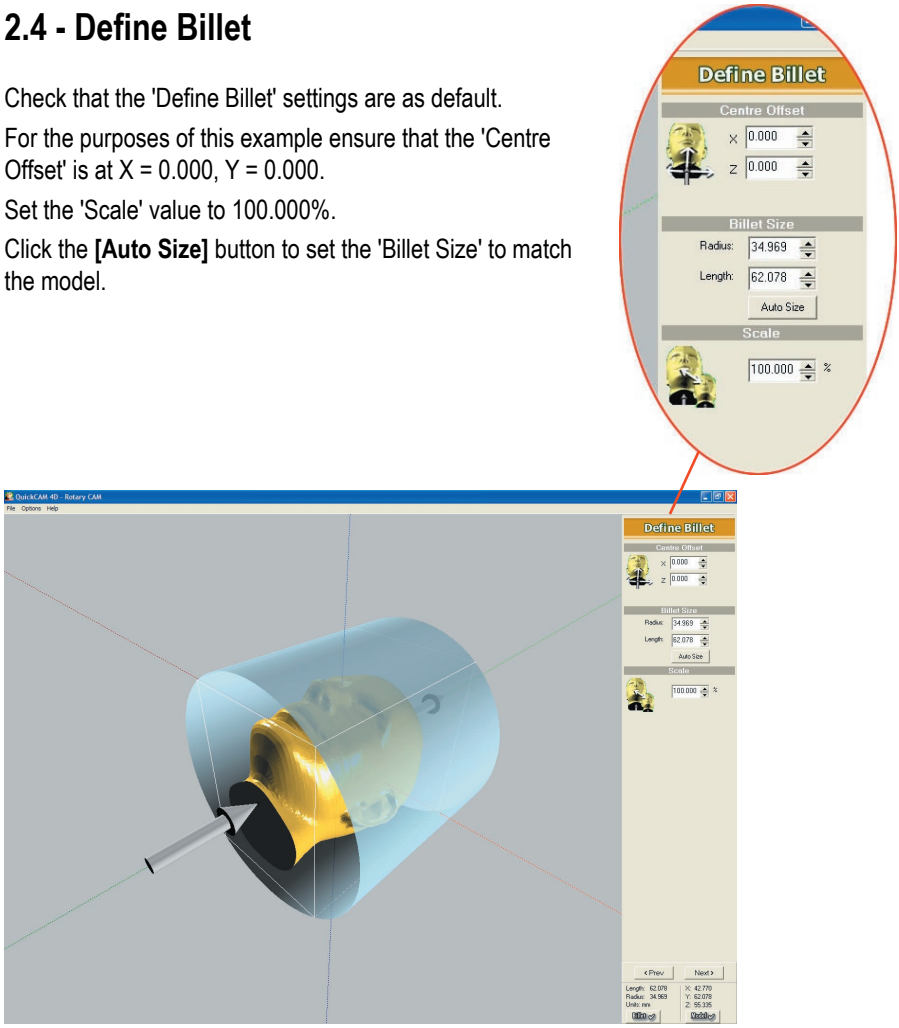
2.4 - Define Billet

Check that the 'Define Billet' settings are as default.

For the purposes of this example ensure that the 'Centre Offset' is at X = 0.000, Y = 0.000.

Set the 'Scale' value to 100.000%.

Click the **[Auto Size]** button to set the 'Billet Size' to match the model.



When we manufacture this part we will require a billet of at least 34.969mm Radius and 62.078mm Length.

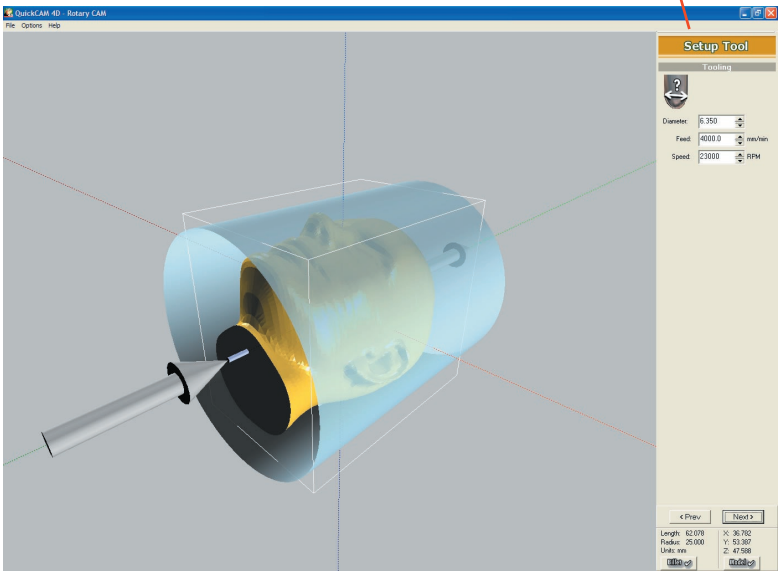
Click **[Next >]** to move to the next screen.

2.5 - Setup Tool

Enter the information about the tool you want to use to manufacture the part.

Enter the diameter of the tool you want to use, a smaller diameter will be able to produce a better finish and achieve more detail but will take longer to cut the part and may require roughing cuts to cut to the final depth.

Spindle speeds and Feed rates will depend on the material you are cutting and the tool you are using. See the Appendix for more information.



The values in the screen shot above are applicable to any Denford MIcroRouter using a 1/4" cutter, cutting Polyurethane foam. A Ball Nose cutter will produce the best results on these 3D models.

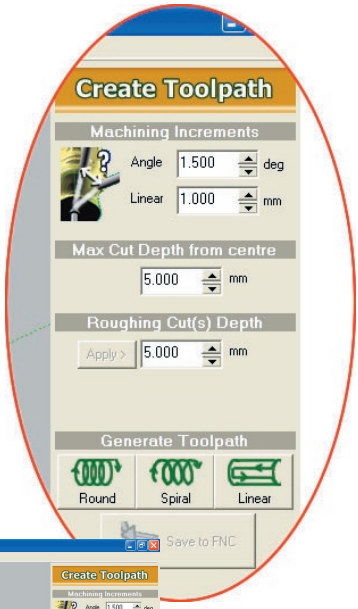
Click **[Next >]** to move to the next screen

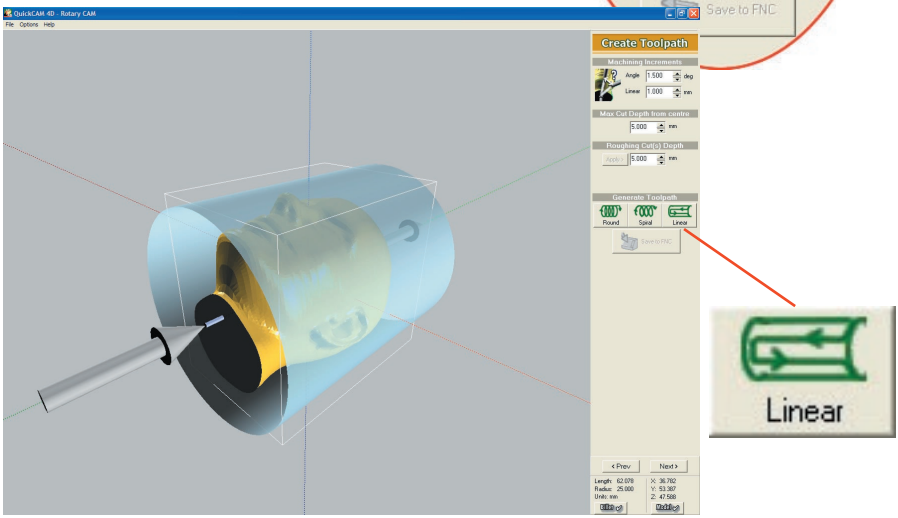
2.6 - Create Toolpath

The “Create Toolpath” stage allows you to set the values that will determine how the model will be machined.

The 'Machining Increments' will determine the resolution of the model and the quality of the finish, Always leave the 'Linear' setting at 1.000mm. It is recommended that the 'Angle' setting is between 1.000 degree and 3.000 degrees. 1 will give a finer finish and 3 a more course finish but in a faster time.

The 'Max Cut Depth from centre' will keep the cutter this distance away from the centre line of the rotary fixture. There needs to be a value here or the part will break off. The value inputted depends on the material, 5mm is the minimum recommended for foam.





There are 3 different 'Toolpath' strategies to choose from: Round, Spiral and Linear. You can experiment with the different strategies on different models but usually the 'Linear' strategy is the fastest, producing good results.

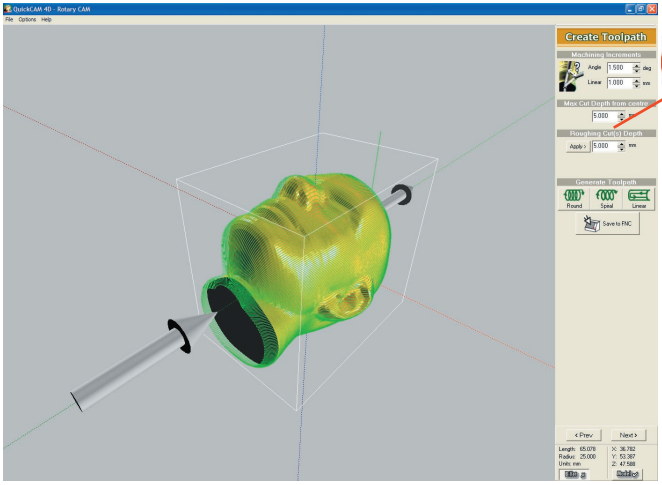
Click the 'Linear' button

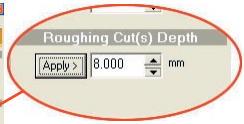
Pressing the Escape key [Esc] whilst the toolpaths are being calculated will cancel the command and enable you to make changes to the values.

