



F1 TEAM IN SCHOOLS CAR MANUFACTURING GUIDE

D-Type

Solid Edge or Pro/DESKTOP 8



QuickCAM 3D



VR CNC Milling V5

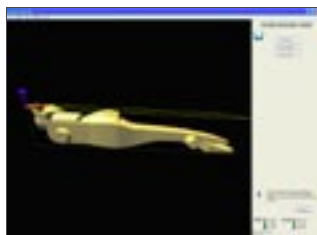


Table of contents

Introduction..... 3

Stage 1 - Using Solid Edge:..... 4

Stage 1 - Using Pro/DESKTOP:..... 8

Stage 2 - QuickCAM 3D:..... 12

Stage 3 - Using Denford VR CNC Milling: 30



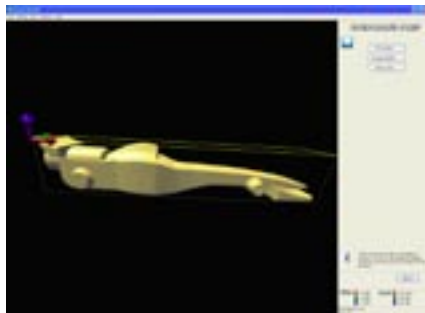
Introduction

This tutorial guide leads you through the process of converting a 3D model file describing the shape of an F1 in Schools, D-Type car into the CNC file used to manufacture the solid 3D car body.

In order to machine the solid 3D car body, you must create two separate CNC files. The first CNC file will be used to machine one complete side of the car body. The balsa blank will then be rotated 180° in the jig and the second CNC file used to manufacture the opposite side.



Stage 1:
Solid Edge or Pro/DESKTOP



Stage 2:
QuickCAM 3D



Stage 3:
Denford VR CNC Milling V5

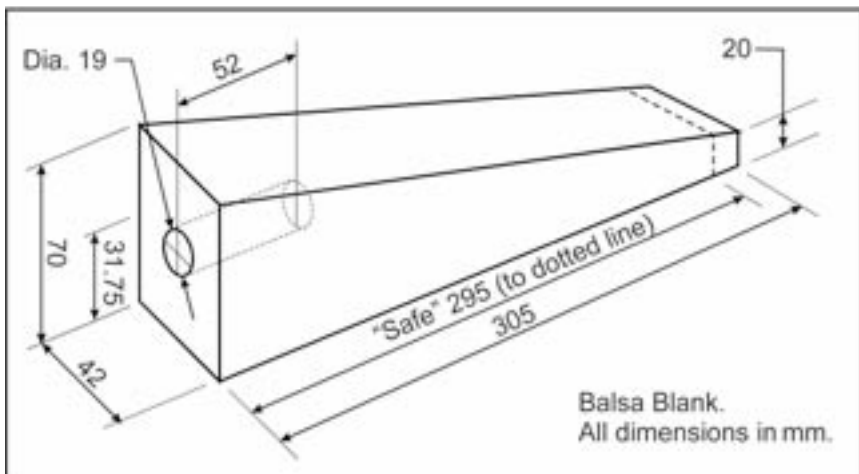


Stage 1 - Using Solid Edge

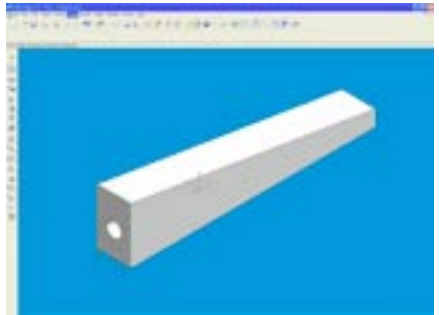
1.1 - Create your car design

Your car design is created using a CAD software package. This guide describes the process using Solid Edge 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.



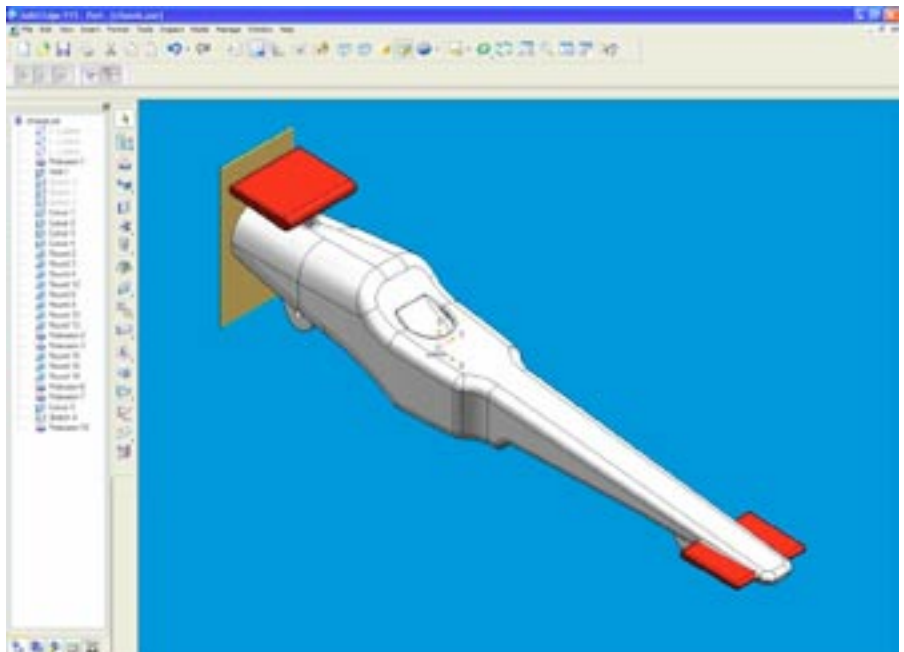
- 2) You can load a predrawn template, describing the exact shape and size of the balsa wood billet. Your car design is created by cutting into and removing sections from the template file. Using this method guarantees that your design will fit inside the balsa wood billet. If you need a copy of this starting template, visit the “downloads” section of the Denford website - www.denford.co.uk If you have QuickCAM 3D installed you can find the Balsa Billet template in 'My Documents\Denford\F1'



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

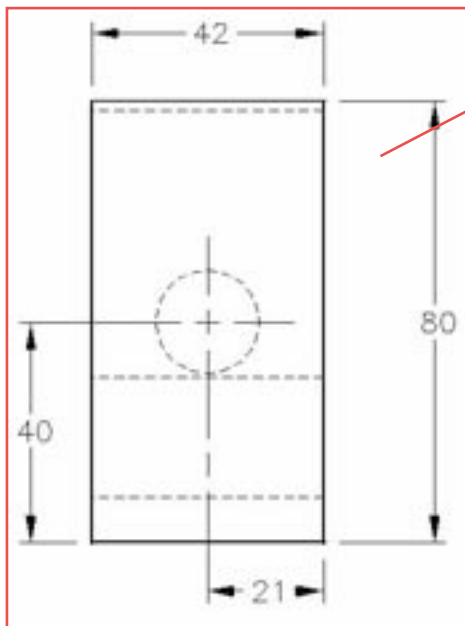
Before saving your car design as an STL file, you must add a small reference plate to the CO2 cartridge end of your 3D model. The reference plate is used to help set the datum in the correct place, making it easier to use QuickCAM in the next stage. Also, if you use the same reference plate on all your 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 "SE V15 Dtype_Balsa_Billet.par" available from Denford (website).

