4th Axis F1 TEAM IN SCHOOLS CAR MANUFACTURING GUIDE

D-Type

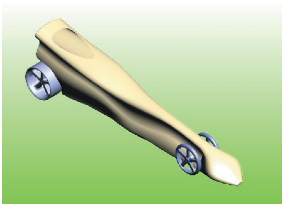
Pro/DESKTOP 8

▼

QuickCAM 3D or QuickCAM Pro

▼

4th Axis Rotary Fixture & VR CNC Milling V5



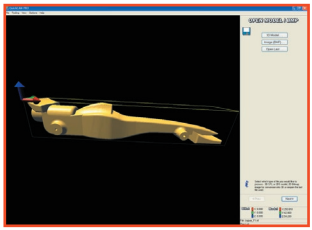
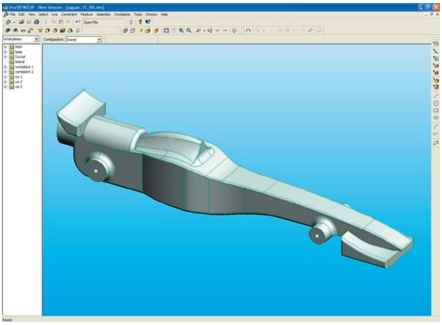


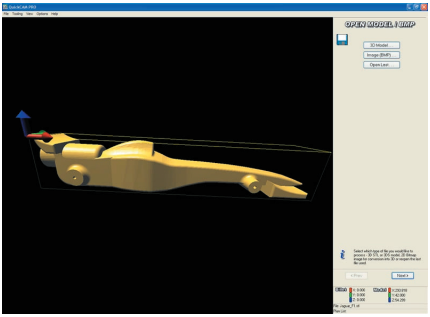
Table of contents

Introduction

This tutorial guide leads you through the process of converting a Pro/DESKTOP 3D Model of an F1 in Schools Car into a CNC file and manufacturing it using the 4th axis rotary fixture.

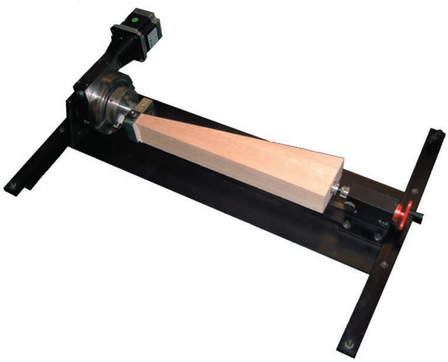


Pro/DESKTOP 8



QuickCAM 3D or QuickCAM Pro

Denford VR CNC Milling V5 & 4th axis rotary fixture

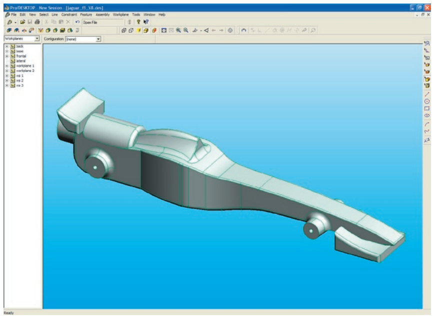


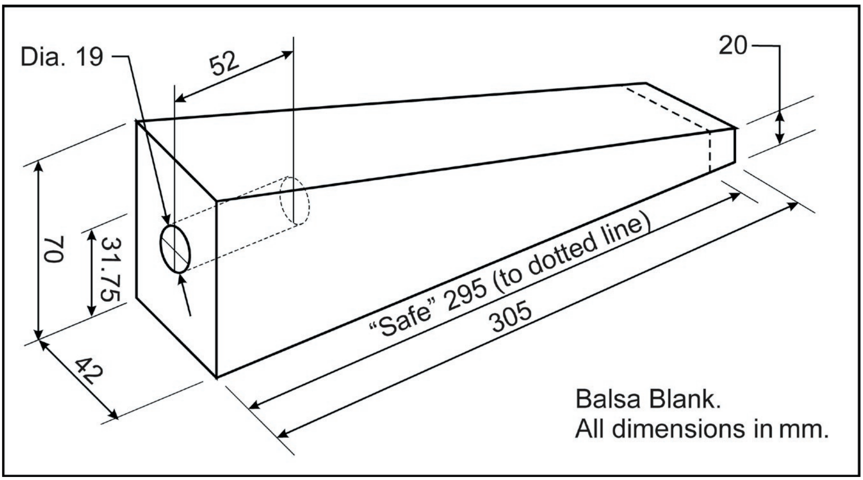
Stage 1 - Using Pro/DESKTOP

1.1 - Create your car design

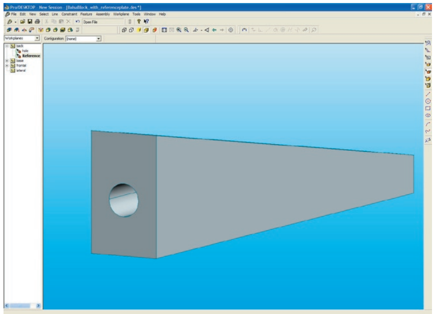
Your car design is created using a CAD software package. This guide describes the process using Pro/DESKTOP but other CAD packages may be compatible. There are two basic ways to create your design:

1) You can create your car design from a blank starting screen. If you use this method, you must make sure your final 3D model can fit inside the dimensions of the balsa wood billet. If you make your car design larger than the balsa wood, you may end up machining the jig used to hold the billet in your CNC machine. All the important dimensions of the balsa wood blank are shown in the diagram below.





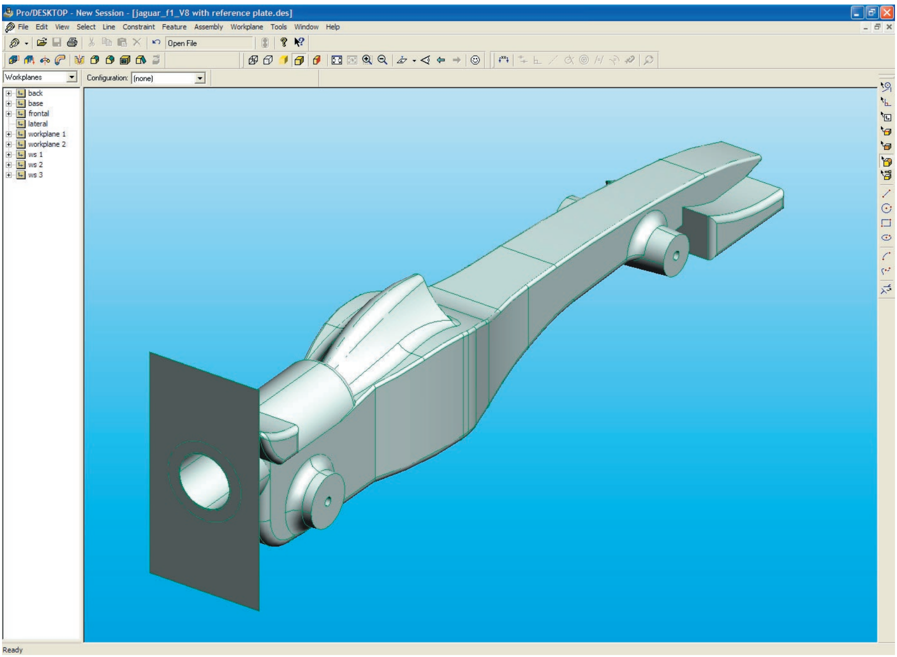
2) You can load a predrawn template, describing the exact shape and size of the balsa wood billet. Your car design is created by cutting into and removing sections from the template file. Using this method guarantees that your design will fit inside the balsa wood billet. If you need a copy of this starting template, visit the "downloads" section of the Denford website - *www.denford.co.uk*



Note - Remember to add the axle holes to your 3D model. Although the 1/4" ballnose cutter used will be too large to drill them, it will mark their position, making their positions easier to find after the main body shape has been completed.

1.2 - Add a reference plate to your finished car design

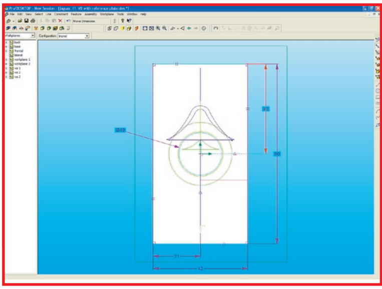
Before exporting your car design as an STL file, you must add a small reference plate to the CO2 cartridge end of your 3D model. The reference plate is used to help set the datum in the correct place, making it easier to use QuickCAM in the next stage. Also, if you use the same reference plate on all your car designs, you only need to configure the machine offsets once.

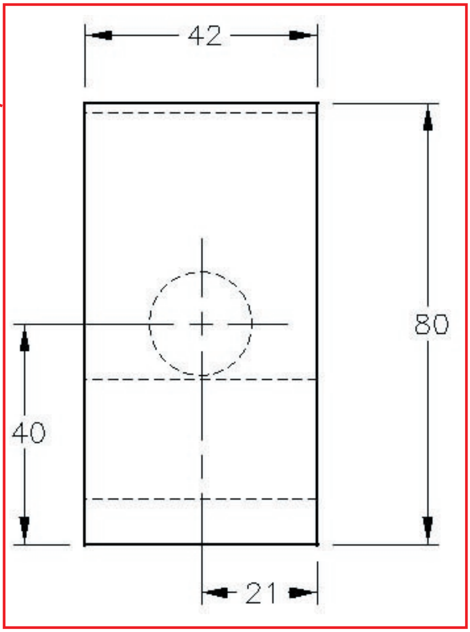


Creating the sketch for the reference plate:

(Note: The sketch is already created on the file balsa_wood_billet.des available from Denford (website). If you have used this to create your design then you can go to number 8) selecting 'Reference Plate' for the sketch name)

1) Select the back face of your car design (the flat face around the CO2 cartridge hole) and create a “New Workplane” on the plane of the object. Create a “New Sketch” called “reference plate”, then view directly onto this workplane.
2) Select a drawing tool, then left click in a blank area of the screen. Select “Reposition Axes” from the pop-up menu that appears. Snap the axes to the centre of the circle used to describe the CO2 cartridge hole. Holding your cursor just inside the perimeter of the circle will highlight its’ centre.
3) Draw a vertical line running exactly through the centre of the CO2 cartridge hole by snapping to X co-ordinate zero. Holding down the [SHIFT] key whilst drawing forces a vertical line. Toggle this to a “construction” line and “fix” it in position.
4) Draw a horizontal line running exactly through the centre of the CO2 cartridge hole by snapping to Y co-ordinate zero. Holding down the [SHIFT] key whilst drawing forces a horizontal line. Toggle this to a “construction” line and “fix” it in position.
5) Draw a rectangle around the CO2 cartridge hole. Attach and define a height dimension of 80mm and a width dimension of 42mm to the rectangle.

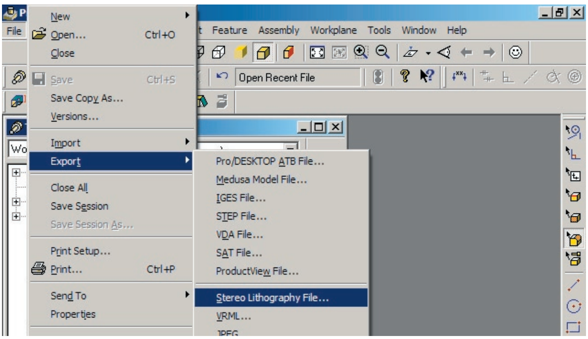




6) Attach a dimension between the vertical construction line and one of the vertical sides of the rectangle, then redefine the dimension to 21mm. Attach a dimension between the horizontal construction line and the bottom side of the rectangle, then redefine the dimension to 40mm.
7) Snapping the start point to the centre of the CO2 cartridge hole, draw a circle. Attach a dimension constraint to the circle, then redefine the value to 19mm (diameter) or 9.5mm (radius). Change to an “Isometric” viewpoint, “Autoscale” the drawing, then further manipulate the view so you can see the back face of the car design.
8) Select the “Extrude Profile” feature, to “Add Material”, “Above the workplane”. Drag the yellow square away from the back face of the car design, adding the rectangular plate containing the CO2 hole. Redefine the extrusion “Distance” to 1mm.
9) Save a copy of the car design as “<your name> with reference plate”.