2.6 - Create Toolpath

The Toolpath may take a few minutes to calculate, depending on the speed of the PC you are using. During this time you will see this message:

When complete you will see the toolpaths displayed in a green colour around the model.

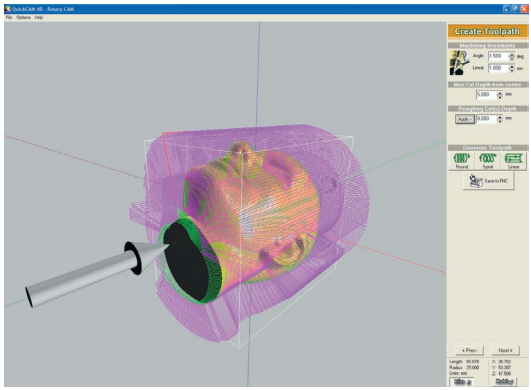




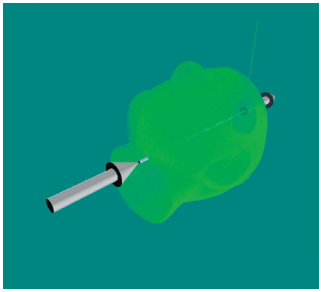
If required enter a roughing cut depth and click the [Apply >] button. Roughing cuts will be applied to your CNC file. The depth entered will depend on the material and tool that are to be used.

Roughing cuts will be displayed in purple.

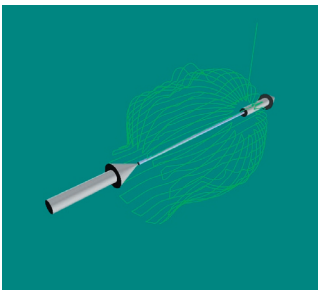
To make it easier to see the toolpaths you can turn off the model by clicking on the model button in the bottom, right hand corner of the screen.



The image below shows the toolpaths set at '1 degree Angle' and '1mm Linear', using a Linear strategy.



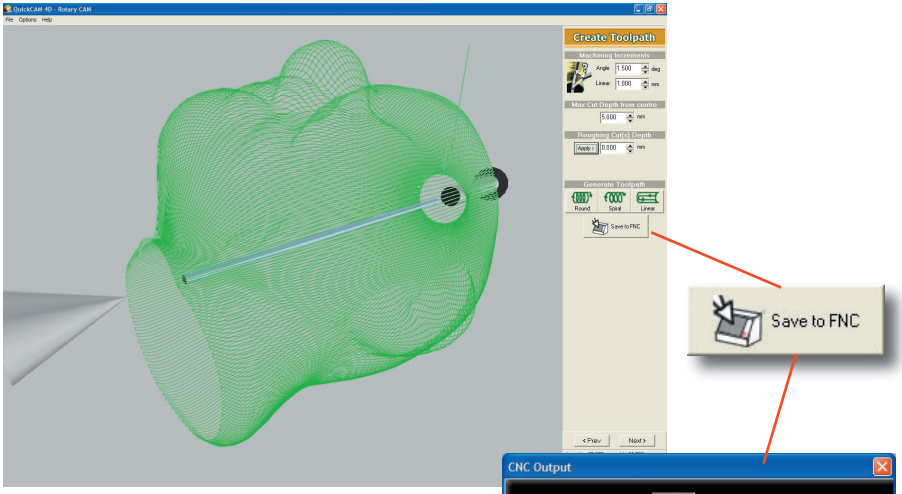
The image right, shows the toolpaths set at '10 degrees Angle' and '1mm Linear', using a Linear strategy.



2.6 - Create Toolpath

When you are happy with the toolpaths you have created, save the toolpaths.

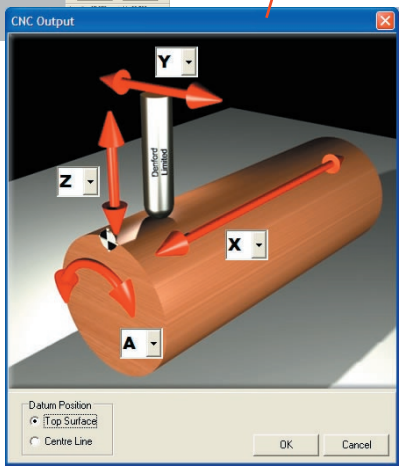
Click the [Save to FNC] button.

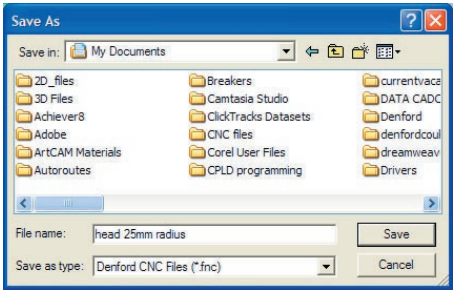


When the [Save to FNC] button has been clicked the 'CNC output' window will open. Here we can change the orientation of the axis, it is NOT recommended that this is done.

We can also select the 'Datum Position'. We can choose between 'Top Surface' and 'Centre Line'.

The Datum is the reference point for the program and we will set this at the CNC machine. Either option can be used but in this example we will select 'Top Surface'.





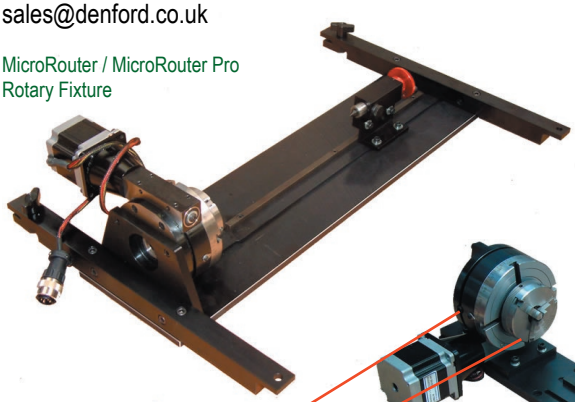
Click [OK] and the 'Save As' window will open. Type a description for the CNC file and click [OK]. Remember the location of the file. If you have Denford VR CNC Milling installed on the PC it will open automatically and load your CNC file ready for the next stage.

4th Axis Programmable Rotary Fixture

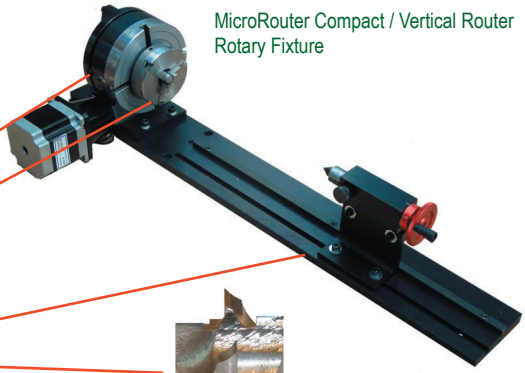
The Denford 4th axis fixture is designed for use with the Denford range of CNC Milling machines and Routers. It allows 360 degree rotation of material.

Unless the 4th Axis was specified when the CNC machine was purchased from Denford Ltd it will have to be retro-fitted as an upgrade to the machine. This upgrade must be carried out at Denford works. Contact Denford for more details on +44 (0)1484 728000 or email: sales@denford.co.uk

MicroRouter / MicroRouter Pro Rotary Fixture



MicroRouter Compact / Vertical Router Rotary Fixture



The 4th axis system comprises:

- Index drive Unit
- Spindle Chuck
- Tail Stock for component support
- Four pronged driver
- Clamping System to locate the unit in the machine
- Electrical Connections
- 4th Axis of Control to the machine CNC System
- QuickCAM 4D Milling (Single User Licence)

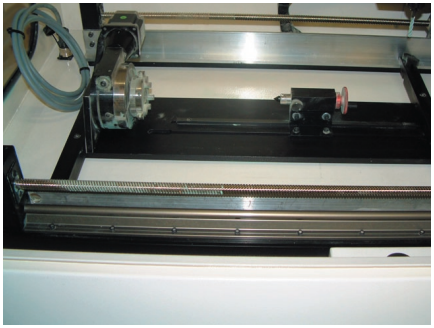
The Clamping System for the 4th axis system differs slightly depending upon which machine you are using. The MicroRouter or MicroRouter Pro CNC routers have the machine drilled and the 4th axis locates using dowels and a clamp. The MicroRouter Compact or Vertical Router use 'Tee-nuts' and bolts to clamp the 4th axis to the machine table.

Dowel & Clamp (Microrouter / Microrouter Pro)

'Tee-nut and bolt' (MicroRouter Compact / Vertical Router

Fit the 4th axis Rotary Fixture in the CNC machine

Locate the 4th axis fixture in the CNC machine, if using a Microrouter or Microrouter Pro locate the dowels in the pre-drilled holes inside the machine on the near side, then tighten the clamps on the far side of the machine. If using a Microrouter Compact or Vertical Router position the 4th axis fixture on the machine table. Move the Tee-nuts to the correct positions, aligned with the predrilled holes on the 4th axis fixture. Secure the 4th axis fixture to the machine table.