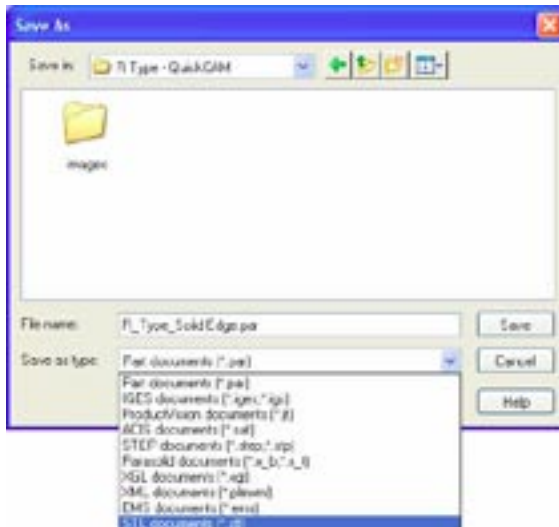
- 1) Select the back face of your car design (the flat face around the CO2 cartridge hole) and create a "protrusion" on the face of the object. (Your design must include a flat area around the hole at the back, this complies with F1 competition rules.)
- 2) Draw a rectangle around the car. Attach and define a height dimension of 80mm and a width dimension of 42mm to the rectangle.
- 3) Attach a dimension between the centre of the hole and one of the vertical sides of the rectangle, then redefine the dimension to 21mm. Attach a dimension between the centre of the hole and the bottom side of the rectangle, then redefine the dimension to 40mm.
- 4) 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.
- 5) Define the protrusion "Distance" to 1mm away from the car.
- 6) Name the protrusion "reference plate" and click [Finish].
- 7) Save a copy of the car design as "<your name> with reference plate".



1.3 - Save your car design as a Stereo Lithography File

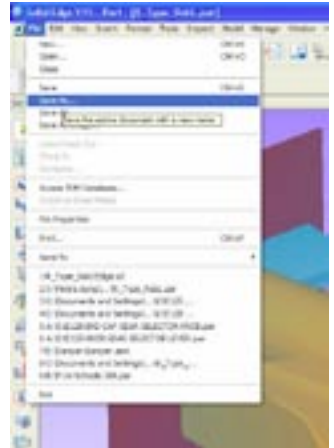
With your design loaded in Solid Edge, select 'File | Save as..'

Click on the drop-down button next to the 'Save in' dialogue box and choose a place to save your file, remember where you saved it to.



Click the **[Save]** button.

The STL model is now created. You can move on to the QuickCAM stage.



In the 'Save As' Window click on the drop-down button next to the 'Save as type' dialogue box.

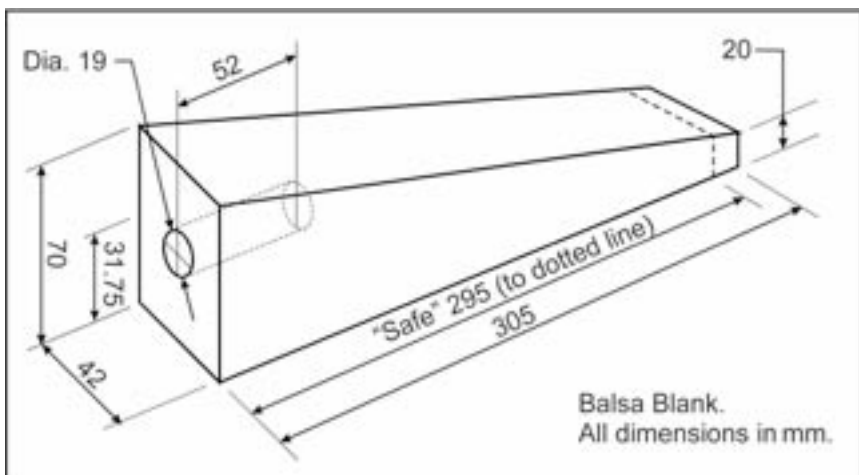
Choose the 'STL documents (*.stl)' option.

Stage 1 - Using Pro/DESKTOP

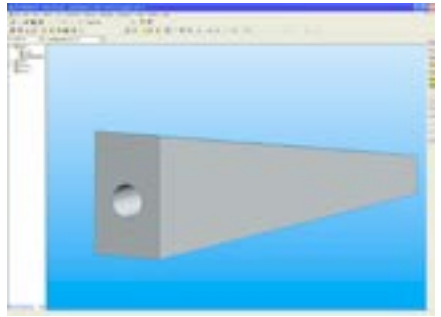
1.1 - Create your car design

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. 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.



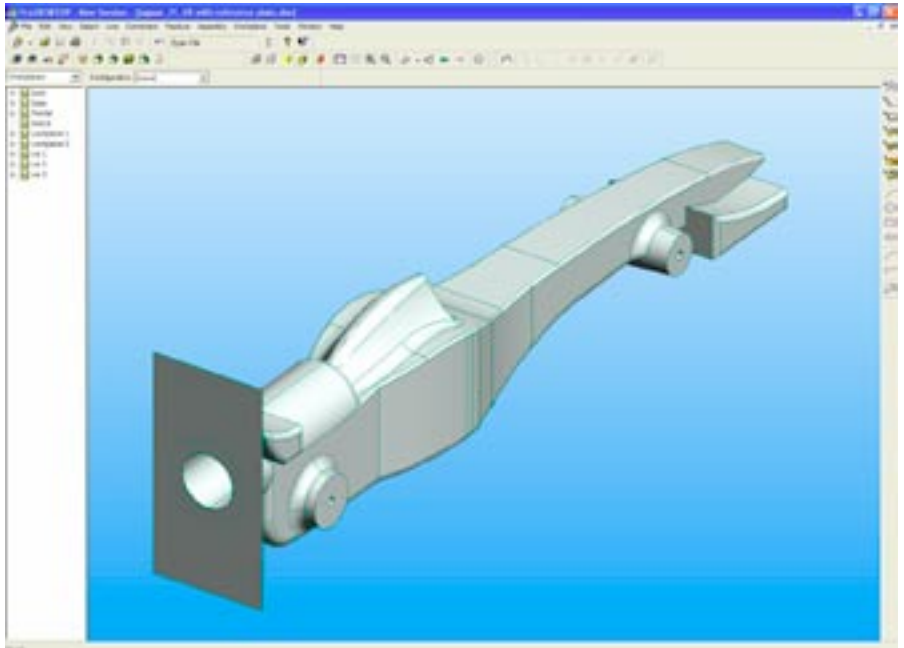
- 2) You can load a predrawn template, describing the exact shape and size of the balsa wood billet. Your car design is created by cutting into and removing sections from the template file. Using this method guarantees that your design will fit inside the balsa wood billet. If you need a copy of this starting template, visit the “downloads” section of the Denford website - www.denford.co.uk If you have QuickCAM 3D installed you can find the Balsa Billet template in 'My Documents\Denford\F1'