1.3 - Export your car design as a Stereo Lithography File

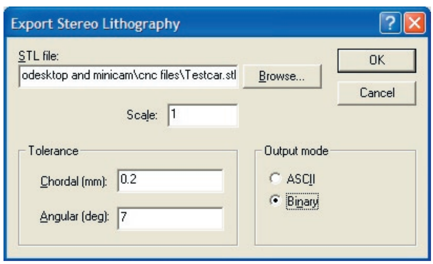
With your design loaded in Pro/DESKTOP, select 'File | Export | Stereo Lithography File...'



Click [Browse..] and choose a place to save your file, remember where you saved it to.

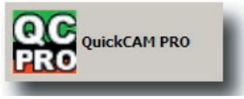
Ensure that the “Scale” is set to “1”.

If you set the “Output mode” to “Binary”, the size of your STL file will be smaller than the alternative “ASCI” option - this is useful if you intend saving your file on floppy disk.



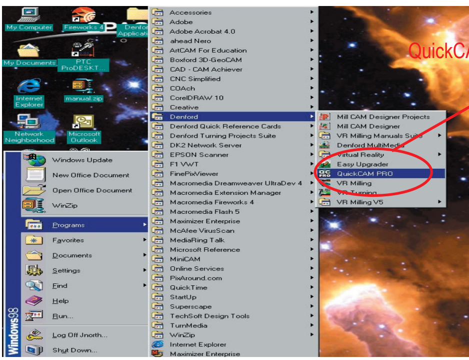
Stage 2 - QuickCAM 3D or QuickCAM Pro

The Denford QuickCAM series are simple to operate, wizard based CAM software packages used to convert your Pro/DESKTOP STL file into a CNC program that can be run on a CNC milling machine or router.



2.1 - Start the QuickCAM software

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

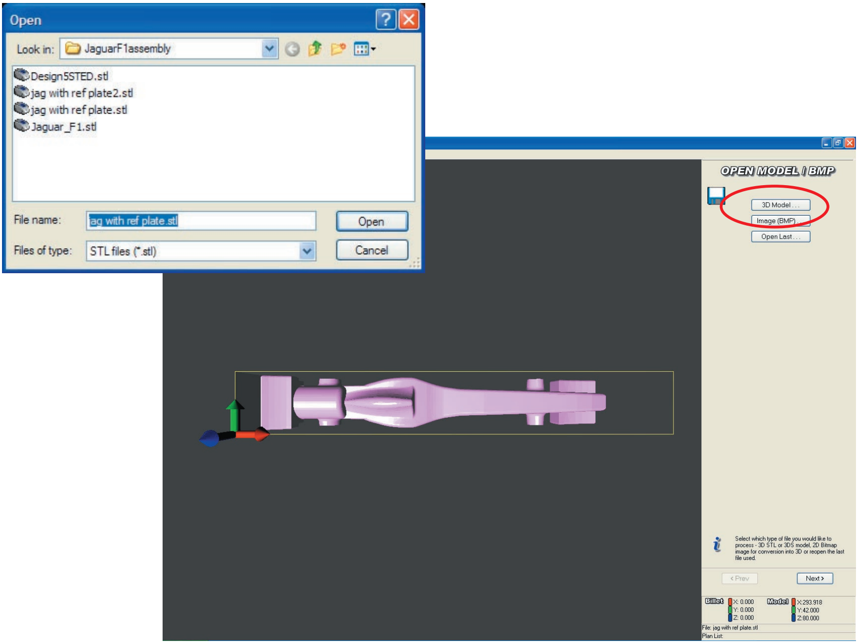


QuickCAM menu icon

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 3D or QuickCAM Pro” icon.

2.2 - Load your STL file

Click the **[3D Model..]** button. Browse for the file that you exported from ProDESKTOP and click **[OK],** your QuickCAM window should look like the example shown below. (Don't worry if the orientation of your model is different)



In the bottom, right hand corner of the screen you will see the dimensions of the model displayed. If your model is in the same orientation as above and you have used the reference plate as described previously, the dimensions for Y and Z will be 42mm and 80mm (the dimensions of the reference plate). The X dimension will vary depending on the length of the car. Each dimension is colour coded and a corresponding arrow on the screen shows the direction of each axis.

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

Manipulating the View

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

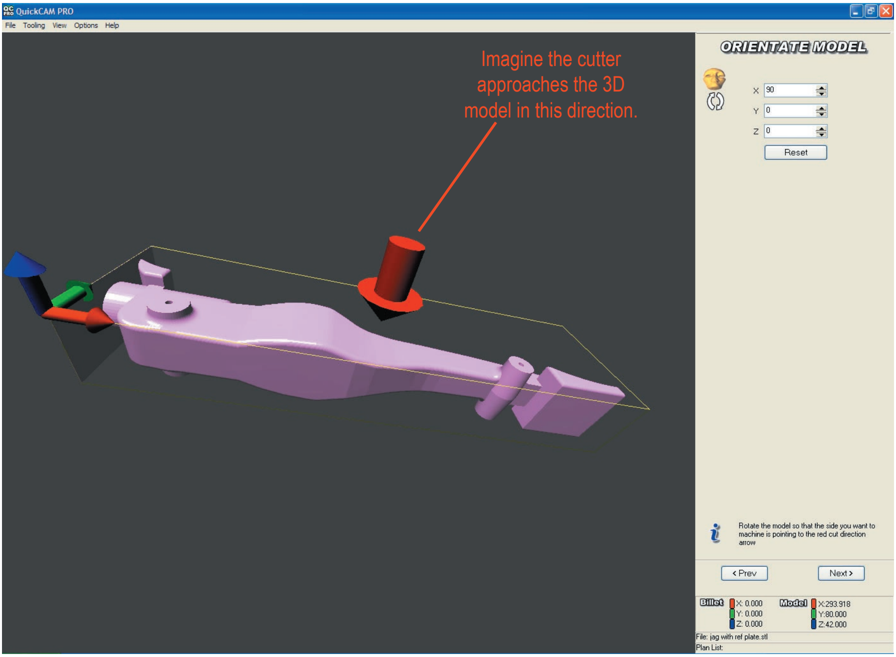
Click and hold the right mouse button whilst moving the mouse to zoom in or out.

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

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

2.3 - Orientating the model

The "Orientate Model" screen allows you to change the orientation of the 3D model within the CNC machine. This can also be described as changing the direction that the model will be cut from.



You should imagine the cutter approaching your part in the direction of the red arrow. If necessary, manipulate the view so that you can clearly see the direction of the arrow.

You will need to change the orientation of the model.

If you have used the balsa wood template file (downloadable from www.denford.co.uk/downloads) then the model will need to be rotated by 90 degrees around the X axis. Don't worry if your model is in a different orientation than shown, you can rotate the model any number of degrees around any axis.

Orientate the model so it appears as above and then click the **[Next>]** button to move to the next screen.

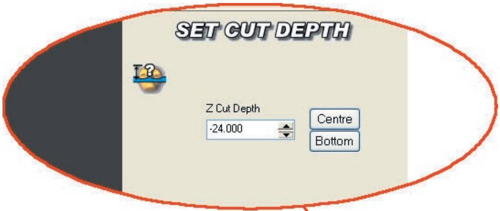
2.4 - Set cut depth

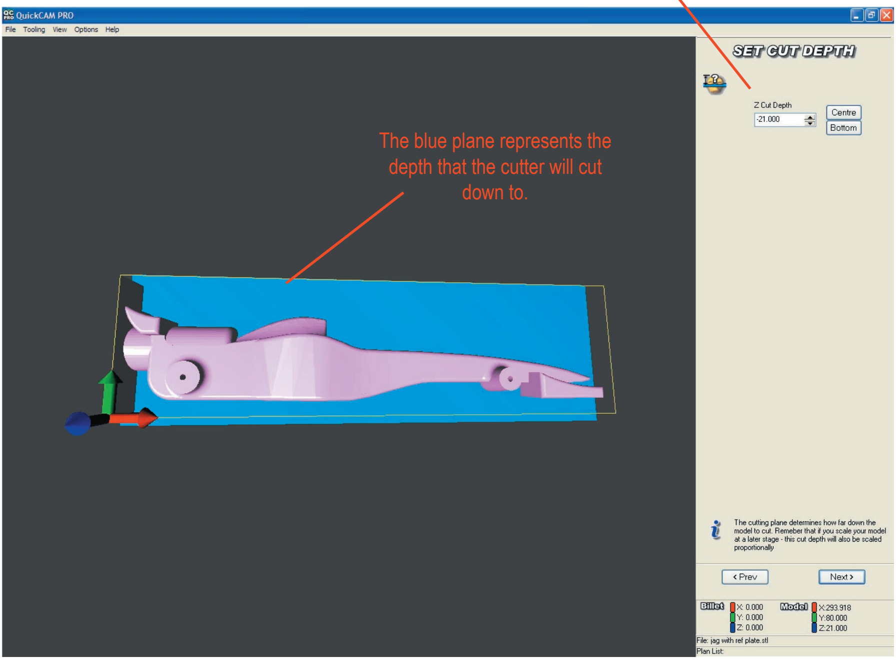
The “Set Cut Depth" screen allows you to determine how far down the model to cut.

We will be cutting half of the model and then rotating the billet in the machine to produce the second side of the car.

The default option on this screen is to cut to the centre of the model, you should find that the "z cut depth" is at -21. We are going to lower this value by the radius of the cutter to prevent a ridge being left around the car.

Set the value to -24



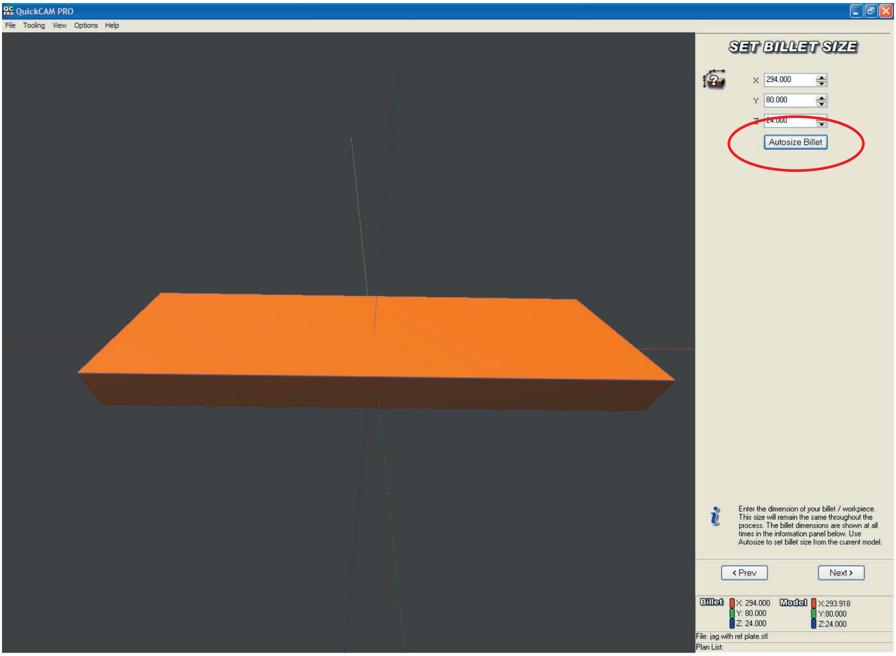


Click the **[Next>]** button to move to the next screen.

2.5 - Set the billet size

This page allows us to set the size of the material we are going to use.

Instead of entering the actual size of the piece of Balsa Wood here, just click the **[Autosize Billet]** button. This will create a piece of material that is as big as the model of your car.



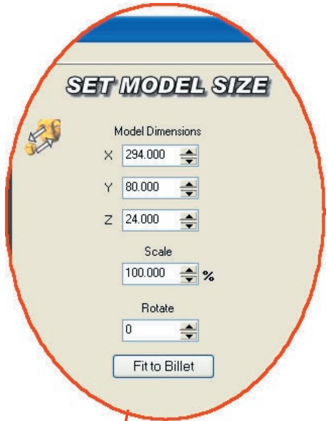
Click the **[Next>]** button to move to the next stage of the process.