Microrouter / Microrouter Pro

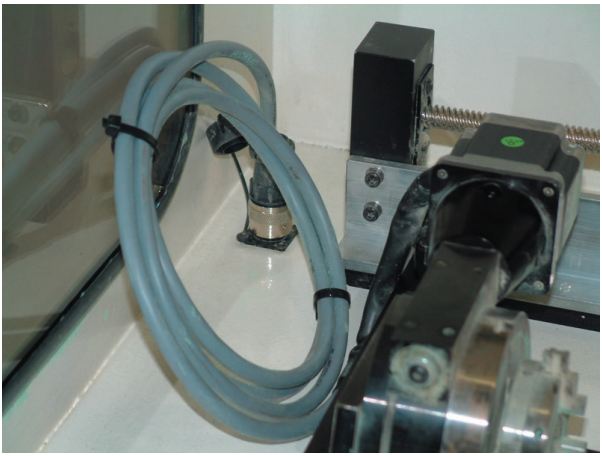
Microrouter Compact / Vertical Router

Make the electrical connection to the 4th axis fixture by plugging the '8 way QM' plug in to the socket found inside the machine. The socket is fitted with a cap to prevent ingress of debris when the 4th axis fixture is not in use. To remove the cap turn it anti-clockwise.

Fit the plug into the socket and secure it with the locking ring. To fit the plug in the socket match the rectangular locator on the plug with the recess on the socket.

Socket with cap fitted

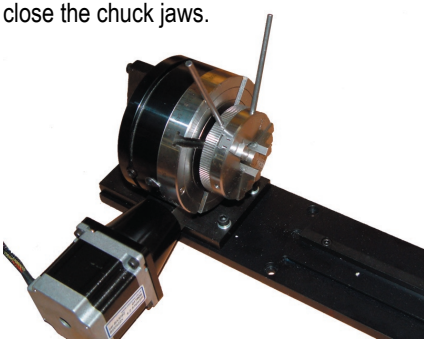
Socket with cap removed



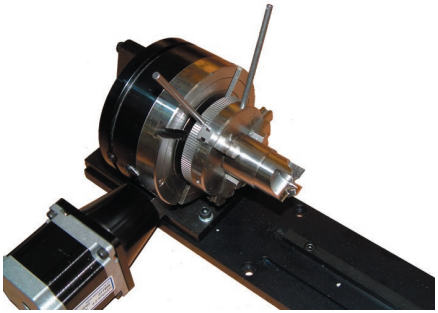
Plug and 4th axis rotary fixture fitted

Fit the billet in the 4th axis rotary fixture

The 4th axis rotary fixture is fitted with a '3 jaw spindle chuck'. This can be used to grip the billet, Alternativley the 'four pronged parrallel driver' can be fitted in the chuck to grip the billet. Two tommy bars are supplied with the 4th axis fixture which are used to open and close the chuck. It is the inner, grooved section of the chuck which is rotated to open and close the chuck jaws.



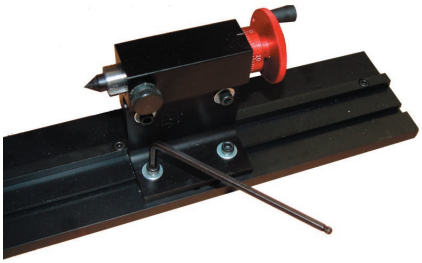
3 jaw spindle chuck with the 2 tommy bars used to open and close the chuck

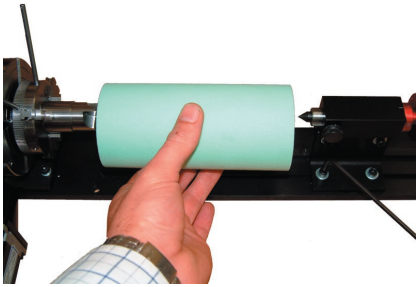


3 jaw spindle chuck with the 4 pronged parrallel driver fitted.

The 3 jaw spindle chuck will grip material up to 30mm in diameter with the chuck jaws in normal position. The chuck jaws are reversible, reversing them will enable a billet of up to 56mm in diameter. See the documentation supplied with the 4th axis programmable rotary fixture for detailed instruction on reversing the chuck jaws.

By using the 4 pronged driver to grip the billet, a diameter of up to 90mm on the Microrouter/ Microrouter Pro machines or 124mm on the Microrouter Compact/Vertical Router can be turned in the 4th axis rotary fixture.



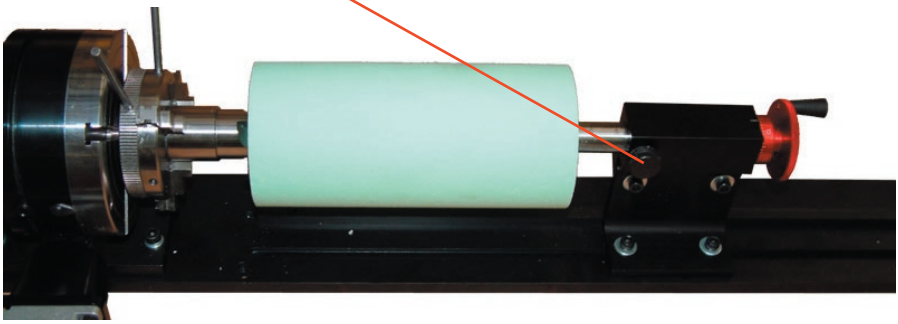


The 4th axis rotary fixture is fitted with an adjustable tailstock. The tailstock can be wound in and out using the red anodised wheel. By using a 4mm Allen key the tailstock can be repositioned along the length of the fixture. To fit the billet in the 4th axis fixture, reposition the tailstock so that the billet can fit between the tailstock and the four pronged driver.

Fit the billet in the 4th axis rotary fixture

Locate the centre of the billet with the centre of the four pronged driver and wind the tailstock into the centre of the billet at the opposite end. The teeth on the driver will bite into the billet and 'drive' the billet around, they need to have a good grip. If you are using a hard material it may be necessary to cut grooves to allow the four prongs to grip the material.

To prevent the tailstock from vibrating loose during the cutting process turn the locking wheel on the tailstock clockwise.

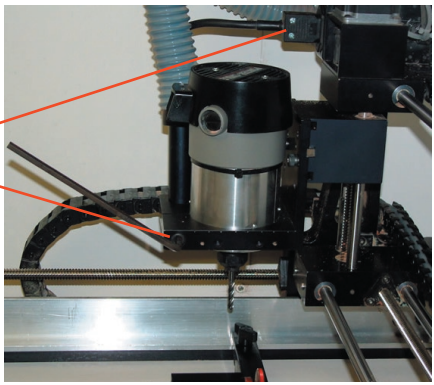


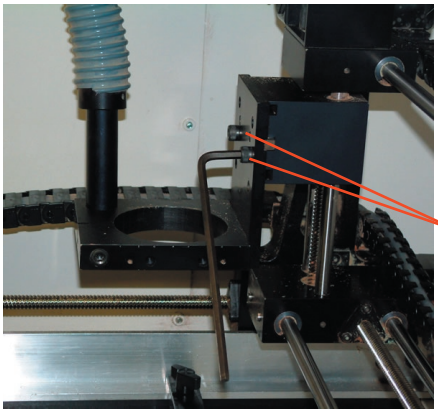
Raise the spindle motor on Microrouter/Microrouter Pro

The spindle motor on the Microrouter/Microrouter Pro may have to be raised to enable billets of larger diameters to be cut. This is not normally necessary on the Microrouter Compact or Vertical Router.

To raise the motor; firstly remove the motor from the 'gantry' in the machine by slackening the 6mm Allen bolt.

Lift the motor out and unplug it





With the motor removed from the gantry acces to the 2 x 6mm Allen bolts can be gained. Remove the 2 bolts and the gantry can be removed. Lift the gantry into the upper position. Refit the 2 x 6mm Allen bolts. Finally refit the motor.

DENFORD

CNC Milling/ Routing Training

www.denford.co.uk

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).

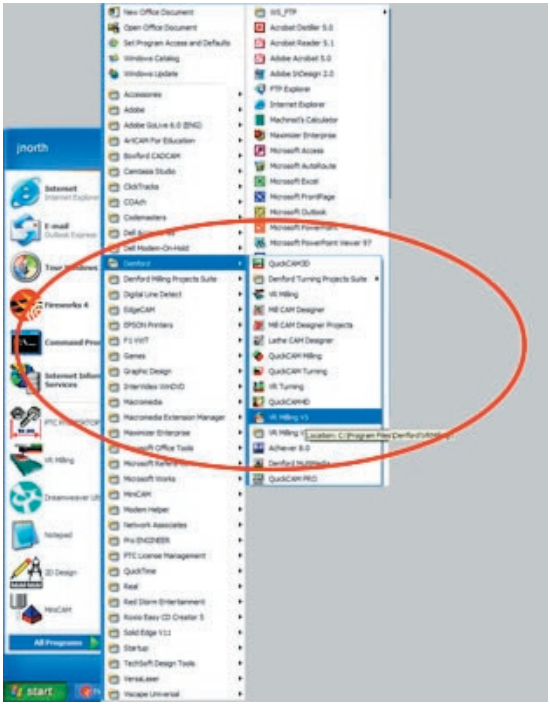
The VR Milling V5 software contains detailed help files including tutorials and animations. Access these by going to Help on the menu.

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.



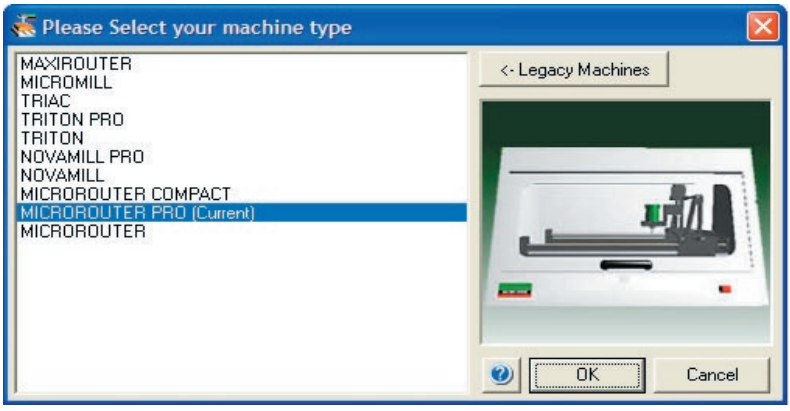
Step 2 - Configure the software for the machine

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.