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

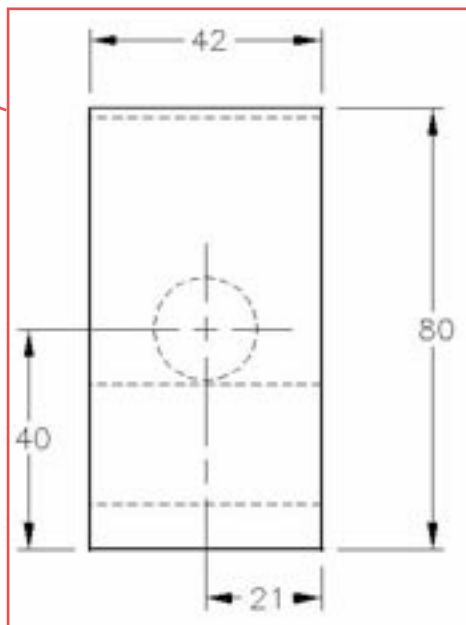
Before exporting your car design as an STL file, you must add a small reference plate to the CO2 cartridge end of your 3D model. The reference plate is used to help set the datum in the correct place, making it easier to use QuickCAM in the next stage. Also, if you use the same reference plate on all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

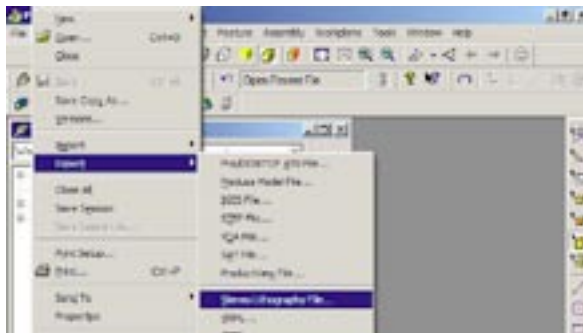
- 1) Select the back face of your car design (the flat face around the CO2 cartridge hole) and create a “New Workplane” on the plane of the object. Create a “New Sketch” called “reference plate”, then view directly onto this workplane.
- 2) Select a drawing tool, then left click in a blank area of the 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 "ASCII" option - this is useful if you intend saving your file on floppy disk.

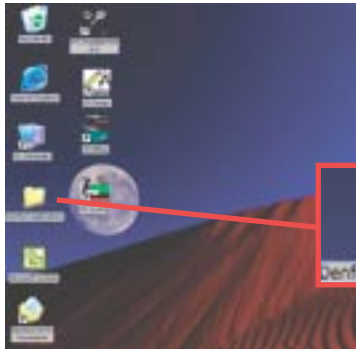


Stage 2 - QuickCAM 3D

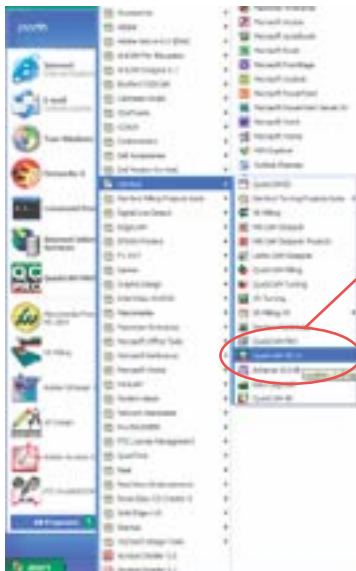
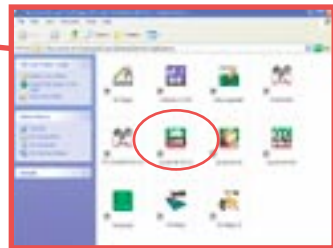
The Denford QuickCAM series are simple to operate, wizard based CAM software packages used to convert your Pro/DESKTOP STL file into a CNC program that can be run on a CNC milling machine or router.



2.1 - Start the QuickCAM software



To start the QuickCAM software open the 'Denford applications' folder on the desktop and double-click the "QuickCAM 3D V3" shortcut icon.

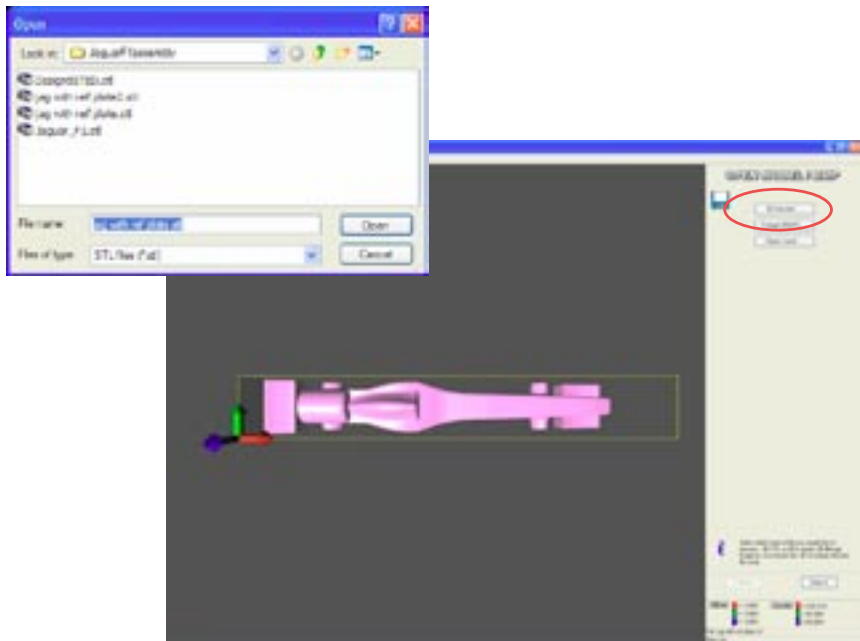


QuickCAM menu icon

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 V3" icon.

2.2 - Load your STL file

Click the **[3D Model..]** button. Browse for the 3D model that was created from the 3D modelling software 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.

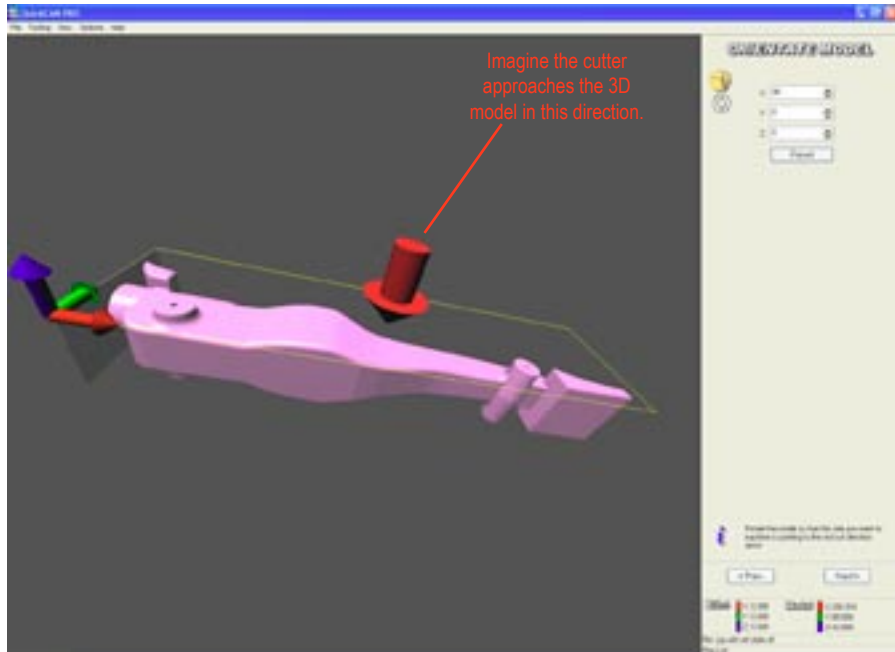
Click and hold the right mouse button whilst moving the mouse to zoom in or out.