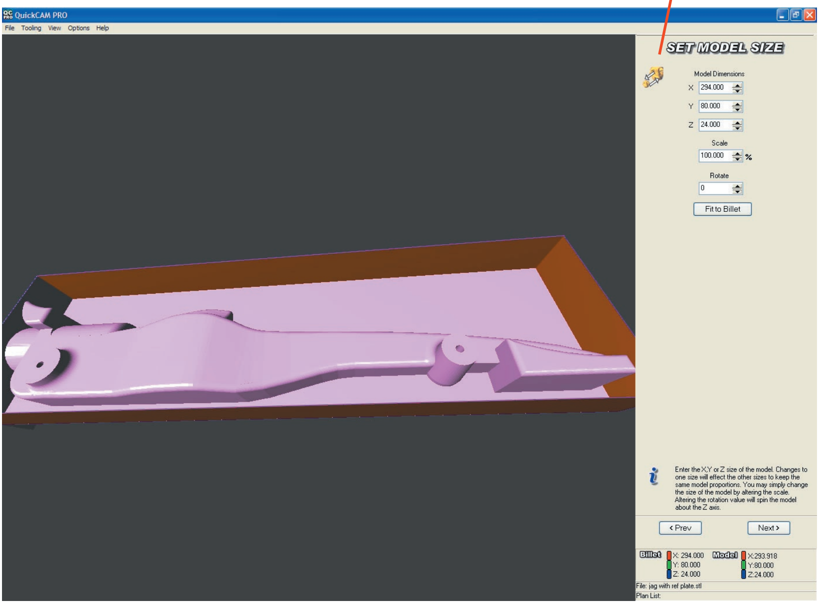
2.6 - Set Model Size

The “Set Model Size" screen can be used to scale your model up or down, this must **NOT** be done when making F1 Team in Schools cars.

The scale **MUST** be at 100%.

(If you were to alter the scale the datum position would change. The datum is the start point for the program and you need to know exactly where that is.)



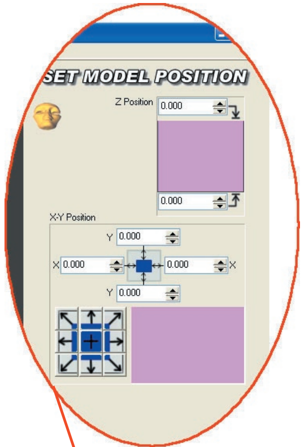


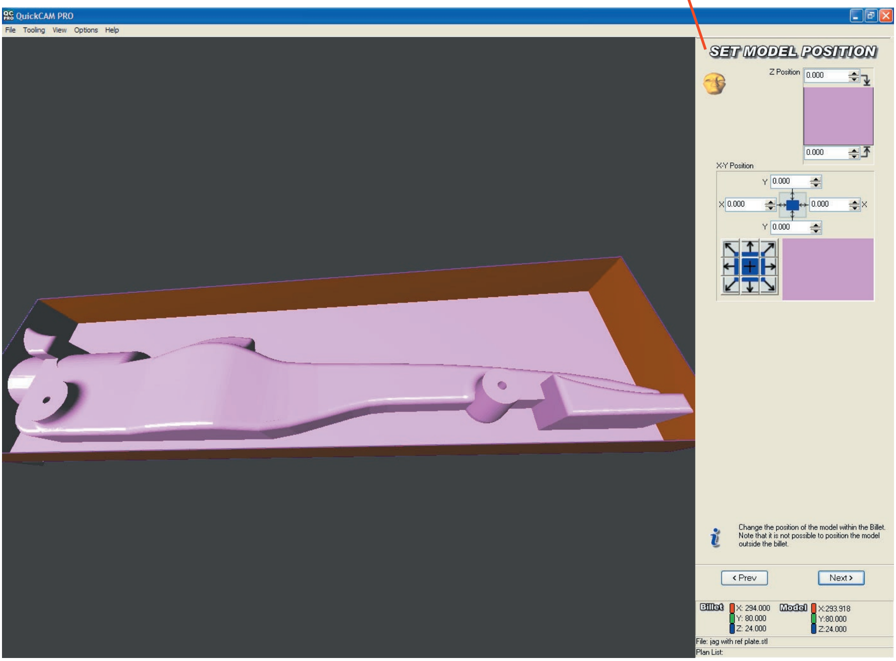
Click the **[Next>]** button to move to the next stage of the process.

2.7 - Set Model Position

The “Set Model Position” screen allows you to position the 3D model within the billet material.
The options on this screen allow you to position the model in the Z axis and the X and Y axis by dragging sliders on screen or entering values.

Because our Billet size was set to match our model size we cannot reposition the model within the material. (there is no excess material around the model)





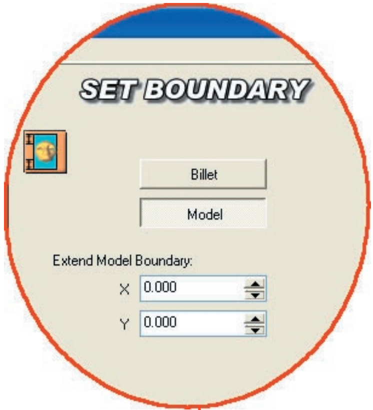
Click the **[Next>]** button to move to the next stage of the process.

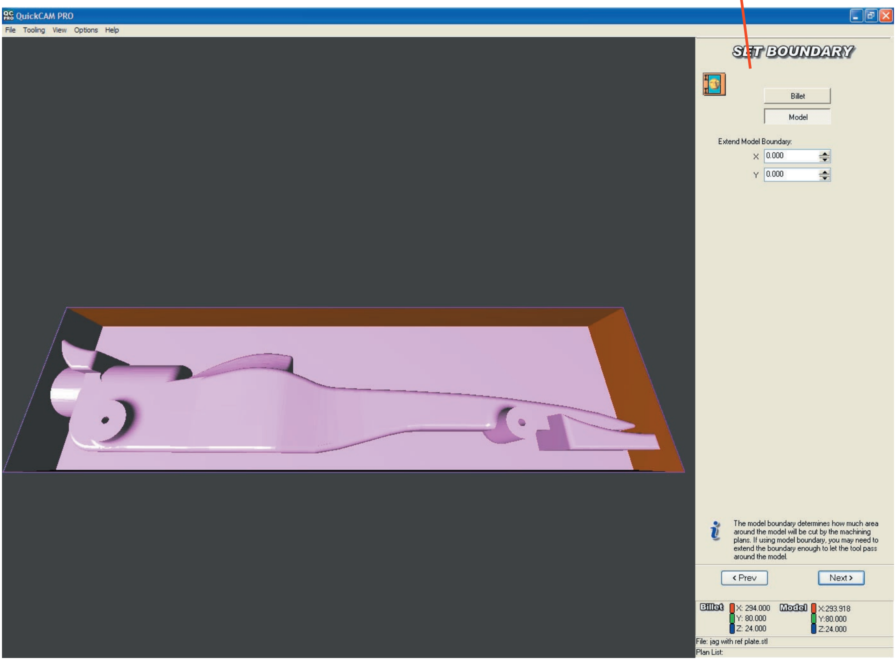
2.8 - Set Boundary

The “Set Boundary" screen offers 2 options: Billet or Model. Here you can choose whether to machine the area the model occupies within the Billet, or to machine the whole Billet.

The choices we have made in the previous screens mean that the Billet and the model are the same size, so, either option will give the same result.

Leave the default option 'Model' and then click the **[Next>]** button to move to the next stage of the process.



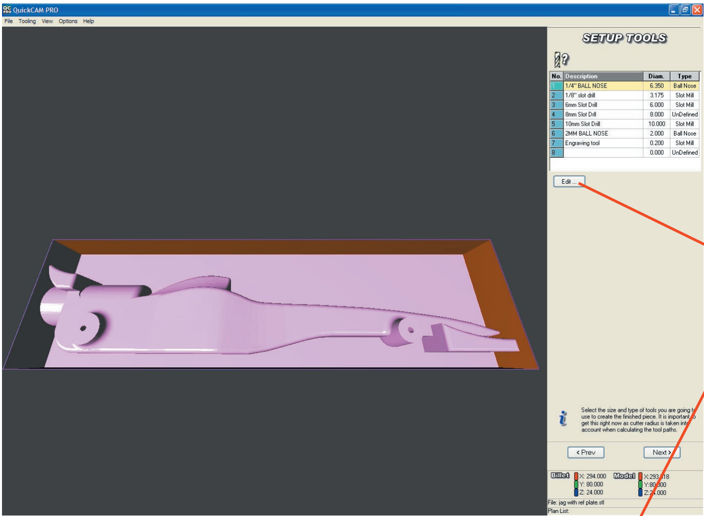


2.9 - Setup Tools

The "Setup Tooling" stage allows you to choose the type of cutting tool you will use when manufacturing the F1 car.

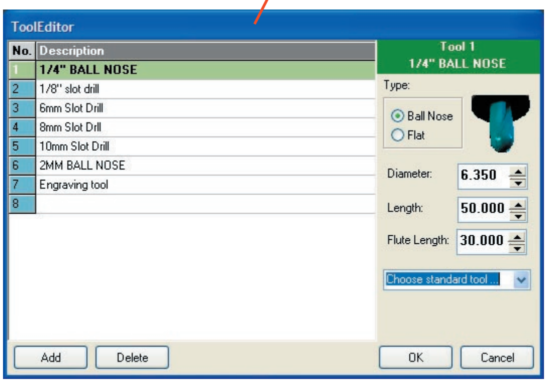
For the F1 Team in Schools car we recommend using the 1/4" Ball Nose Cutter which is supplied with every Denford CNC Router.

Ensure that Tool No1 is the 1/4" Ball Nose Cutter. To change the tool highlight Tool No1 in the list and click the Edit ... button.



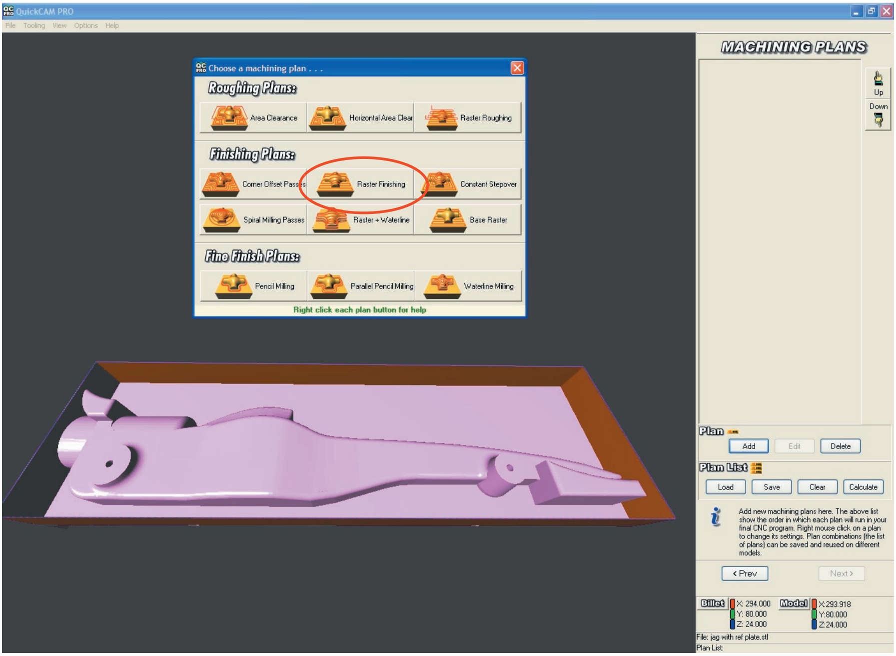
Choose the 1/4" Ball Nose Cutter from the list of standard tools and click **[OK]**.

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



2.10 - Machining Plans

This section will affect the outcome of the part and the way it will be machined.