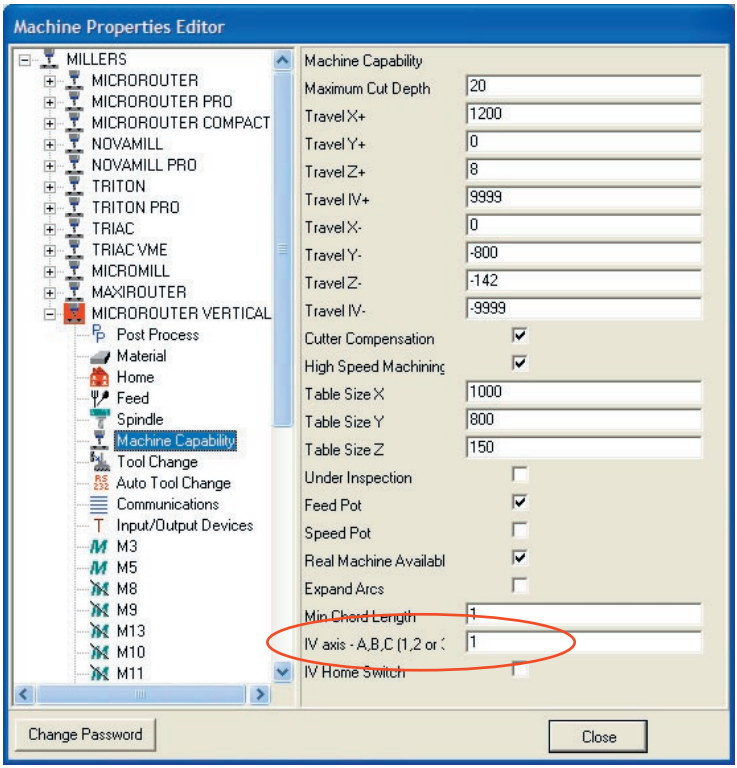
Step 3 - Enable the 4th axis.

Note: This step can only be performed if the software is **not** connected to the CNC machine

To reconfigure VR Milling V5 to enable the 4th axis you need to change the following options:

1. From the pulldown menu select "Setup" then "Setup Machine Parameters"
2. Enter the password. The default password is "denny".
3. The 'Machine Properties Editor' window will open. The current machine your software is configured for is highlighted red in the list.

   Click on 'Machine Capability' in the list
4. Type a 1 in the box next to 'IV axis - A,B,C(1,2 or 3)' to activate the 4th axis option as circled red below.
5. Click the [Close] button to exit.



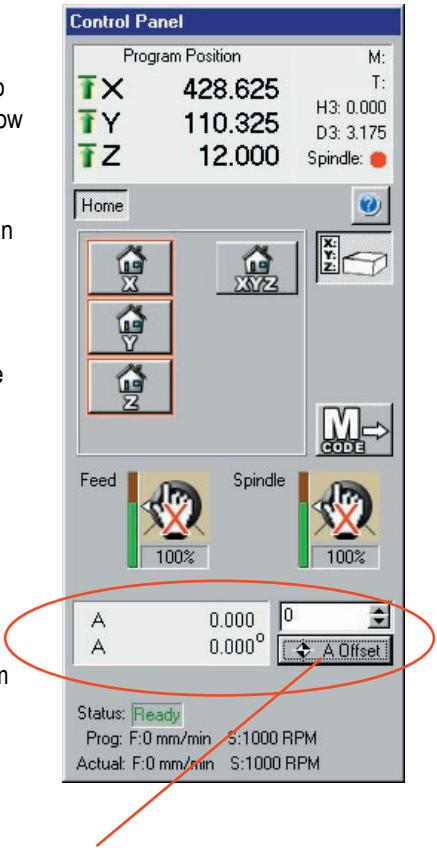
Step 3 - Enable the 4th axis.

Once VR Milling V5 has been reconfigured to enable the 4th axis, the 'Control Panel' will now show the 4th axis display.

It is possible to control the 4th axis from within the software.

Once the machine is homed the 4th axis can be rotated by using the **[.]** and **[/]** keys on the QWERTY keyboard when in jog mode.

The angle the 4th axis has rotated to is displayed in the 4th axis display at the bottom of the control panel.



A Offset button

It is possible to set the work offset for the A axis (Rotary axis). This is not normally necessary because the billet will be rotated through 360 degrees during the program so the angle from which it starts is irrelevant.

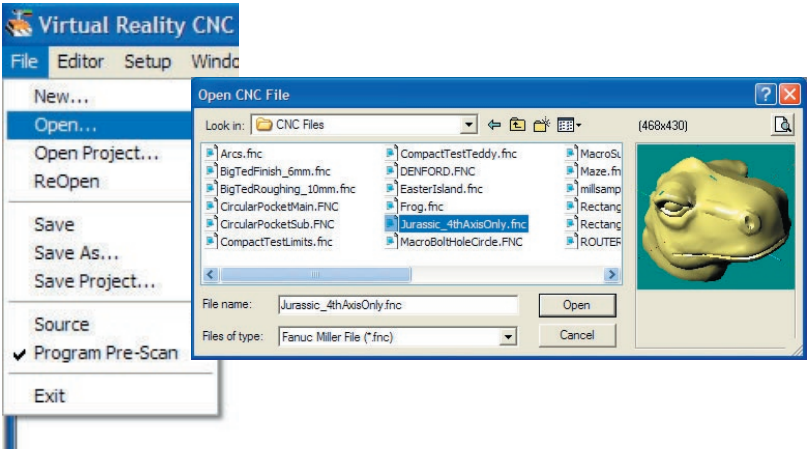
There may be times where you want to start from a specifc angle for example when using a rectangular billet. In cases like that jog the 4th axis fixture to the required angle and click the **[A Offset]** button to set the angle you would like to start from.

Note: When the machine is powered off the angle the 4th Axis is currently in becomes zero the next time the machine is powered on. This can be used to set the A offset. By jogging the 4th axis to the desired angle and then turning the machine off and back on the 4th axis position becomes zero meaning the A offset does not need to be set.

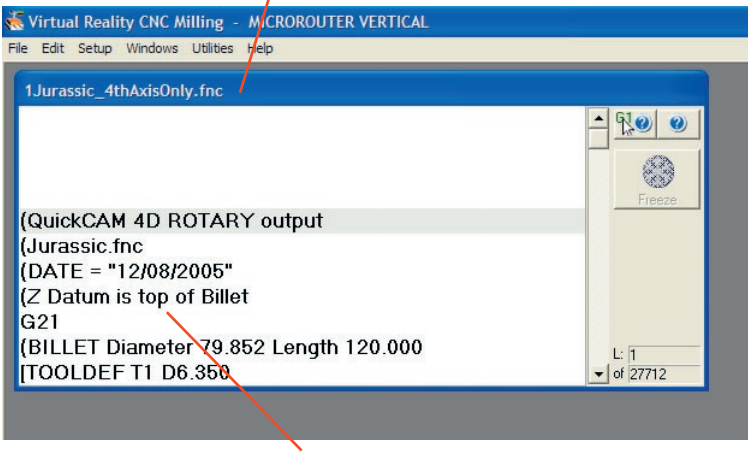
Step 4 - Load your CNC file.

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.



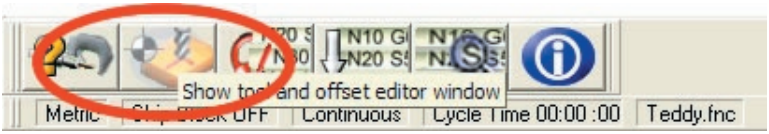
The contents of your CNC file will be displayed in the Editor window. As the name suggests, CNC files can be further edited here or you could even write one from scratch.



QuickCAM 4D puts a comment in the file regarding the Z datum position. This can serve as a useful reminder when setting the datum later.

Step 5 - 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.

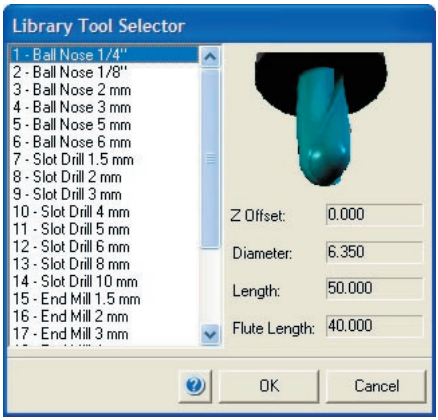
“Save tool to Library”

“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.

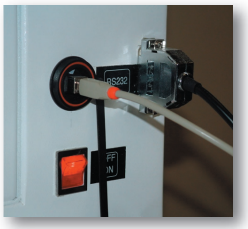
To add the ¼” Ball Nose cutter, highlight tool position 1, then click the “Insert Library Tool” button.

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



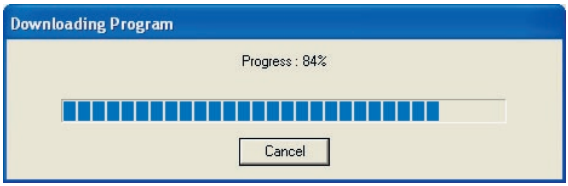
Click the [Tool and Offset Editor Window] button again to close the Window down.