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.

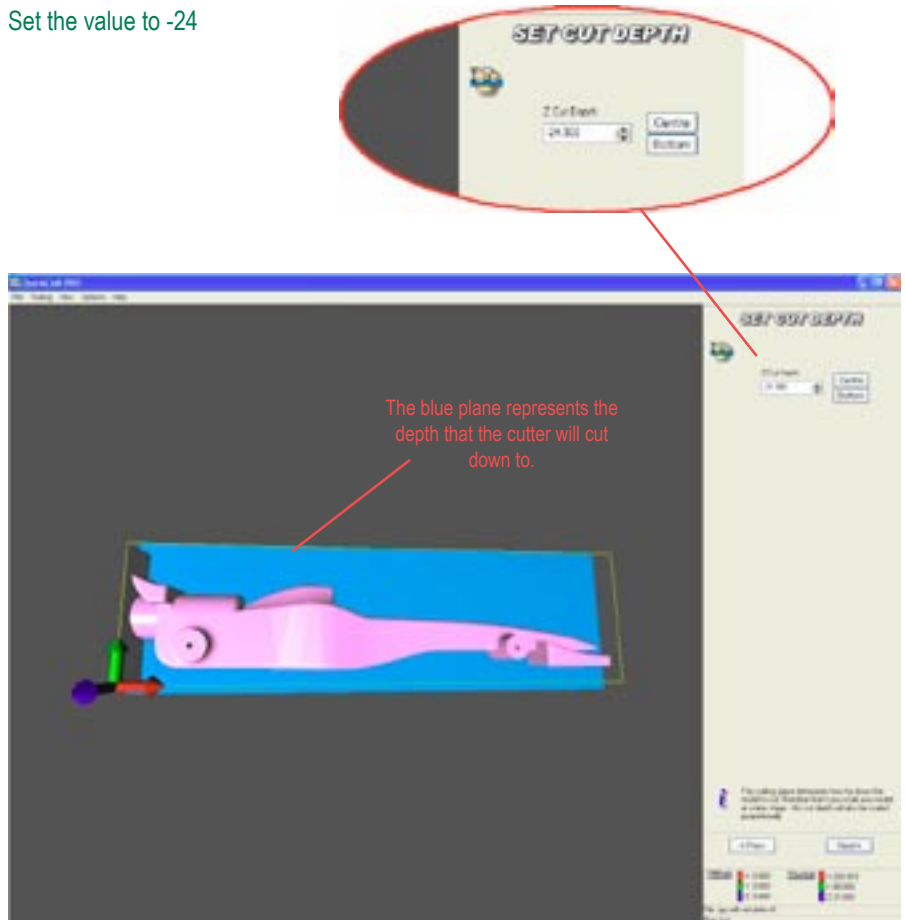
2.4 - Set cut depth

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

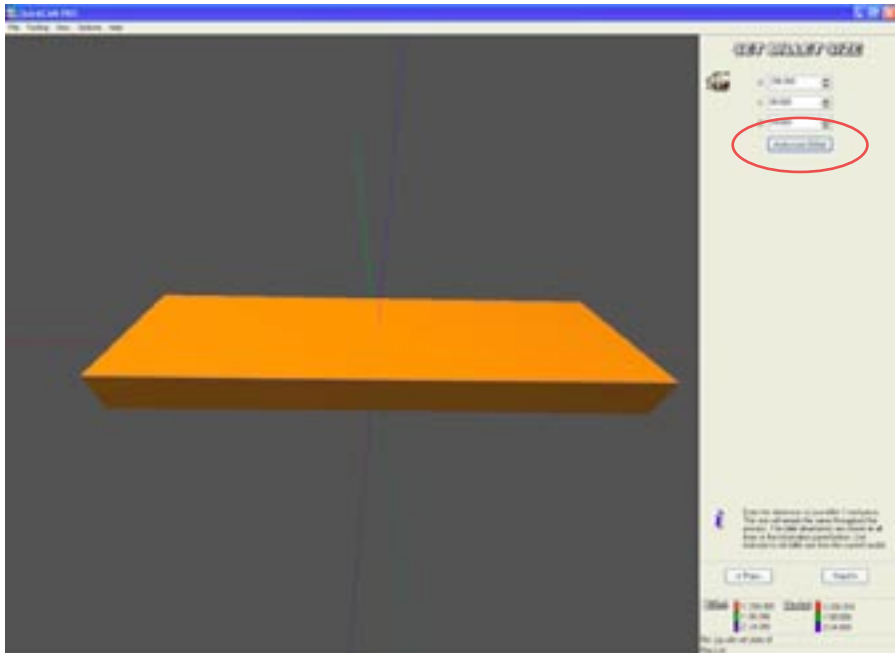


Click the **[Next>]** button to move to the next screen.

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.



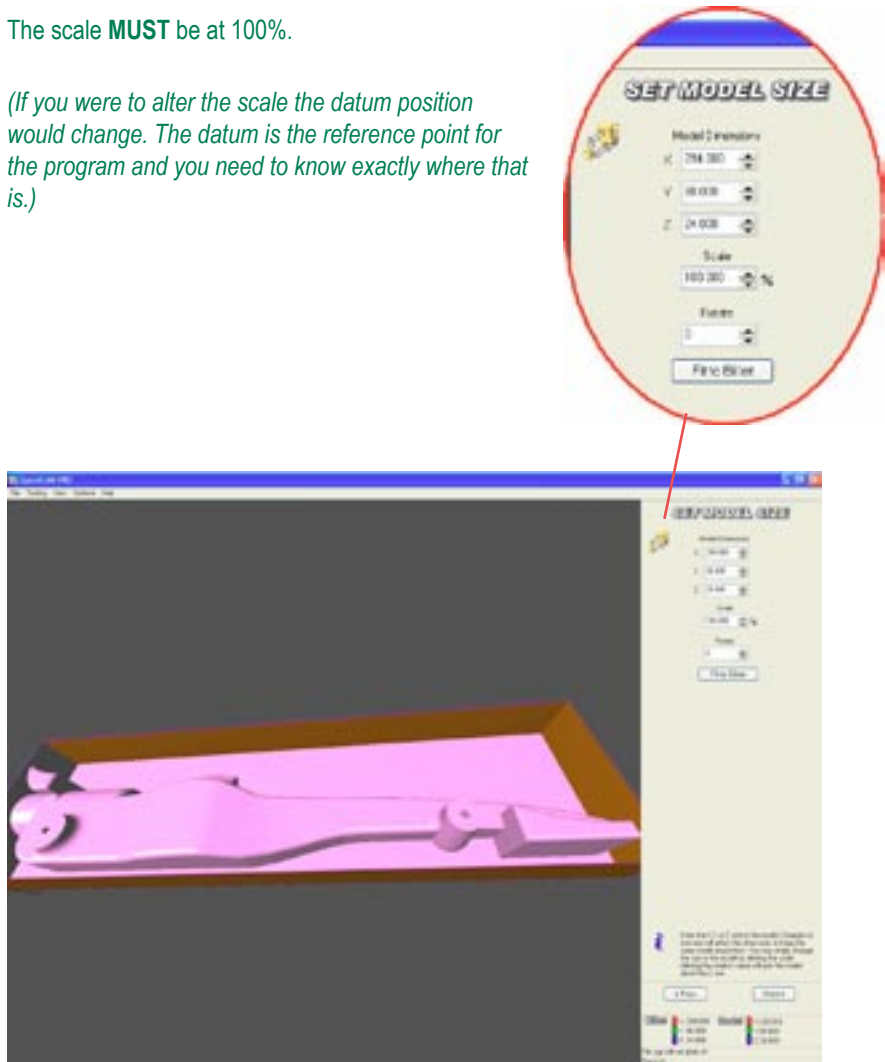
Click the **[Next>]** button to move to the next stage of the process.