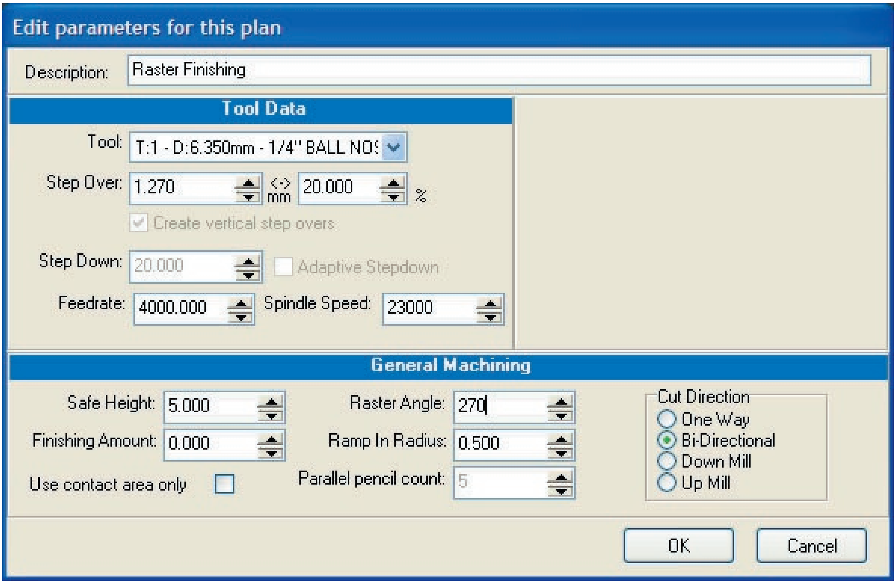
The Machining Plans section of the software is the only section which differs between QuickCAM Pro and QuickCAM 3D. QuickCAM Pro offers more machining plans than QuickCAM 3D.

If you have QuickCAM Pro you can experiment using different Plans to give a better finish to the model. For this example we will use a Plan which is common to both software.

Choose the 'Raster Finishing' Plan.

2.11 - Edit Plan Parameters

Edit the plan parameters so they appear as below.



•The 'Step Over' dictates the finish quality of the model, the lower the value the better the finish. However, the lower the value the longer the machining time. A compromise between finish quality and time needs to be found. 1.27mm or 20% of the cutter diameter is a good compromise for the F1 cars.

•The Feedrate when using a Denford Microrouter should be 4000mm/min

•The Spindle Speed should be 23,000 rpm

•The 'Raster Angle' should be set at 270 degrees. This will mean the cutter will start at the left of the car as you look into the machine and will cut in the Y axis, this will provide a good finish. Changing this angle will change the direction of cut and will affect the quality of finish.

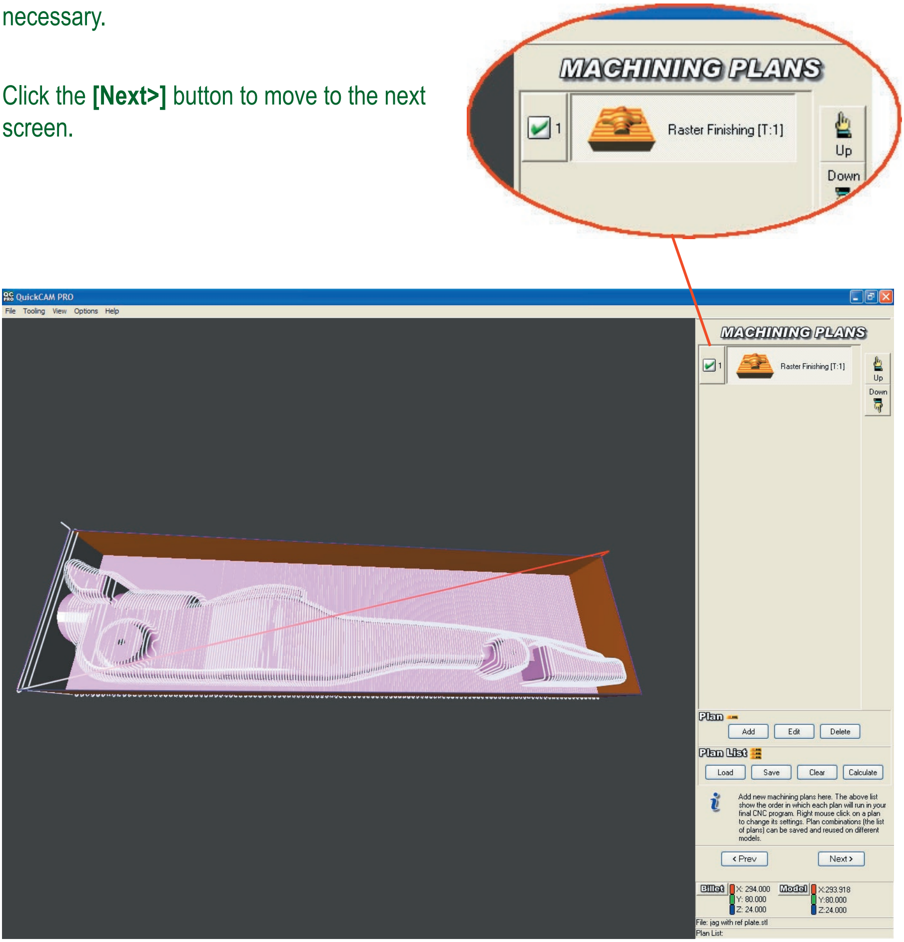
Click **[OK]** and the toolpaths will be calculated.

2.12 - Create the toolpath

Once you have clicked **[OK]** to the 'Edit parameters for this plan' dialogue box you will have to wait a minute or so while the software calculates the toolpaths.

When the Toolpaths have been calculated you will see them appear as lines on the model. The plan will appear in the list. When machining the F1 cars only this plan is necessary.

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



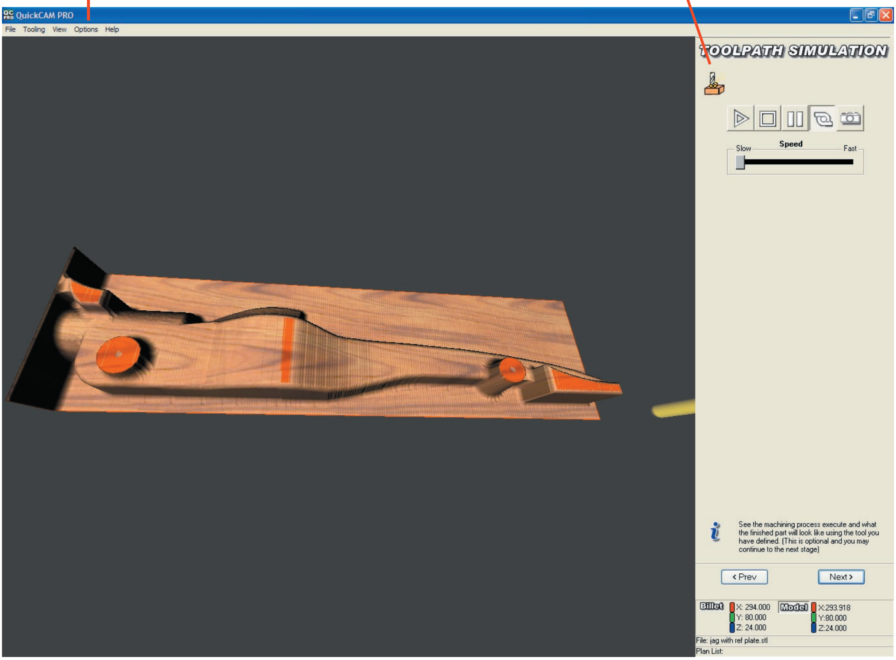
2.13 - Simulate the toolpath

On this screen the toolpaths can be simulated. Press the Play buton to start the simulation, to speed the simulation up, adjust the slider.

The screen can be captured at any point during the simulation by clicking the camera icon.

The simulation can be rendered in different materials such as wood in the image below.

Choose 'Options > Simulation Colours' from the menu to change the material



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

2.14 - CNC File Output

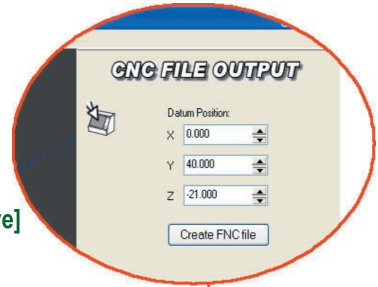
This is the final screen in QuickCAM and is where the CNC file is saved. Before we click the [Create FNC file] button, the 'datum' position will be changed. The 'datum' is the start point for the program. We will move the datum to the centre of the hole in the Balsa Wood blank. Drawing the reference plate in ProDESKTOP makes the centre of the hole easy to find. The reference plate was 80mm x 42mm, and was centered around the hole.

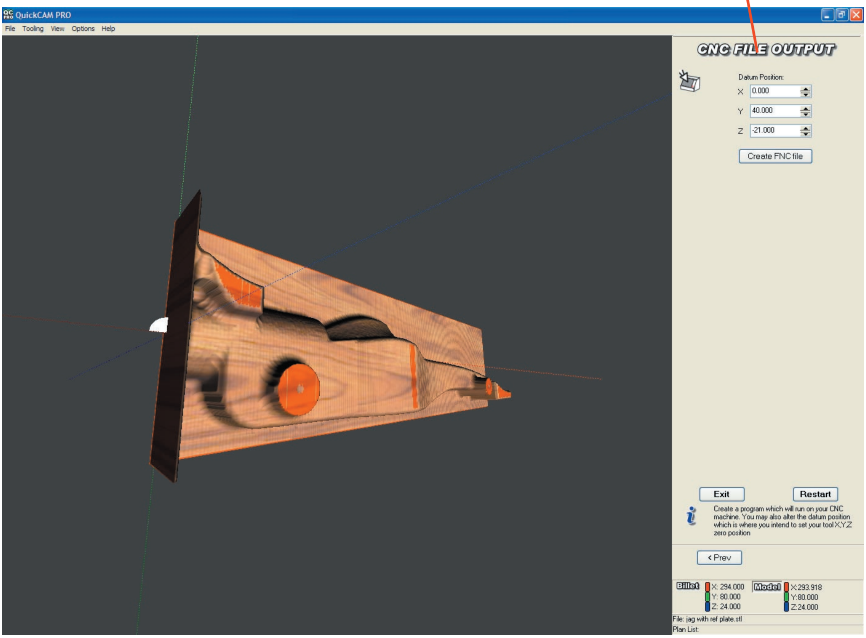
We therefore need to change the datum position to X = 0, Y = 40mm and Z = -21mm to find the centre of the hole.

Click the **[Create FNC file]** button.

Browse and choose a location to save the file.

Remember where you saved the file to and click **[Save]**



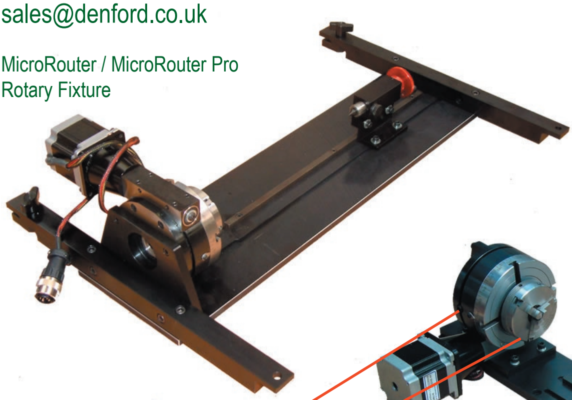


4th Axis Programmable Rotary Fixture

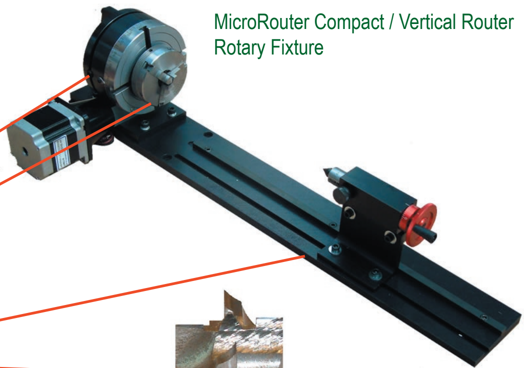
The Denford 4th axis fixture is designed for use with the Denford range of CNC Milling machines and Routers. It allows 360 degree rotation of material.

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

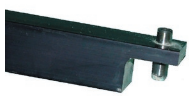


The 4th axis system comprises:

- 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.