Step 6 – Connect to, and Home the CNC machine



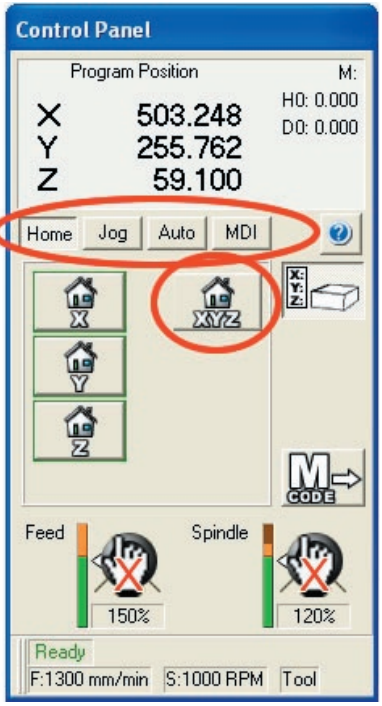
At this point ensure that the cable is connected from the PC to the Machine (either RS232 or USB) and that the machine is switched on.

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.



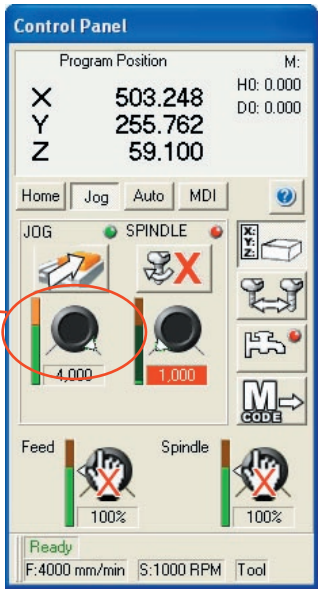
Step 7 - Move the machine head and fit the cutting tool.

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 single,straight arrow is displayed, signifying ‘Jog Continuous’ mode.

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.



"Jog Continous"

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.

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.

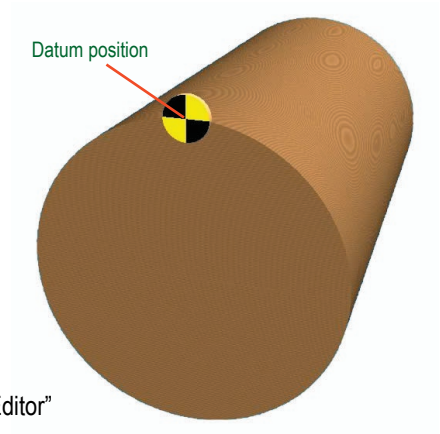
"Jog Step"

Step 8 – Set the work offsets

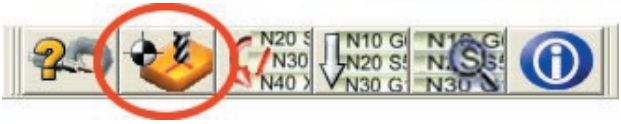
What are offsets?

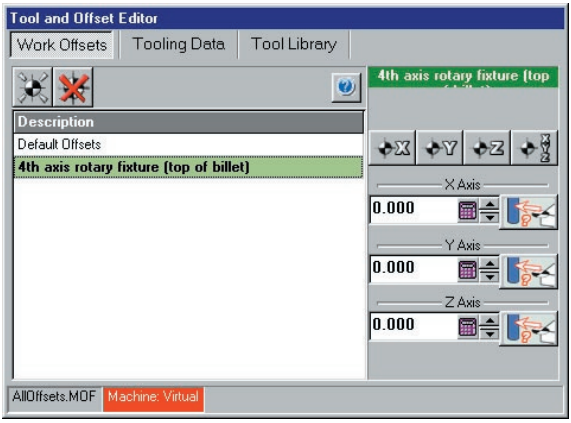
Offsets are the distances the cutter needs to travel, from its 'Home' position to the datum/reference point of the program in X Y & Z.

We are now going to define an offset for the top surface of the billet because that was the datum/reference point for the program created from 'QuickCAM 4D'



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.

"New work offsets"

•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.

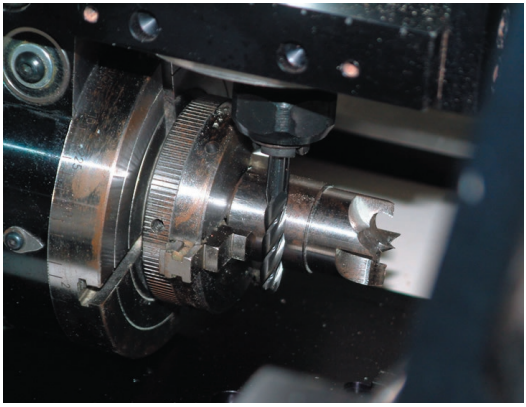
"activate offset"

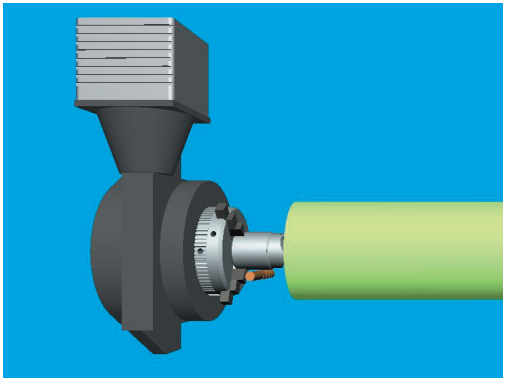
•Click the 'Activate' button to activate your new offset.

The active offset is highlighted in green.

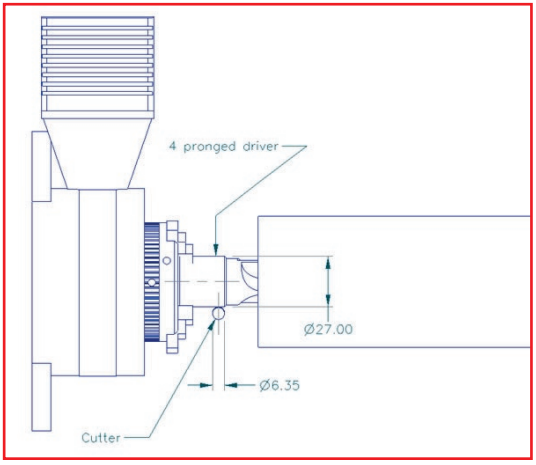
Step 8 - Set the work offsets - Y axis

In 'jog continuous mode', move the tool a few millimetres away from the four pronged driver. We are going to touch onto the four pronged driver at its largest diameter. The 4 pronged driver shown is 27mm at its largest diameter.
(It is advisable to measure the diameter of the 4 pronged driver at the point you intend to touch with the cutter.)

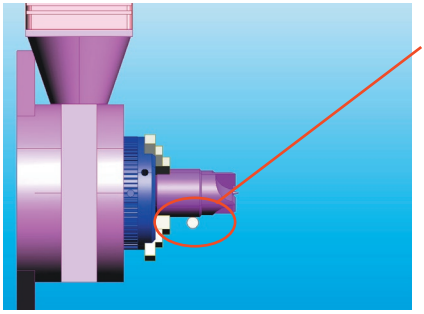




Now change to 'jog step mode' for fine incremental movements and position the cutter so that it just touches the four pronged driver. You can place a thin strip of paper between the cutter and the four pronged driver to detect precisely when the cutter touches.



Step 8 - Set the work offsets - Y axis

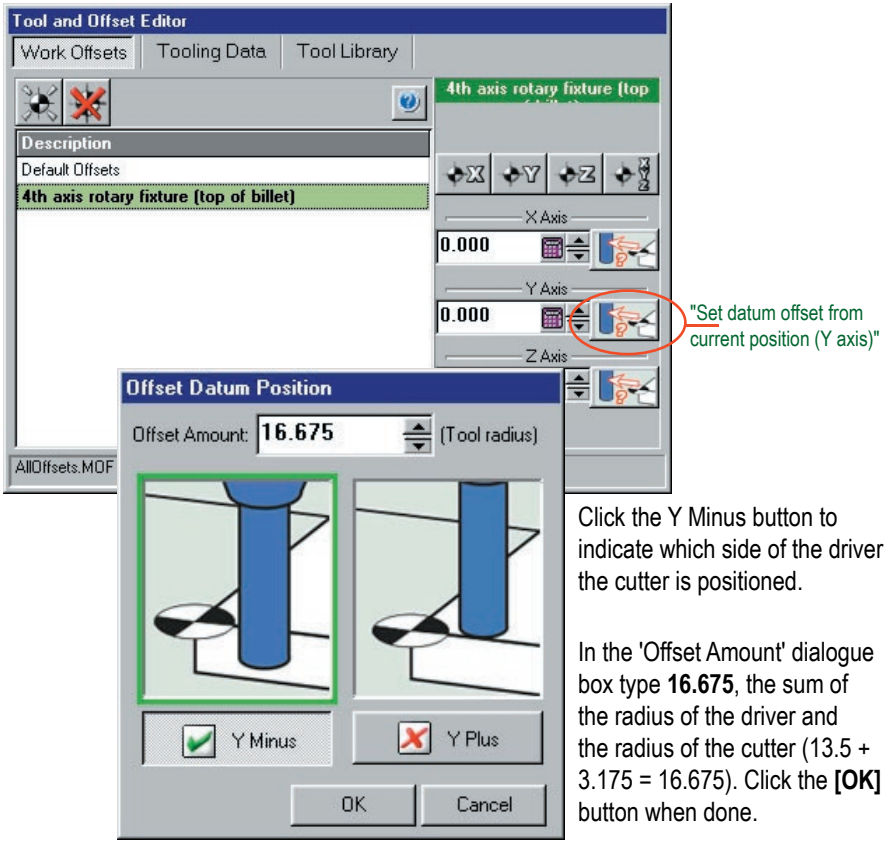


With the cutter in this position we will find the centre of the four pronged driver.