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 reference point for the program and you need to know exactly where that is.)



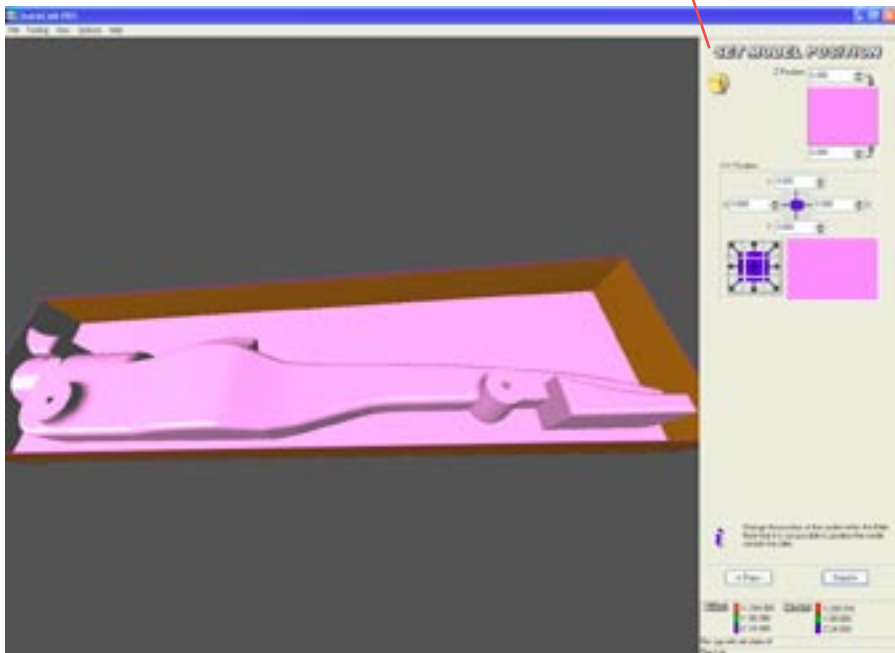
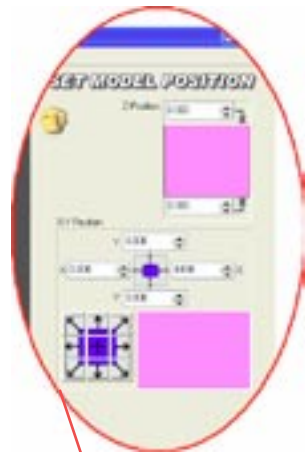
Click the **[Next>]** button to move to the next stage of the process.

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)



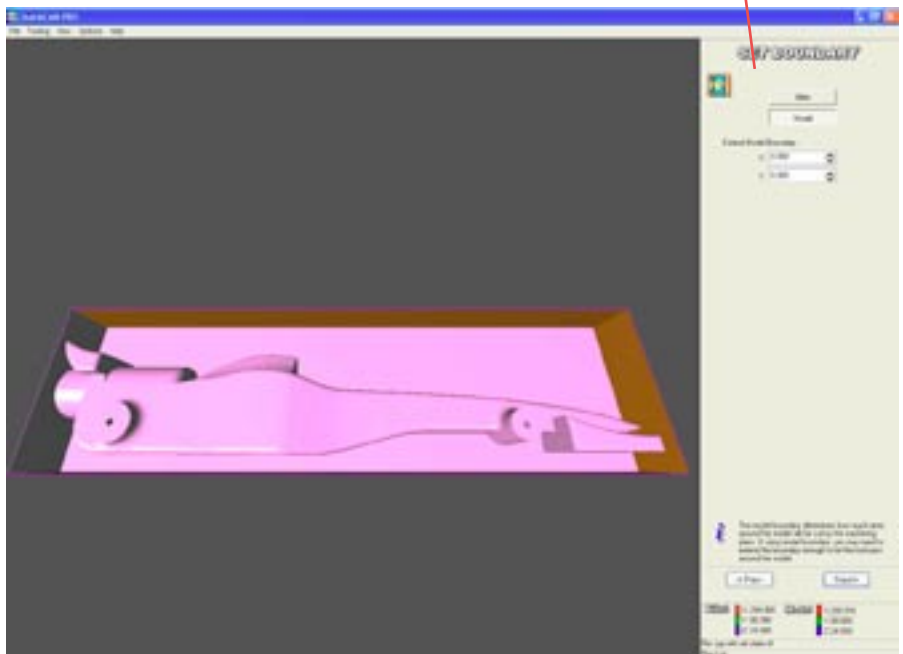
Click the **[Next>]** button to move to the next stage of the process.

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.

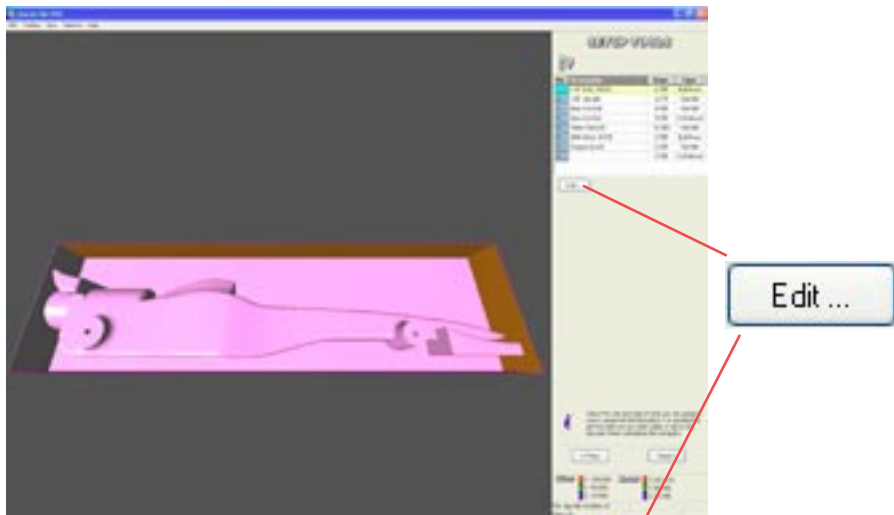


2.9 - Setup Tools

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

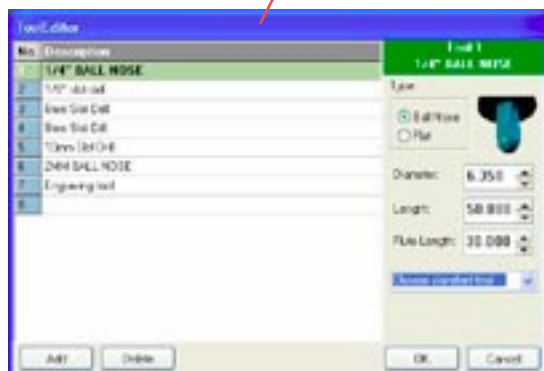
For the F1 Team in Schools car we recommend using the 1/4" Ball Nose Cutter which is supplied with every Denford CNC Router.