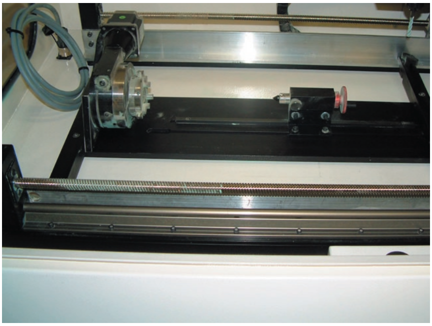
Dowel & Clamp (Microrouter / Microrouter Pro)



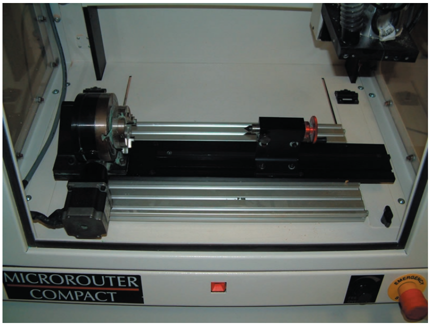
'Tee-nut and bolt' (MicroRouter Compact / Vertical Router

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



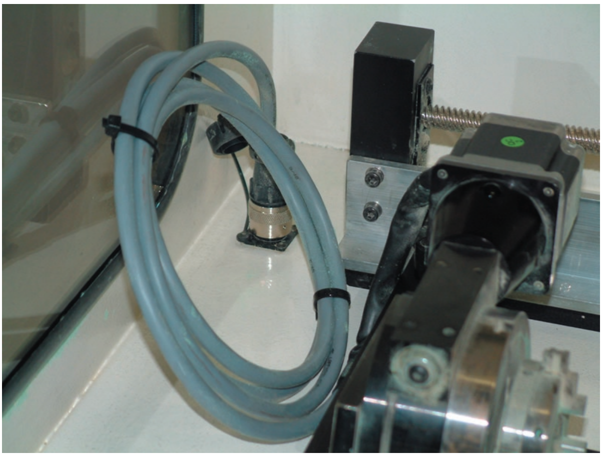
Microrouter Compact / Vertical Router

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

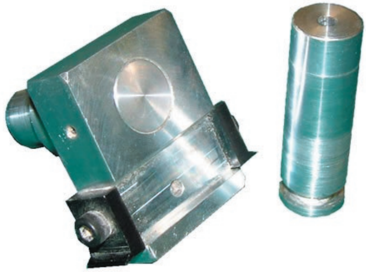
Socket with cap removed



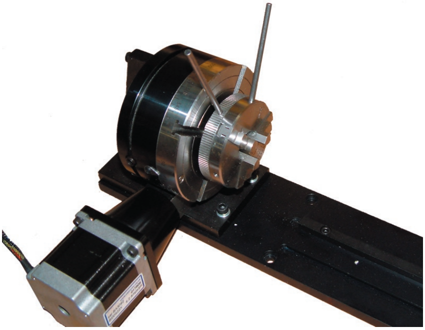
Plug and 4th axis rotary fixture fitted

Fit the billet in the 4th axis rotary fixture

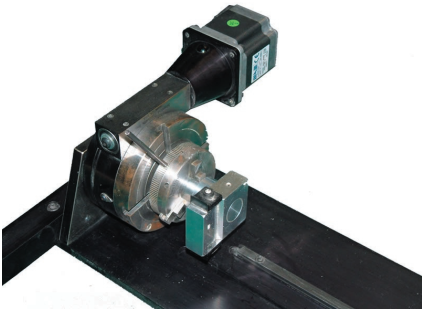
In order to produce 'F1 Team in Schools' cars using the 4th axis rotary fixture it is necessary to purchase a '4th axis F1 conversion kit' from Denford Ltd. The kit comprises of the 'face driver' and 'rear shaft'. Contact the Denford sales team on 01484 728090 for more information.



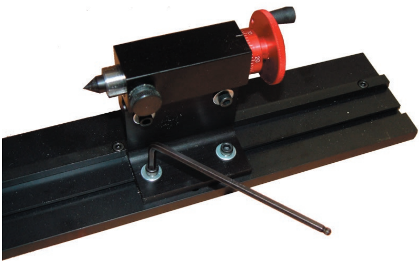
The 4th axis rotary fixture is fitted with a '3 jaw spindle chuck'. Two tommy bars are supplied with the 4th axis fixture which are used to open and close the chuck. It is the inner, grooved section of the chuck which is rotated to open and close the chuck jaws.

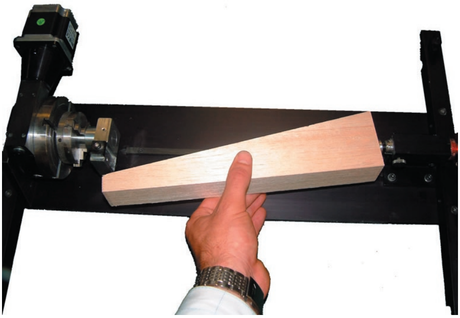


3 jaw spindle chuck with the 2 tommy bars used to open and close the chuck



3 jaw spindle chuck with the face driver fitted.



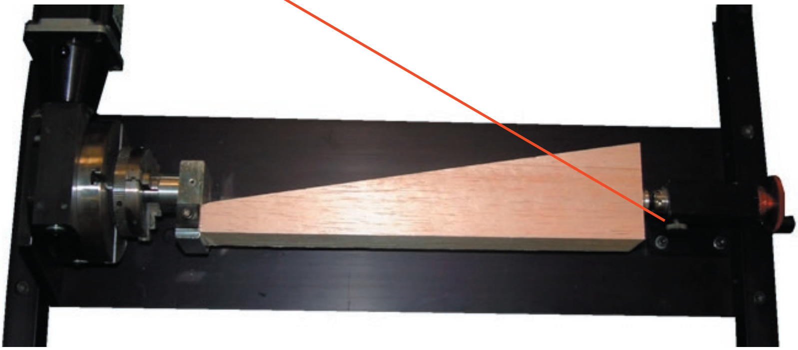


The 4th axis rotary fixture is fitted with an adjustable tailstock. The tailstock spindle can be wound in and out using the red anodised wheel. By using a 4mm Allen key the tailstock can be repositioned along the length of the fixture. To fit the billet in the 4th axis fixture, reposition the tailstock so that the billet can fit between the tailstock and the face driver. Push the 'rear shaft' into the hole in the balsa wood billet.

Fit the billet in the 4th axis rotary fixture

Locate the narrow end of the billet with the recess in the face driver and wind the centre on the tailstock spindle into the hole in the rear shaft.

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

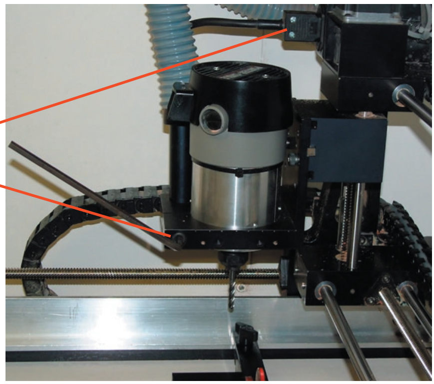


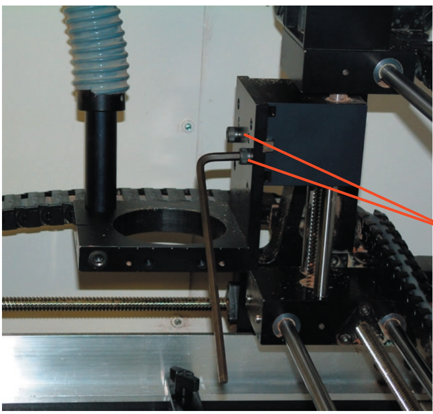
Raise the spindle motor on Microrouter/Microrouter Pro

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.

3.15 - Preparing the Programs

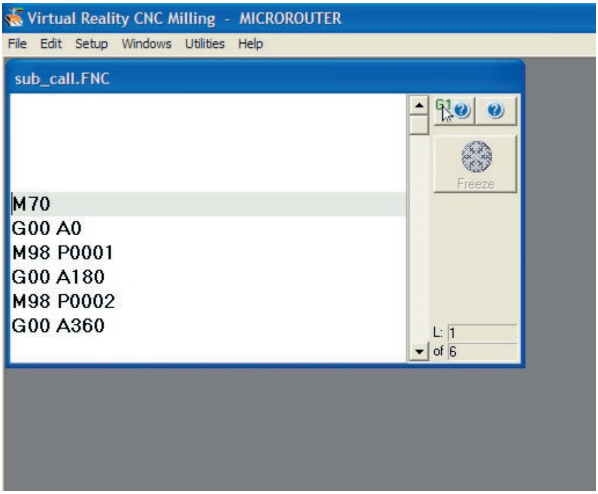
The Main Sub Call Program

The program shown below is the main program that is used to machine both sides of the car. If you do not have this file already, type it into the editor and save it as '**F1_sub_call.FNC**'.

This program ***must*** be saved into the same folder as the two sub programs.

This main, sub call program asks the 4th axis rotary fixture to set at 0 degrees then call program 0001.fnc then rotate the fixture to 180 degrees and finally call program 0002.fnc

Note: Comments are not permitted in this program.



The Two Sub Programs

Two programs are required to machine each side of the car. They ***must*** reside in the same folder as the main 'sub_call' program. The filename for the fist side ***must*** be called '0001.FNC' and the filename of the second side ***must*** be called '0002.FNC'.

How to create the two programs:

Stage 1 - Load your CNC file saved from the QuickCAM software into the VR CNC Milling editor.

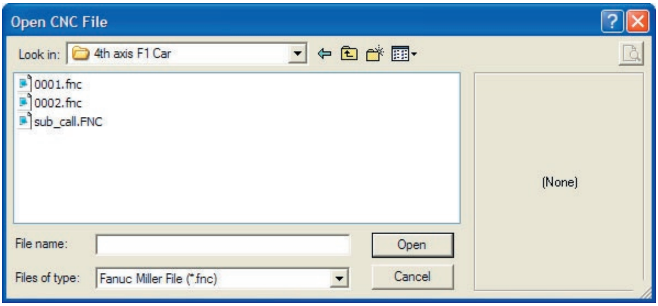
Stage 2 - Scroll down to the end of the program and replace the 'M30' with '**M99**'.

Stage 3 - Use 'Save as...' option to save the file as '**0001.FNC**'

Stage 4 - Scroll back up to the top of the program and insert an '**M71**' as the first line.

Stage 5 - Use 'Save as...' option to save the file as '**0002.FNC**'

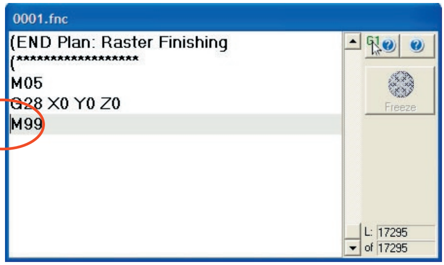
As mentioned previously it is important that the main 'sub_call.fnc' program is saved in the same folder as the programs that create the 2 sides of the car. In the example below all 3 programs are saved into a folder called '4th axis F1 car'

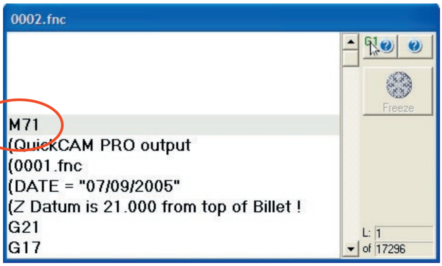


Remember to edit the program '0001.fnc'.

Scroll to the bottom of the editor

Replace the M30 with M99 at the end of the program





Scroll back to the top of the program

Press the [Insert] key on the keyboard and type M71 to mirror the program in the Y axis

Save the file as '0002.fnc'

Explanation of 'sub_call.fnc'

The first line of 'sub_call.fnc' - M70, commands all coordinates to be mirrored in the X axis. This means the car will be manufactured in the correct orientation. The second line 'G00 A0' asks the 4th axis to start at 0 degrees. The 3rd line M98 P0001 calls program 0001.fnc. The 4th line G00 A180 tells the 4th axis to rotate by 180 degrees. The 5th line, M98 P0002 calls program '0002.fnc'

```
M70
G00 A0
M98 P0001
G00 A180
M98 P0002
G00 A360
```

Stage 3 - Using Denford VR Milling V5

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).