To find the centre of the driver we are going to include the sum of the radius of the driver plus the radius of the cutter.

The radius of the driver is 13.5mm and the radius of the cutter is 3.175mm. (13.5 + 3.175 = 16.675mm).

With the cutter in the above position click on the "Set datum offset from current position (Y axis)" button in the "Tool and Offset Editor" window.



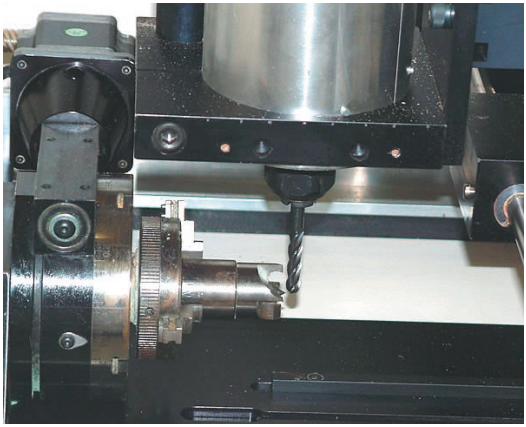
Click the Y Minus button to indicate which side of the driver the cutter is positioned.

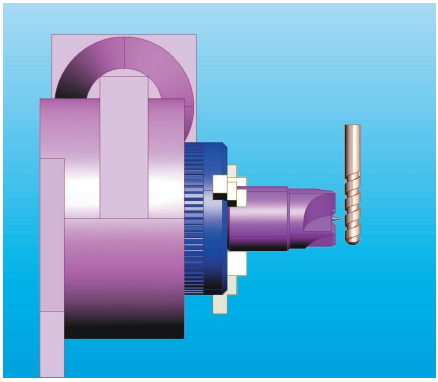
In the 'Offset Amount' dialogue box type **16.675**, the sum of the radius of the driver and the radius of the cutter (13.5 + 3.175 = 16.675). Click the **[OK]** button when done.

Step 8 - Set the work offsets - X axis

In 'jog continuous mode', move the tool a few millimetres away from the end of the four pronged driver.

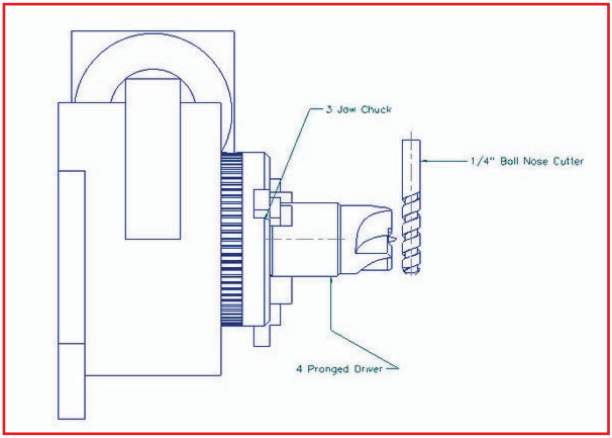
Setting the X axis work offset/datum in this position will ensure that the cutter will not hit the four pronged driver.



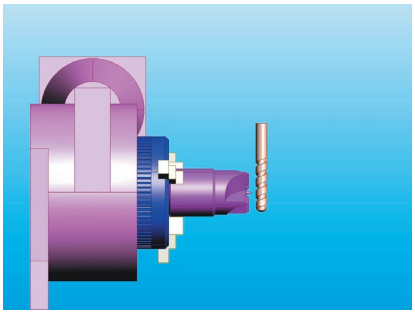


The X axis work offset/datum is best set with the cutter not quite touching the four pronged driver. Setting the X axis work offset here will produce the design as close as possible to the end of the billet.

If you have a design that you would like to machine further along the billet move the cutter further away from the four pronged driver.

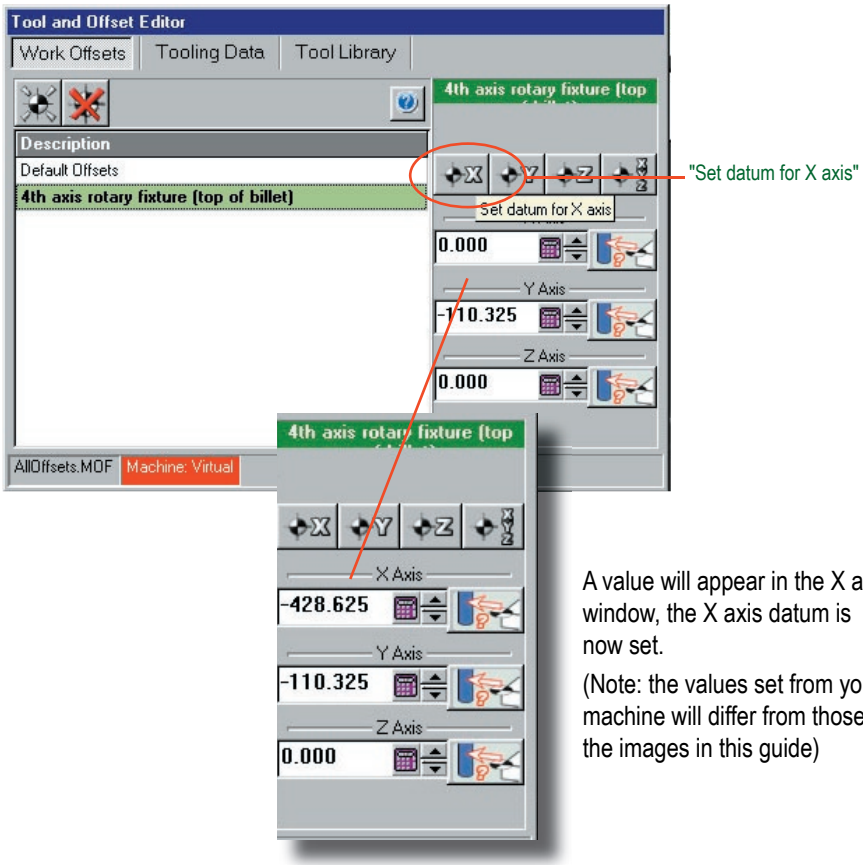


Step 8 - Set the work offsets - X axis



With the cutter in this position we will set the X axis datum as the centre of the cutter.

With the cutter in the above position click on the "Set datum for X axis" button in the "Tool and Offset Editor" window.

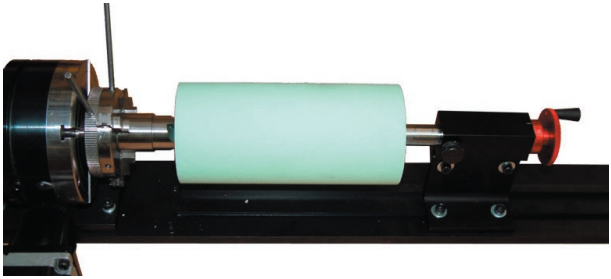


A value will appear in the X axis window, the X axis datum is now set.

(Note: the values set from your machine will differ from those in the images in this guide)

Step 8 - Set the work offsets - Z axis

Home all axes to enable the billet to be fitted in the 4th axis rotary fixture as described earlier in this guide.

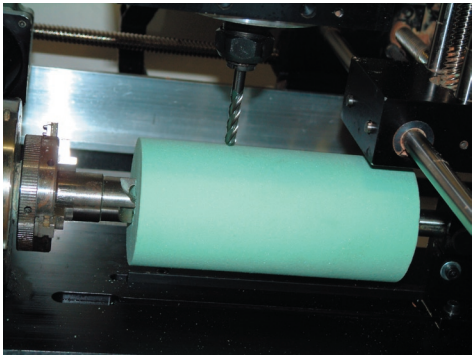


You could use the jog keys to position the cutter over the centre of the billet to enable the setting of the Z work offset, however there is a more accurate method described below:

Click on the MDI tab in the control panel and type Y0 *(zero not the letter O)*

Press Play on the file control.

The cutter will move to the Y position set previously. This should be directly in line with the centre of the billet, if not the Y offset is incorrect, repeat the previous operation.

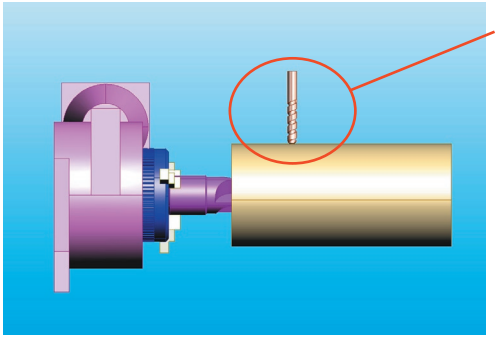


Jog the X axis until the cutter is over the centre of the billet.

Jog down and touch the top surface of the billet.