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.



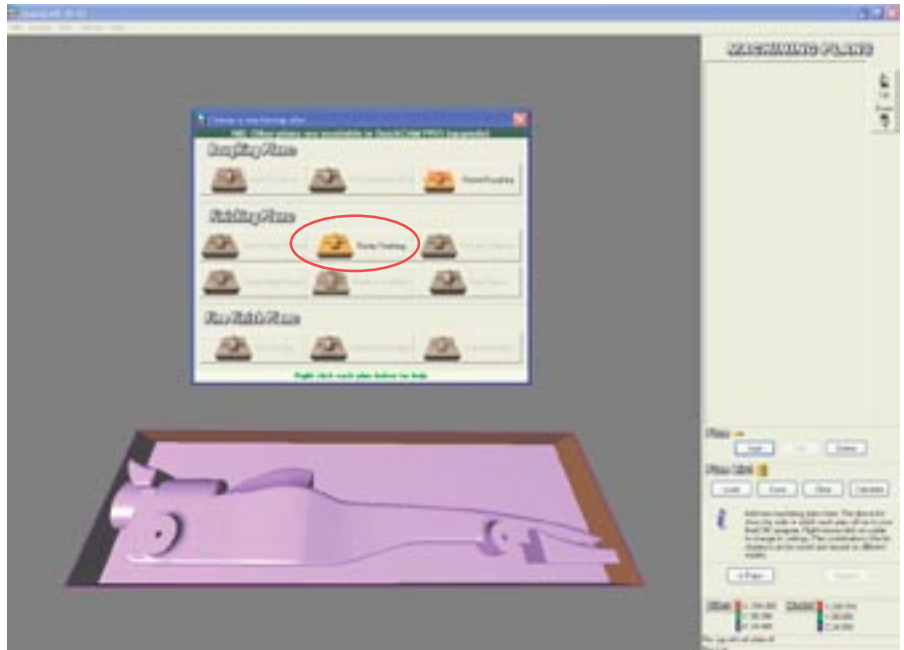
Choose the 1/4" Ball Nose Cutter from the list of standard tools and click **[OK]**.

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



2.10 - Machining Plans

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



The tool needed to machine the D-Type F1 cars is the 1/4" Long Series Ball Nose Cutter (has a 31mm flute length). This cutting tool is supplied as standard with a Denford CNC Router.

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 5000mm/min

- The Spindle Speed should be 23,000 rpm

- The 'Raster Angle' should be set at 90 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.

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

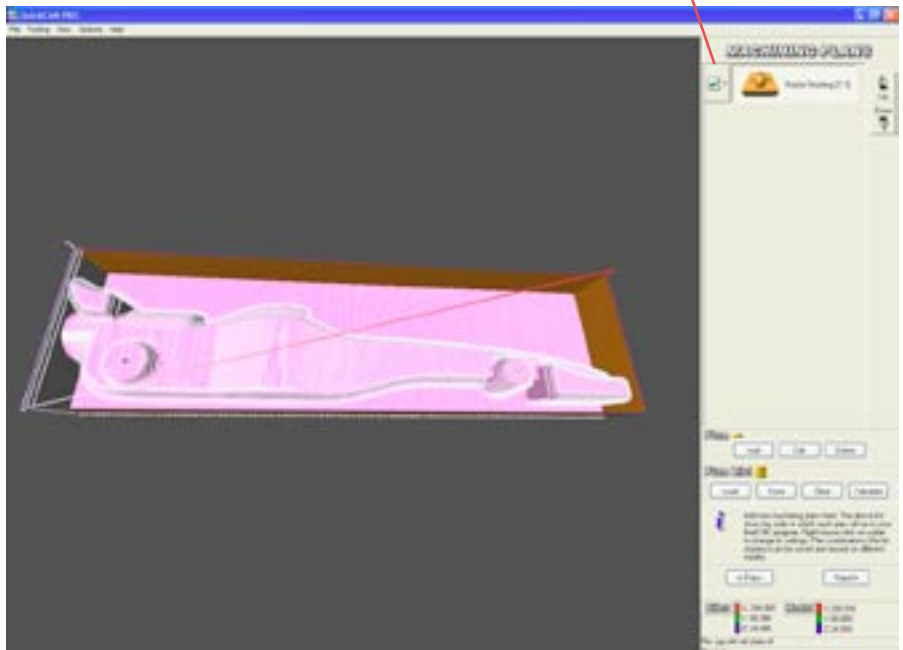
2.12 - Create the toolpath

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.



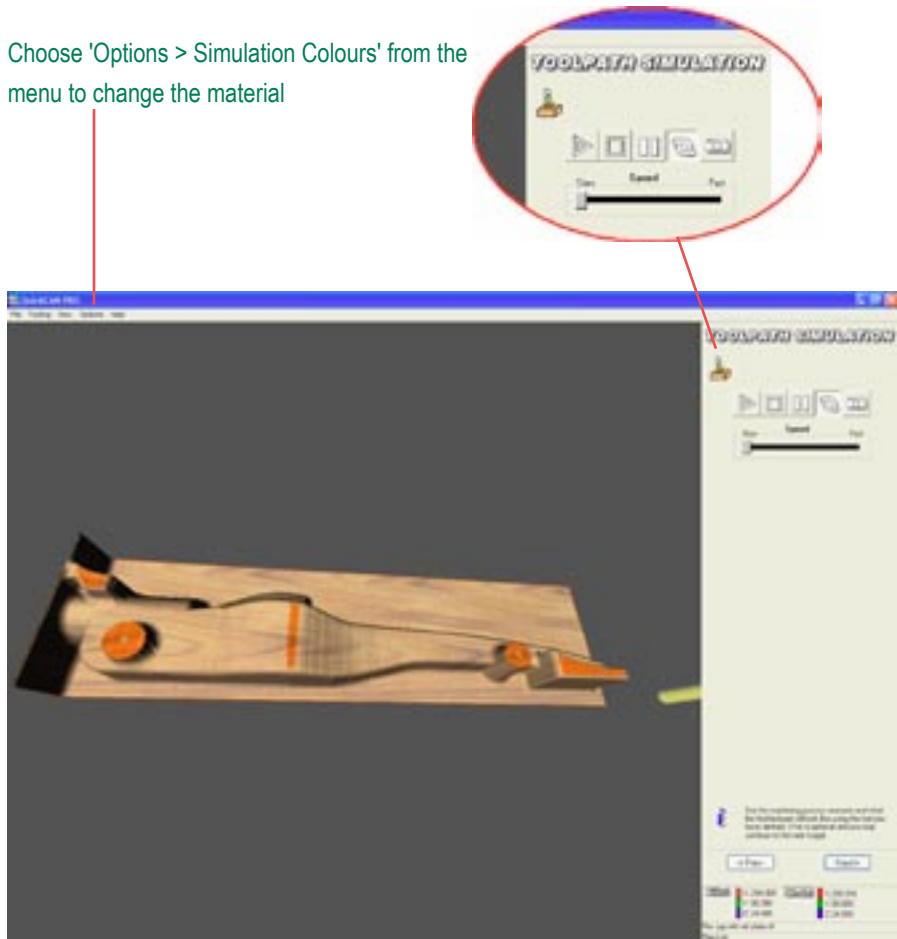
2.13 - Simulate the toolpath

On this screen the toolpaths can be simulated. Press the Play button 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