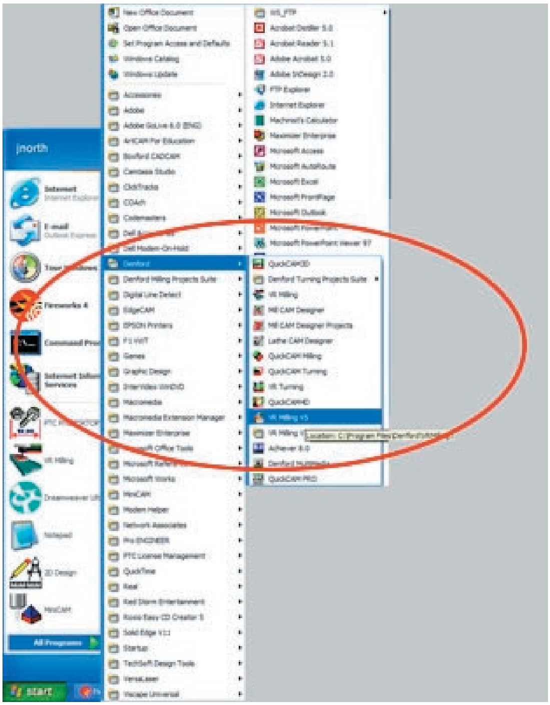
The VR Milling V5 software contains detailed help files including tutorials and animations. Access these by going to Help on the menu.

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.

3.1 - 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.



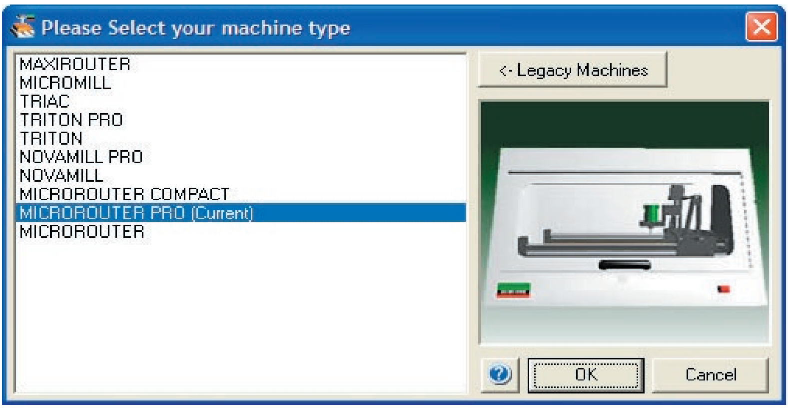
3.2 - Configure the software for the machine

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.

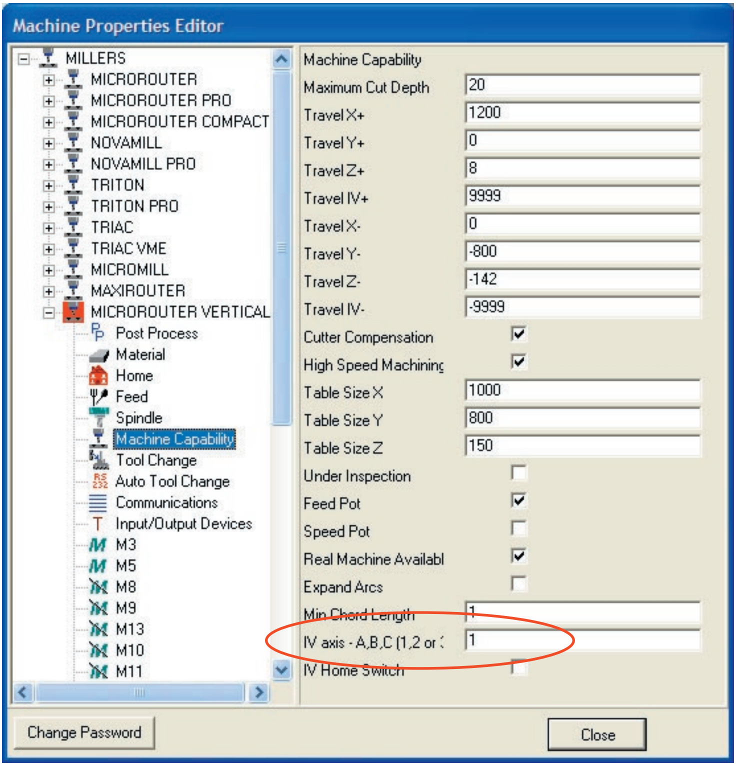
Step 3 - Enable the 4th axis.

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

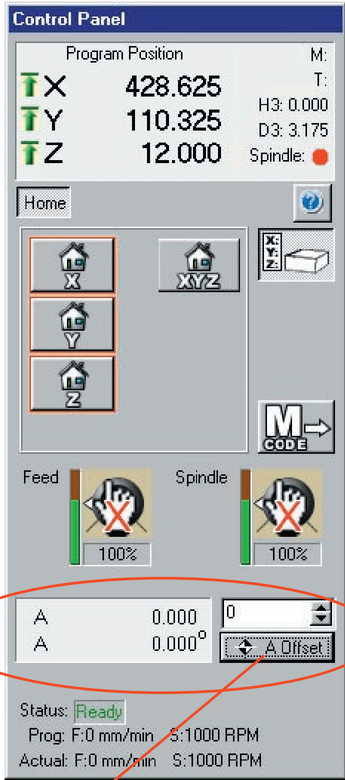
To reconfigure VR Milling V5 to enable the 4th axis you need to change the following options:

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 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).

When manufacturing the F1 cars with the 4th axis the billet will need to be rotated to the correct angle before the program is run. 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.

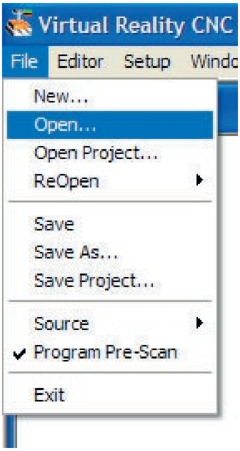
Note: When the machine is powered off the angle the 4th Axis is currently in becomes zero the next time the machine is powered on. This can be used to set the A offset. By jogging the 4th axis to the desired angle and then turning the machine off and back on the 4th axis position becomes zero meaning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

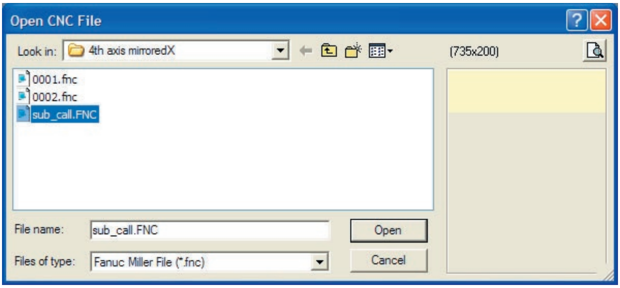
3.3 - Load your CNC file.

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

Browse to the drive and folder containing your CNC files (remember the files '0001.fnc' and '0002.fnc' must exist in the same directory as the 'sub_call.fnc' program)

Find 'sub_call.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.

Should you need to edit the CNC file or write one from scratch the following key commands can be used:

INSERT :	will insert a line at the current highlighted line
SHIFT+DELETE :	will delete the whole of the current line
CTRL+HOME :	will jump the editor to the start of the program
CTRL+END :	will jump the editor to the end of the program
PAGE UP/PAGE DOWN :	will scroll the editor up or down by the number of lines shown at any one time.

The sub call program required to rotate the 4th axis rotary fixture and call the 2 programs needed to machine each side of the car is printed below:

```
M70
G00 A0
M98 P0001
G00 A180
M98 P0002
G00 A360
```

3.4 - 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.

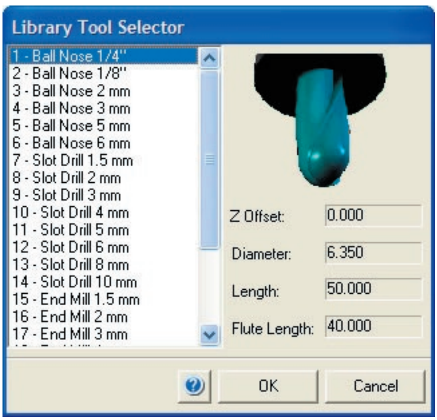
“Save tool to Library”

“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.

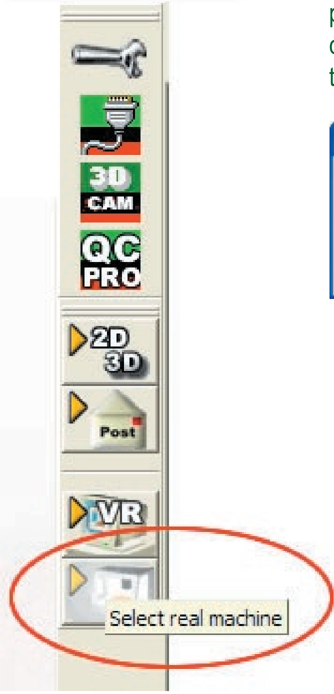


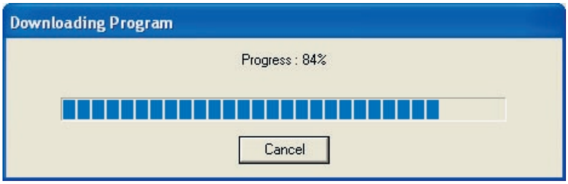
Click the [Tool and Offset Editor Window] button again to close the Window down.

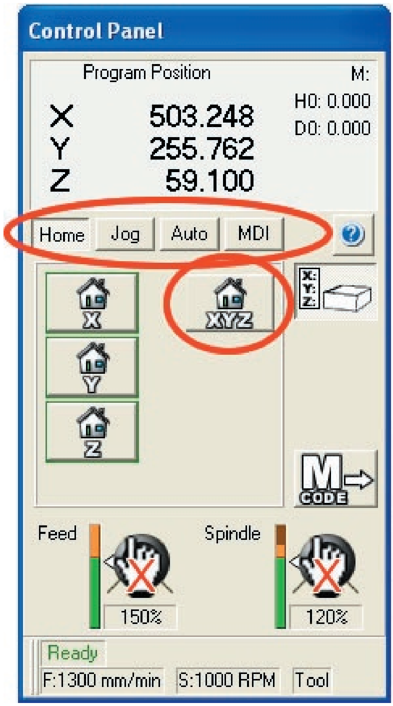
3.5 – Connect to, and Home the CNC machine

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.

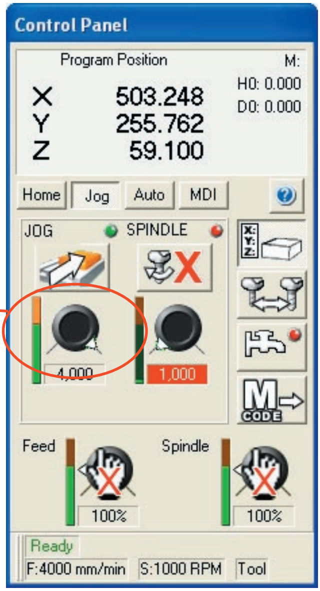
3.6 - Move the machine head and fit the cutting tool.

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.

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.



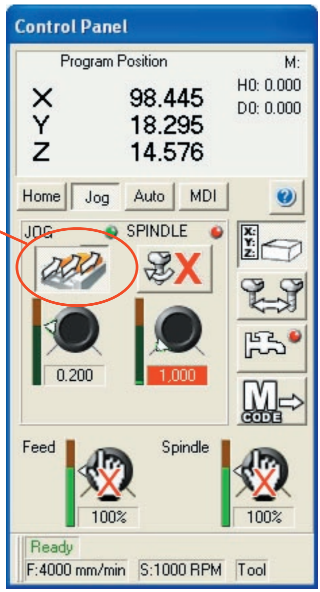
"Jog Continous"

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.

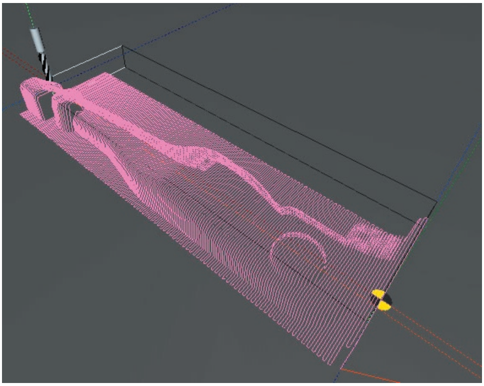
Jog the machine head to an appropriate position, then, fit the cutting tool. The procedure for this will vary depending on the machine type. See the machine manual for more detailed information on this procedure.