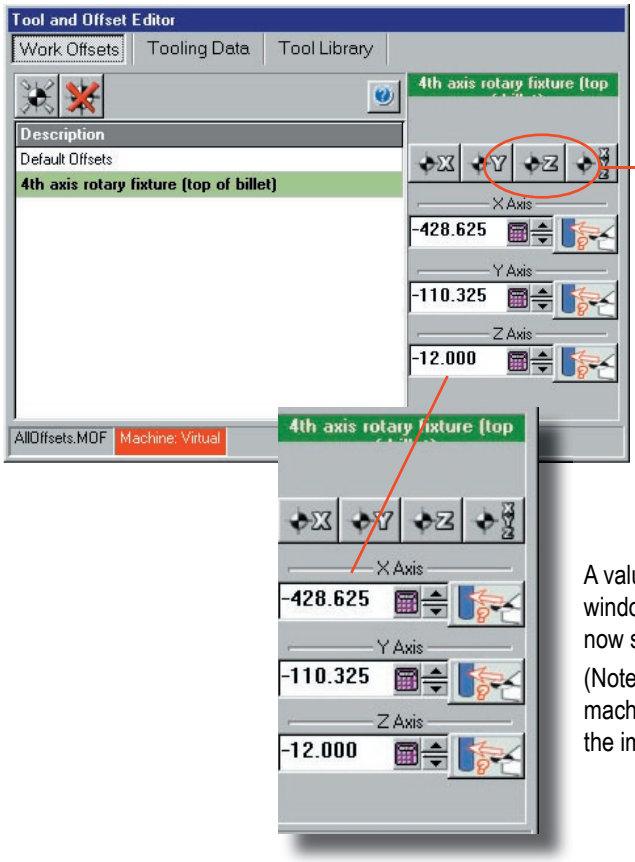
Use incremental jog if necessary to find the surface of the billet accurately.

Step 8 - Set the work offsets - Z axis



With the cutter in this position we will set the Z axis datum as the tip of the cutter.

With the cutter in the above position click on the "Set datum for Z axis" button in the "Tool and Offset Editor" window.



A value will appear in the Z axis window, the Z axis datum is now set.

(Note: the values set from your machine will differ from those in the images in this guide)

Step 8 - Set the work offsets

The datum point for the program is now set. We have stored 3 values for X,Y & Z. These values are the reference point for the program, the point from which all the coordinates within the program are taken.

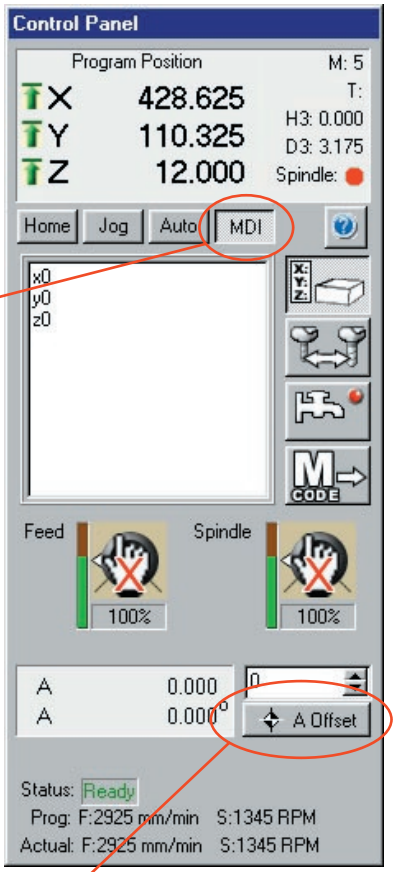
We can verify the work offsets before we run the program.

Click on the MDI tab in the control panel and type:

x0 *(zero not the letter O)*

y0 *(zero not the letter O)*

z0 *(zero not the letter O)*



Press Play on the file control.

The cutter will move to the X,Y & Z positions set previously. This should be directly in line with the centre of the billet, touching the surface of the billet.

Setting the A axis work offset

It is possible to set the work offset for the A axis (Rotary axis). This is not normally necessary because the billet will be rotated through 360 degrees during the program so the angle from which it starts is irrelevant.

There may be times where you want to start from a specifc angle for example when using a rectangular billet. In cases like that jog the 4th axis fixture to the required angle and click the **[A Offset]** button to set the angle you would like to start from.

Note: When the machine is powered off the angle the 4th Axis is currently in becomes zero the next time the machine is powered on. This can be used to set the A offset. By jogging the 4th axis to the desired angle and then turning the machine off and back on the 4th axis position becomes zero meaning the A offset does not need to be set.

Step 9 – Run the Program.

Before running the program 'Home' all the axis to ensure the cutter is clear of the billet.

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 over ride and Spindle Speed over ride 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/4D CAD/CAM software are run with turbo mode on.

Cylindrical Wrapping of a 2D/3D program

It is possible to 'wrap' a normal 2D/3D program around a cylinder. This could be a program created in "Techsoft 2D Design Tools" or another 2D CAD package. For example instead of engraving some text into the surface of a rectangular piece of material you could engrave around a cylindrical piece of material. The same applies to programs created from either a ProDesktop or ArtCAM design via MiniCAM or QuickCAM 3D/Pro.

Cylindrical wrapping of a program is achieved by editing the program to include 2 extra lines of information. The first line of code to add to the program asks the cutter to move over the centre of the billet. The second line of code to add asks the machine to translate all susequent moves in the Y axis into rotational movements for the 4th axis.

The two lines to add are: **G90 G0 Y0**

G107 Y-30

The first line **G90 G0 Y0** asks the machine to make a rapid move to Y0 (the position where the Y work offset was set, over the centre of the billet)

The second line **G107 Y-30** asks the machine to turn on cylindrical wrapping (G107) in the Y axis. **-30** indicates the radius of the billet (the number is negative so that the 4th axis turns the billet counter clockwise, this means the design will be produced as it is seen on the screen, a positive value may result in the design being produced as a mirror image.)

```
G21
[BILLET X150.000 Y172.000 Z30.000
[EDGEMOVE X0 Y0
[TOOLDEF T1 D6
(MATERIAL Modelling Foam
G91 G28 X0 Y0 Z0 M05
G90 M6 T1
M03 S18000
G0Y0
G107 Y-30
G0 X0.001 Y0.001 Z5.000
G1 Z-4.675 F2000.0
G1 X0.261 F4000.0
```

To the left is a sample program created by MiniCAM from an ArtCAM CAD model.

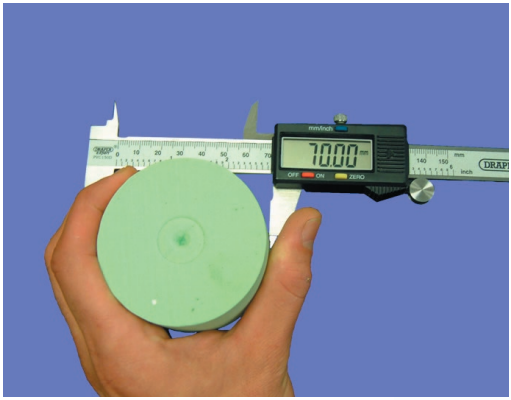
The program has been edited to include the lines necessary to wrap the program around a cylinder.

Please note: it is only recommended that this process is used with a cylindrical billet and not a rectangular billet.

Cylindrical Wrapping example using Techsoft 2D Design Tools

The following example shows how to create a simple program using Techsoft 2D Design Tools and how to 'wrap' it around a cylindrical billet using the 4th axis programmable fixture. Some prior knowledge of Techsoft 2D Design Tools is assumed and the example is not intended to be a comprehensive guide to Techsoft. New users may wish to refer to the Denford 'Techsoft 2D Design Tools training guide'.

Prepare the CNC file.



Measure the billet to find the radius. It may be easier to measure the diameter and divide by 2.
70mm/2 = 35mm radius

The next step is to establish the depth of cut.

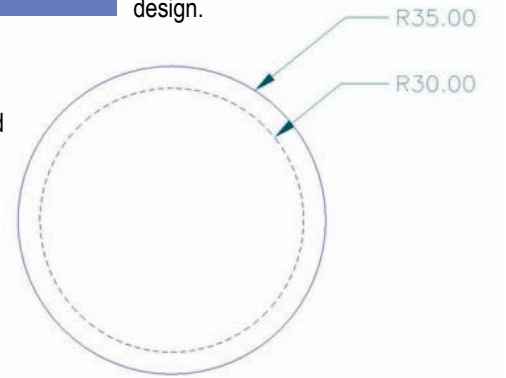
If using ArtCAM or ProDesktop the depth of cut would be dictated by the settings in the CAM software.

If using Techsoft you would set the depth of cut by the colour in your design.

In this example the design will be cut to a depth of 5mm.

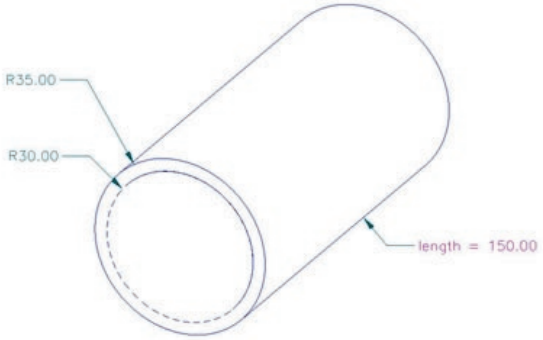
The CNC file will not be wrapped around the outer circumference of the billet, but wrapped around the circumference of the billet minus the depth of cut.

If the radius of the billet is 35mm take away the value of the deepest cut in the program to find the inner radius. In this example that is 35mm - 5mm = 30mm



The next step is to ensure that the program will fit around the inner circumference. To calculate the circumference (πr^2 or πd).
In this example where r = 30 the circumference is 188.5mm

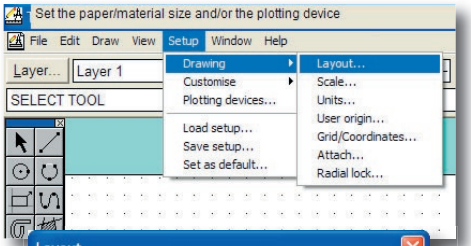
188.5mm will serve as our Y axis value when creating the Techsoft design, to find the X axis value measure the length of the billet, in this example 150mm.