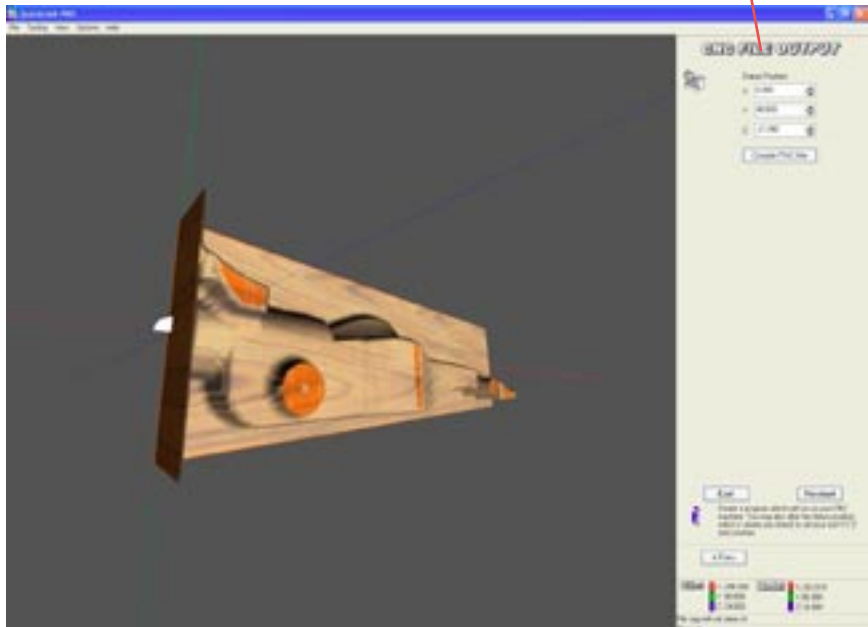
This is the final screen in QuickCAM and is where the CNC file is saved. Before we click the [Create FNC file] button, the 'datum' position will be changed. The 'datum' is the reference point for the program. We will move the datum to the centre of the hole in the Balsa Wood blank. Drawing the reference plate in the 3D CAD software 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]**

If you have VR Milling installed on your PC it will start to run automatically and the FNC file will be loaded in ready for the next stage.




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

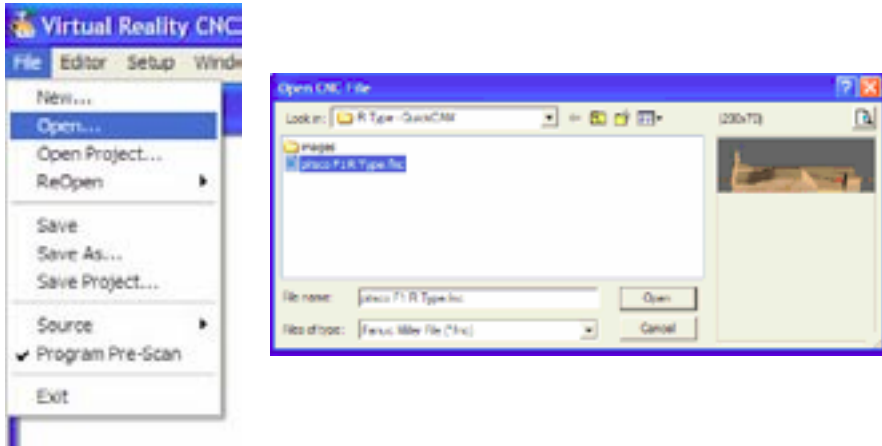


Legacy machines: These are older machines that are no longer in production but are still compatible with VR CNC Milling V5 software. Click the [Legacy Machines] button to list these types of machine.

3.3 - Load your CNC file.

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

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



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

Note:

When a program is opened you may see 2 messages:

Colour Formatting

The Colour formatting option will format the CNC file, allowing groups of program words to be highlighted by colour and/or style, if you have a large file this may take a few minutes, turning this option off will open the file more quickly.

Program Pre-scan

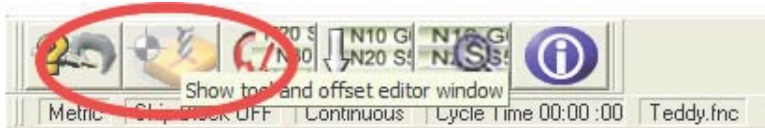
When this facility is activated, any subsequent CNC programs are scanned before they are loaded into the editor. During the scan they:

- are checked for syntax errors (invalid characters, usually typing errors)
- are checked for invalid codes (for a list of valid G codes and M codes refer to the 'CNC Programming' help section)



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:

- 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.
- 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”



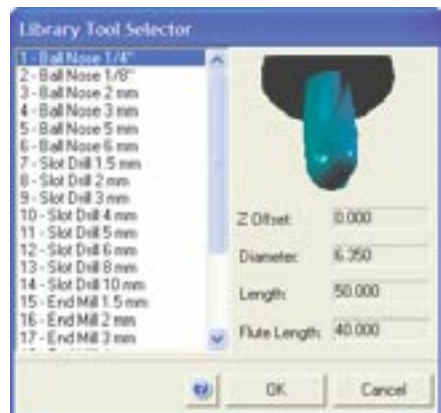
“Insert Library Tool”

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 - Running a simulation.

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



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

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

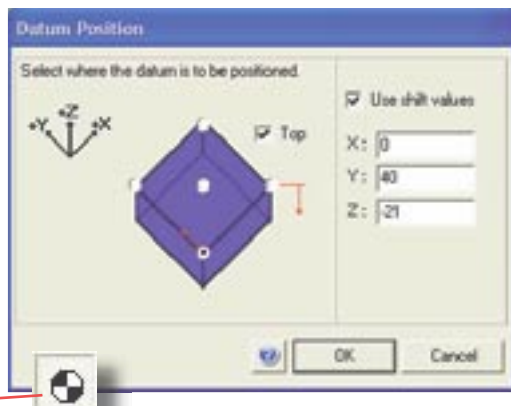
Under the “Simulation” menu choose “Billet Materials ...” to change the appearance of the simulation including the material with which the model is rendered.