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



"Jog Step"

3.9 – Set the work offsets



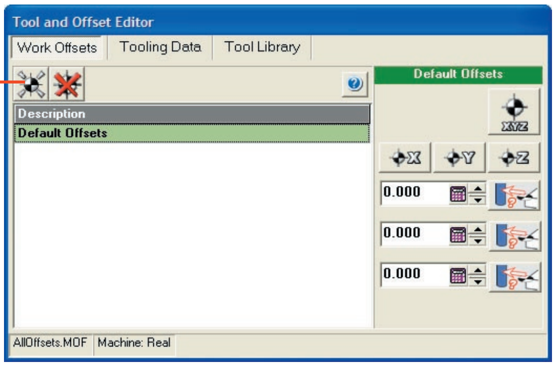
What are offsets?

Offsets are the distances the cutter needs to travel, from its 'Home' position to the reference point or datum for the program in X Y & Z. In QuickCAM the datum was set as the centre of the hole

We are now going to define an offset for the centreline of the hole in the Balsa Wood Blank. We will need to create a new offset, name it, and then set the X,Y and Z values.

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

"New work offset" button



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.

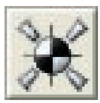
•Click the "New work offset" button.

•Click on the ‘blank’ offset that has been added to the list to select it.

•Type in a description for your new offset.

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

The active offset is highlighted in green.

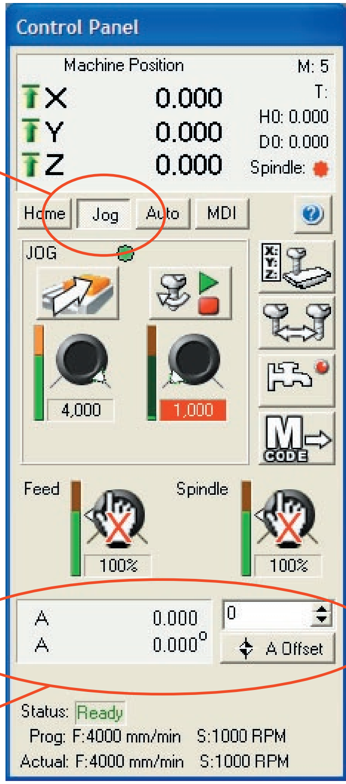


"New work offsets"

"activate offset"

3.9 - Set the A axis position

In Jog mode in the Control Panel, jog the A (rotary) axis until the face plate is at 90 degrees to the bottom of the 4th axis rotary fixture. The axis will jog more slowly if the guard is open.

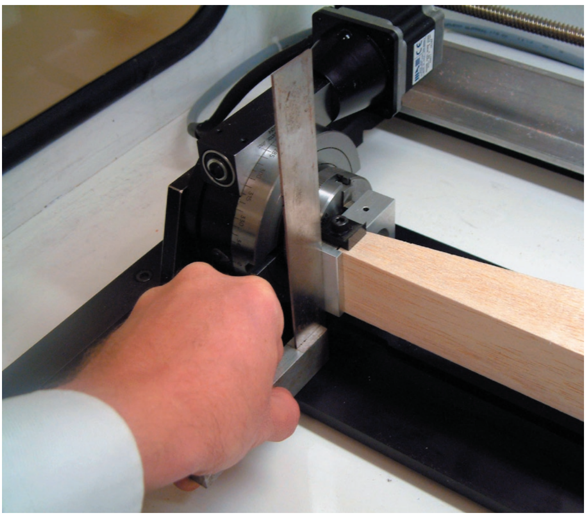


The 4th axis can be rotated by using the **[.]** and **[/]** keys on the QWERTY keyboard when in jog mode.

A Offset button

It is possible to set the work offset for the A axis (Rotary axis).

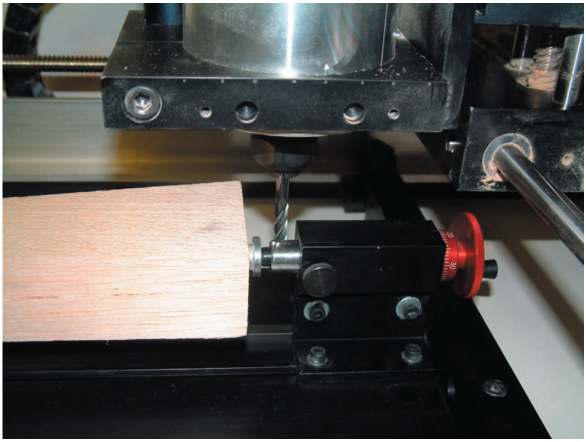
When manufacturing the F1 cars with the 4th axis the billet will need to be rotated to the correct angle before the program is run. Jog the 4th axis fixture to the required angle and click the **[A Offset]** button to set the angle you would like to start from.

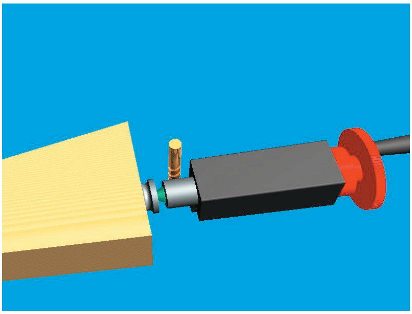


Jog the 4th axis until the face plate is at 90 degrees to the base plate of the 4th axis rotary fixture.
Use an engineers square when aligning the fixture in the chuck

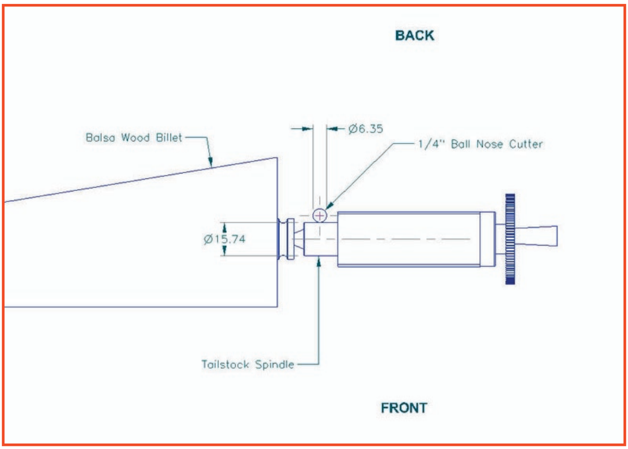
3.9 - Finding the centre of the location fork - Y axis

In 'jog continuous mode', move the tool a few millimetres away from the tailstock spindle. We are going to touch onto the tailstock spindle.
(The tailstock spindle shown here is 15.74mm, in diameter.)

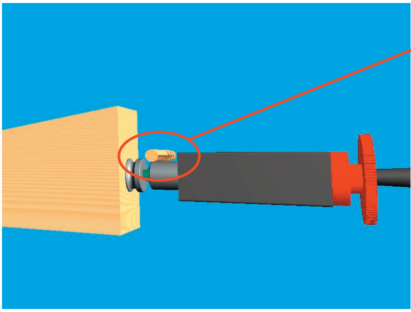




Now change to 'jog step mode' for fine incremental movements and position the cutter so it just touches the tailstock spindle. You can place a thin strip of paper between the cutter and the tailstock spindle to detect precisely when the cutter touches.



3.9 - Setting the Y axis offset

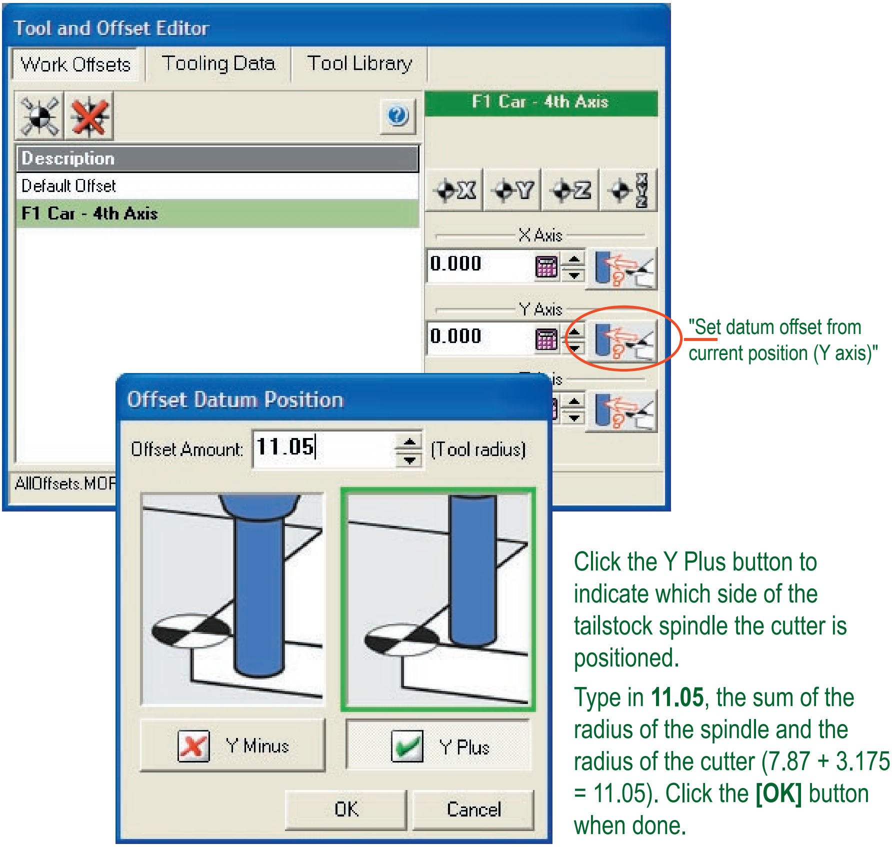


With the cutter in this position we will find the centre of the tailstock spindle.

To find the centre of the spindle we are going to include the sum of the radius of the spindle and the radius of the cutter.

The radius of the spindle is 7.87mm and the radius of the cutter is 3.175mm. (7.87 + 3.175 = 11.05mm).

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.



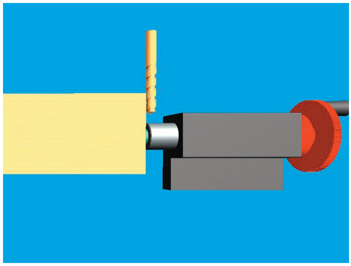
Click the Y Plus button to indicate which side of the tailstock spindle the cutter is positioned.

Type in **11.05**, the sum of the radius of the spindle and the radius of the cutter (7.87 + 3.175 = 11.05). Click the **[OK]** button when done.

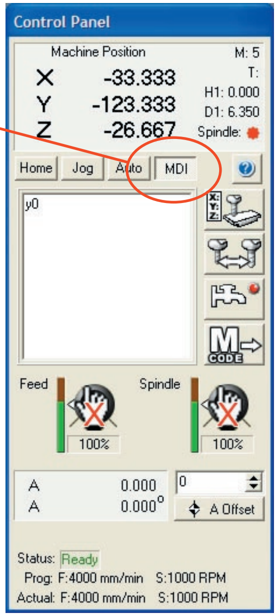
3.9 - Finding the centre of the location fork - Z axis

Click the 'Jog' tab to go back to the jog mode.

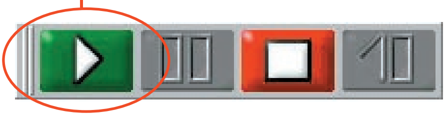
Using the jog keys, move the tool towards you and away from the tailstock spindle. Use the 'Page Up' key to lift the tool above the tailstock spindle, ensure that the tool is above the tailstock spindle before proceding to the next step.



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

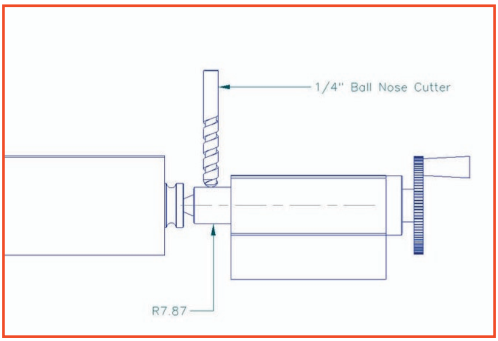


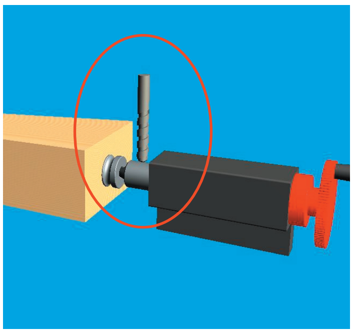
Press Play on the file control.