Techsoft 2D Design Tools

Start the Techsoft 2D Design Tools software.

Select 'Setup | Drawing | Layout'

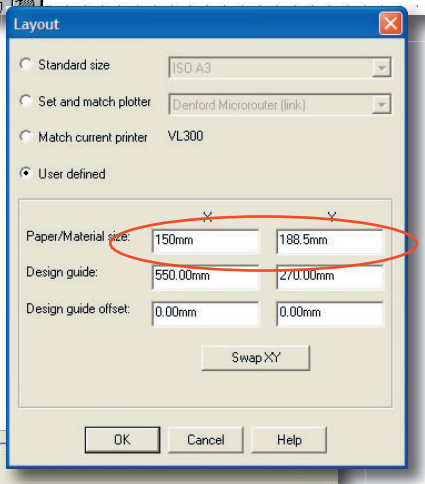


Click the 'User Defined' radio button.

In the 'Paper/Material' size box, enter the values for the X and Y dimensions of your workpiece. (Y being the circumference value).

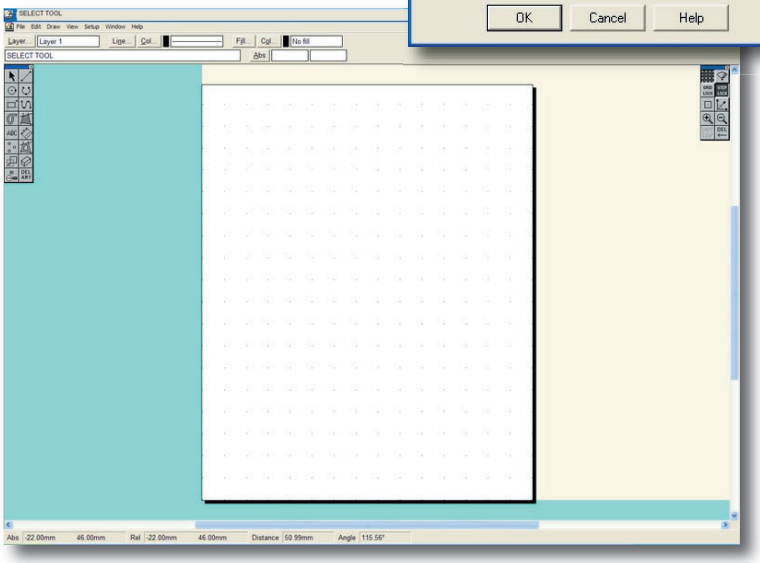
X = 150mm Y = 188.5

Click [OK]

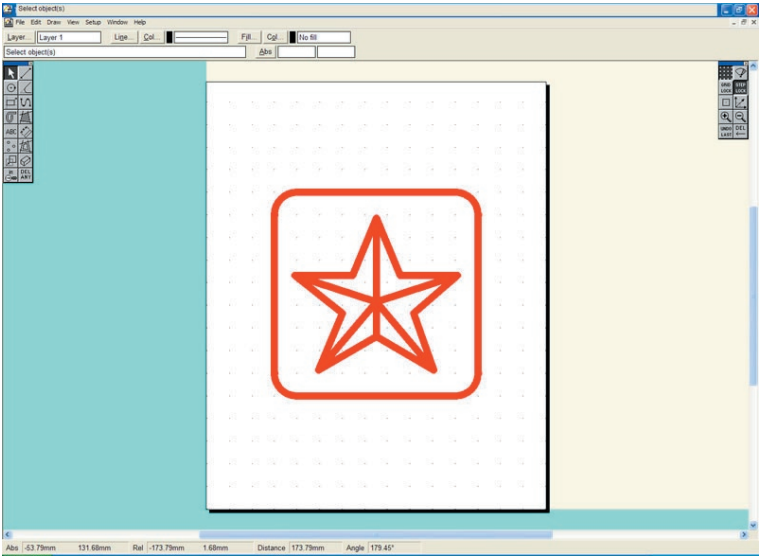


There will now be a white 'design area' on the screen which is the dimensions of the length and circumference of the billet.

As long as the design remains within this white area it will be able to be wrapped around our billet.

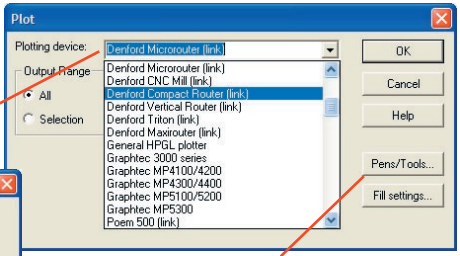


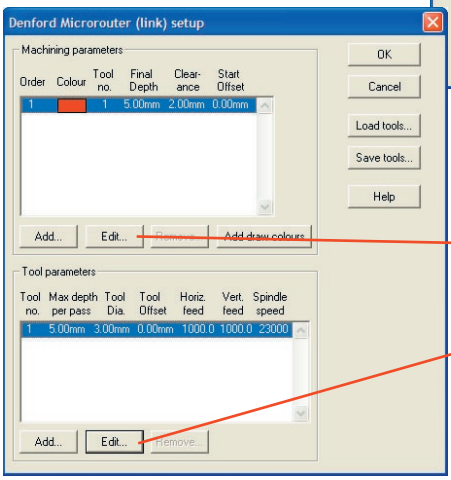
Create your design, in this example a simple design has been created, the line thickness has been changed to 3mm to represent the width of the cutter used and the line colour has been changed to red. The design does not go all the way to the top and bottom of the white design area, but if it did, it would wrap all the way around the circumference of the cylinder.



Create the CNC file by selecting 'File | Plot'

Ensure that you have the correct machine selected from the drop down menu.

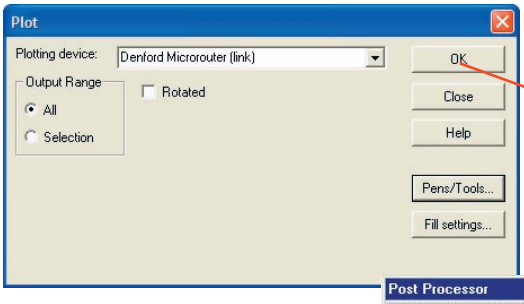




Click the [Pens/Tools..] button to define the depth of cut and order of manufacture for each colour used.

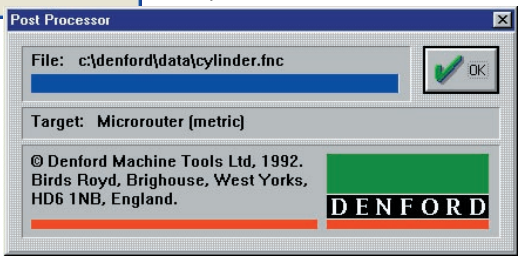
Set the depth of cut for red (5mm in this example)

Don't forget to set the Tool Parameters (in this example the 'Max depth per pass' is set as 5mm, the feedrate is 1000mm/min and the spindle speed is 23000rpm)



Click [OK] to the dialogue boxes until you are back to the 'Plot' dialogue box. When you click [OK] here the CNC file will be written.

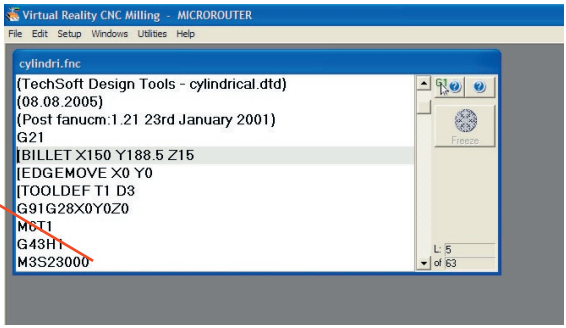
The CNC file is given the same name as the Techsoft file and by default is sent to the C:\ drive of the computer.



The default path to the file is:
C:\denford\data*filename*.fnc
Note: some establishments may have changed this path.

Edit the CNC program

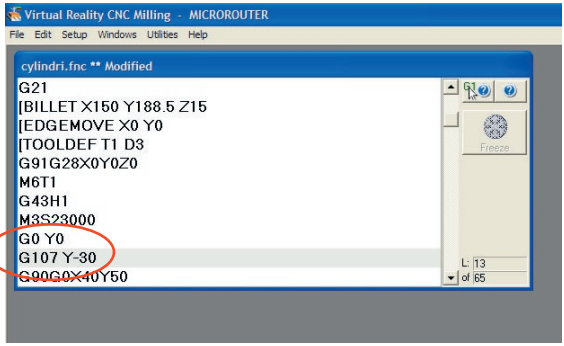
Now the CNC program has been created it needs to be edited to enable the 4th axis movement. Start the VR Milling V5 software and open the CNC program that has just been created from Techsoft 2D Design. Put the cursor after the Spindle Speed line (M03S23000) and press [Enter].



Type: **G90 G0 Y0**
G107 Y-30

Cylindrical wrapping in the Y axis has now been activated in the program.

Refer to the VR Milling V5 section of this guide for instruction on setting up the machine and manufacturing the component.



Pictured below are some products created using G107 cylindrical wrapping. At the front is the Techsoft example used in this guide. The other 3 products were created from ArtCAM using the same technique, they are 3D masks and a 3D relief of the UK.
The cylindrical billets used here are available from Denford Ltd, contact the sales department for more information.

Contact Details (Denford):
Denford Limited,
Birds Royd,
Brighouse,
West Yorkshire,
HD6 1NB,
ENGLAND.

Telephone: 01484 728000
Fax: 01484 728100
E-mail: info@denford.co.uk
Technical Support: Monday to Friday 8.30am - 4.30pm GMT
For international dialling from outside the UK: Add “44” to the number and remove the first “0” from each city code.