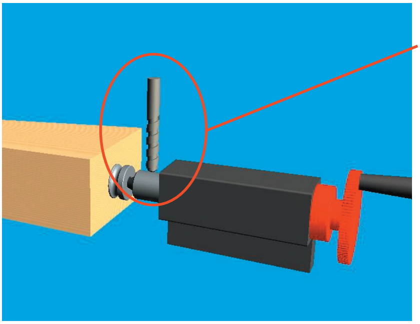
The cutter will move to the Y position set previously. This should be directly over the centre of the location tailstock spindle, if not the Y offset is incorrect, repeat the previous operation.

Using the jog keys in 'jog step mode', bring the nose of the cutter down so it just touches the section of the tailstock spindle where the diameter is 15.74mm as shown.





3.9 - Setting the Z axis offset

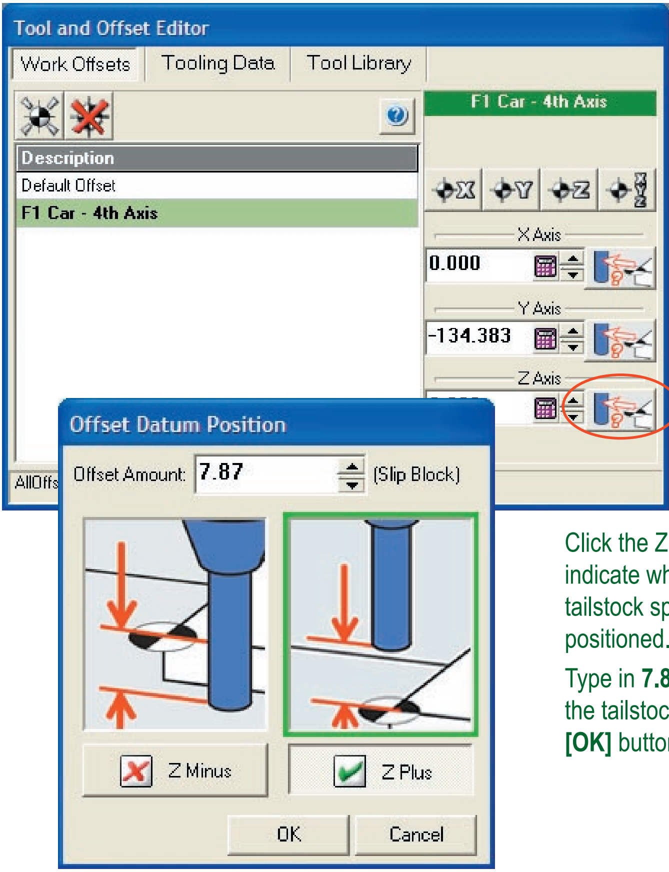


With the cutter in this position we will find the centre of the tailstock spindle.

To find the centre of the tailstock spindle we are going to include the radius of the tailstock spindle. We will tell the machine to travel an extra -7.87mm

(The radius of the tailstock spindle is 7.87mm)

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



"Set datum offset from current position (Z axis)"

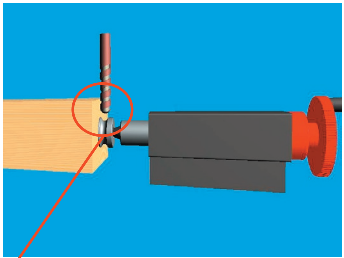
Click the Z Plus button to indicate which side of the tailstock spindle the cutter is positioned.

Type in **7.87mm,** the radius of the tailstock spindle. Click the **[OK]** button when done.

3.9 - Finding the edge of the billet - X axis

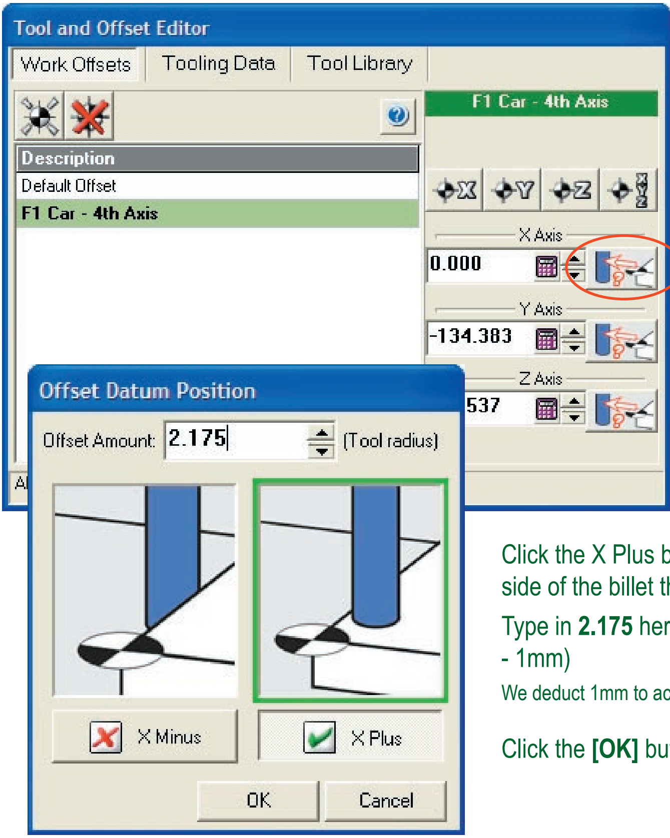
Click the 'Jog' tab to go back to jog mode.

Using the jog keys, move the cutter as shown so it just touches the side of the balsa wood billet.



3.9 - Setting the X axis offset

With the cutter in this position we will find the edge of the billet. To find the edge of the billet we are going to include the radius of the tool.
With the cutter in the above position click on the "Set datum offset from current position (X axis)" button in the "Tool and Offset Editor" window.



Click the X Plus button to indicate which side of the billet the cutter is positioned.

Type in **2.175** here. (The cutter radius - 1mm)

We deduct 1mm to account for the datum plate.

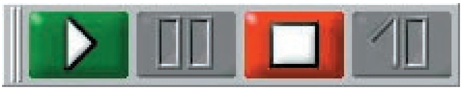
Click the **[OK]** button when done.

3.10 – 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.

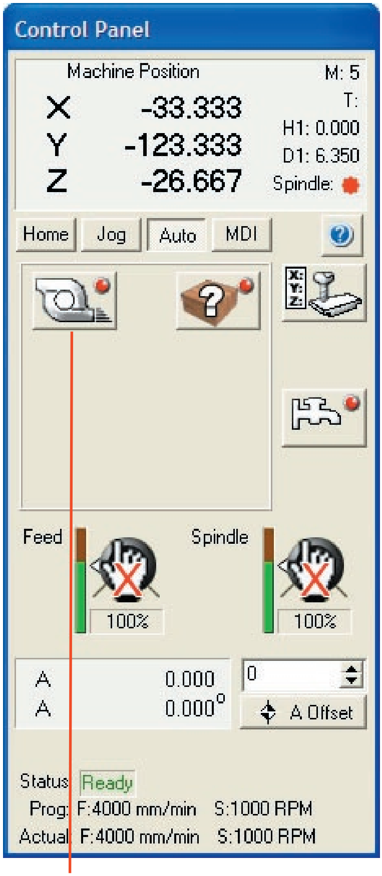


The program will begin to run. A message may appear asking you to change to Tool Number *. Check you have the correct tool and click [OK]. The spindle will start and the program will begin.

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

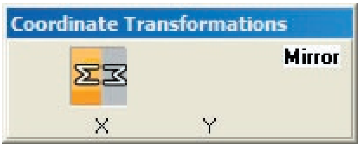


Turbo Mode

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 CAD/CAM software are run with turbo mode on.

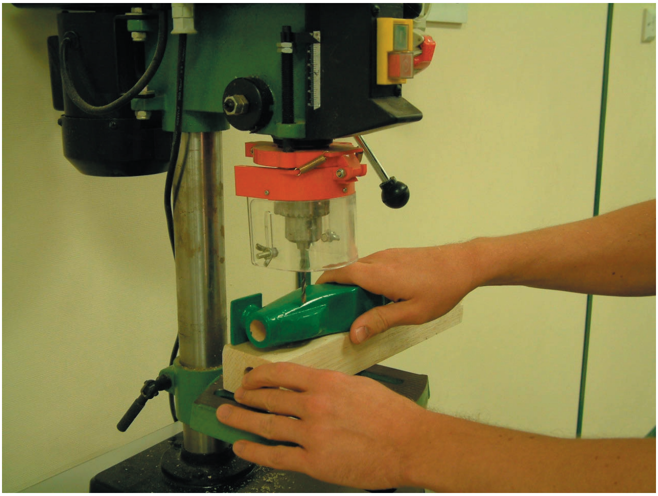
3.10 – Run the Program.

As soon as the program begins to run you will see the 'Coordinate Transformations' window appear. This confirms that the program coordinates are being mirrored in the X axis. Once program 0001.fnc has finished the 4th axis will rotate 180 degrees and then begin to run program 0002.fnc. When program 0002.fnc is reached, both the X and Y coordinates are mirrored.



3.11 - Drilling the axle holes in the model

Provided that the completed car model has a flat reference on the body side (parts of the billet have been left unmachined), a pillar drill can be used to drill the two axle holes. Use a 4mm diameter drill.



Alternatively, you could place the 4mm drill into the CNC machine, place the model back into the jig and manually drill the holes. Ensure the drill is positioned in the spindle so it can machine completely through the full width of the model. Use “Jog Mode” to move the CNC machine head to align with the first axle marker position. Click the [M Codes] button and select “M03 - Spindle Forward” to start the spindle. Use the computer keyboard jog keys to drill the hole, then click the [M Codes] button and select “M05 - Spindle Stop” to switch off the spindle. Take care not to drill into the machine floor - use the Z co-ordinate display to gauge when the full width of the model has been drilled. Move to the second axle marker position and repeat the process.

Need further help?

You can contact Denford Customer Services for further help developing and machining your 3D solid models. Before contacting Denford for support, please read your hardware and software manuals and check the FAQ section on our website - www.denford.co.uk

When you request support, please be at your computer and CNC machine, with your hardware and software documentation to hand. To minimise delay, please be prepared to provide the following information:

- CNC Machine Serial Number (from the machine ID panel).
- Registered user's name / company name.
- The CNC machine control software name and version number.
- The wording of any error messages that appear on your computer screen, if applicable.
- A list of the steps that were taken to lead up to the problem.

Please note that on-site visits by our engineers may be chargeable.

Denford Contact Details:

Denford Customer Services, Birds Royd, Brighouse, West Yorkshire, HD6 1NB, UK.

Telephone: 01484 728000
Fax: 01484 728100
ISDN: 01484401157:01484401161

For international dialling from outside the UK: Add “44” to the number
and remove the first “0” from each city code.

E-mail: customerservices@denford.co.uk

Technical Support: Monday to Friday 8.30am - 4.30pm GMT

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

For Customer Services (Denford):
Telephone: 01484 722733
Fax: 01484 722160
ISDN: 01484401157:01484401161
E-mail: customerservices@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.

Contact Details (F1 in Schools):
F1 in Schools Limited,
22 Old Queen Street,
London,
SW1H 9HP,
UK.

Disclaimer:
Please note that due to nature of hardware and software developments, the specifications and features of the products featured in this guide can change without notice. The information contained in this guide is correct at the date of printing only - April, 2003. No liability can be accepted by Denford Limited or F1 in Schools Limited for loss, damage or injury caused by any errors in, or omissions from, the information supplied in this guide. Denford Limited reserves the right to alter any specifications and documentation without prior notice.

Language:
This guide is written using European English.

Questions and Comments:
Any questions and comments regarding this guide should be referred to the following e-mail address: customerservices@denford.co.uk
Alternatively, telephone Denford Customer Services on 01484 722733 and ask to speak to a member of our Technical Authoring Team.