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

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

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

Enter the values X = 0, Y = 40 and Z = -21, these are the same values used to shift the datum in the QuickCAM software. Press the Datum button to display the datum in the simulation window.

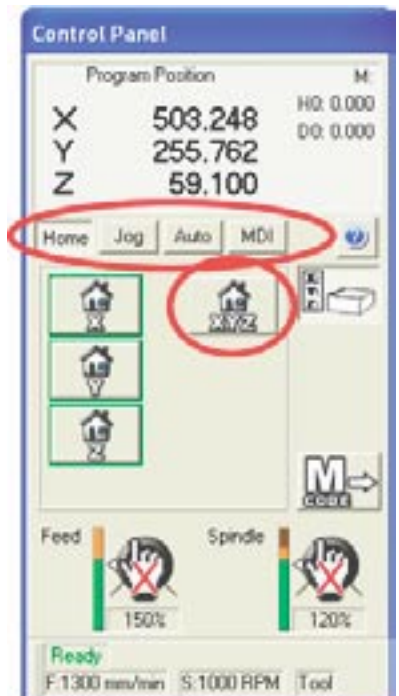


3.6 – Connect to, and Home the CNC machine



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.

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

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

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

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

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

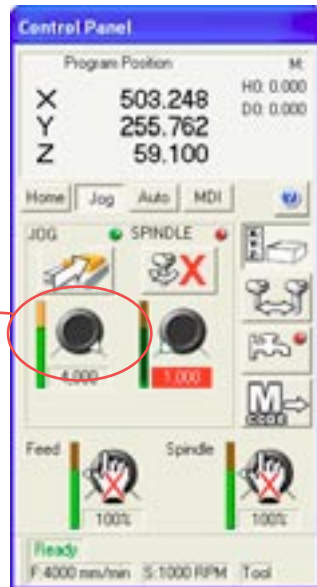


To change the position of the machine head incrementally, click the Jog button until the image changes from a single arrow to three small, stepped arrows, signifying Jog Step Mode.

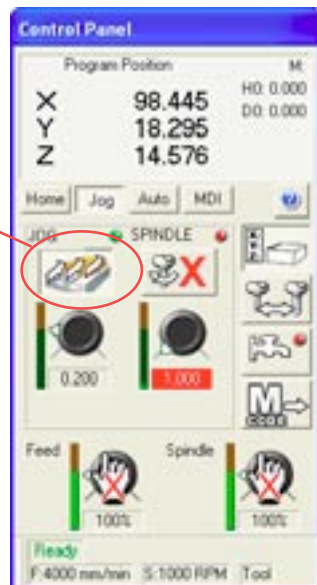
Click and drag the Jog Feed control knob to adjust the increment. When you press the cursor keys the cutter will move by the amount set.

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



"Jog Continuous"



"Jog Step"

3.8 - Fit the jig used to hold the billet inside the CNC machine.

The easiest method of holding the balsa wood billet in your CNC machine is by using a jig, or fixture. Once set into position, the jig allows billets to be accurately placed into the CNC machine offering the potential for small scale production of CO2 cars.

The jig, fitted to a Denford Microrouter in the photograph right, can be supplied by Denford Limited and is purpose-built for manufacturing CO2 cars.

The circular fork on the left of the jig fits into the CO2 cartridge hole on the back face of the balsa wood billet.

The taper end of the billet is tightened against the bracket on the right side of the jig.

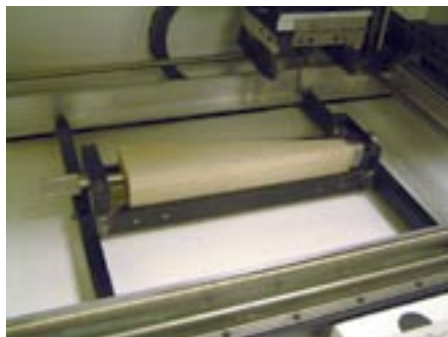


The CNC file describing the right side of the model is then machined, before revolving the billet 180° inside the jig. The CNC file describing the left side is loaded and then machined to complete the car.

It is essential that whatever jig you decide to use is positioned and fixed square (ie, at 90°) to the machine axes. When fitting and adjusting the position of the jig, use an engineers square with any available reference edges. Since the balsa wood billet is revolved 180° inside the jig, any inaccuracies will be multiplied by two.

Depending on the size of the working area in your CNC machine, it may be possible to position the jig so the cutter can never hit the right side bracket (ie, the bracket lies outside the effective movement area of the cutting tool).

Fit the balsa wood billet in the jig as shown.



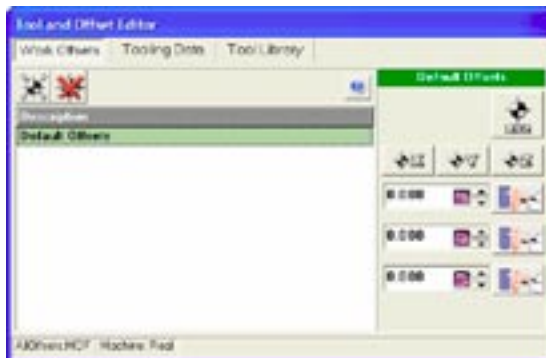
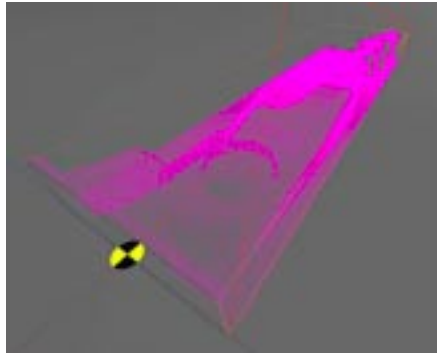
3.9 – Set the work offsets

What are offsets?

Offsets are the distances the cutter needs to travel, from its 'Home' position to the point from which the program starts in X Y & Z.

We are now going to define an offset for the centreline of the location fork that is part of the jig and is located in 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"



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.



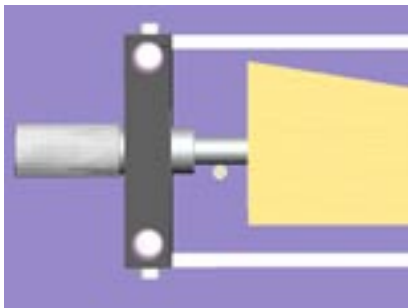
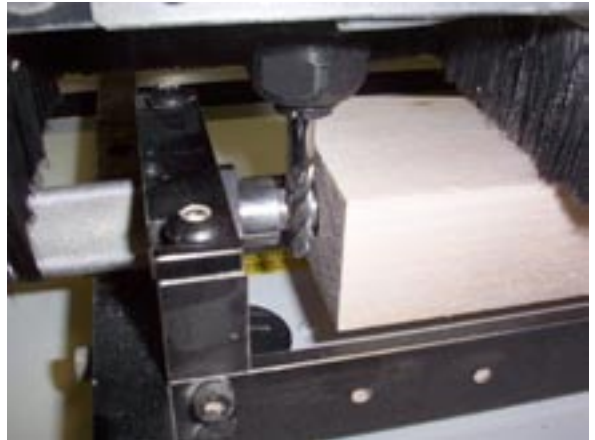
"New work offsets"

< Activate

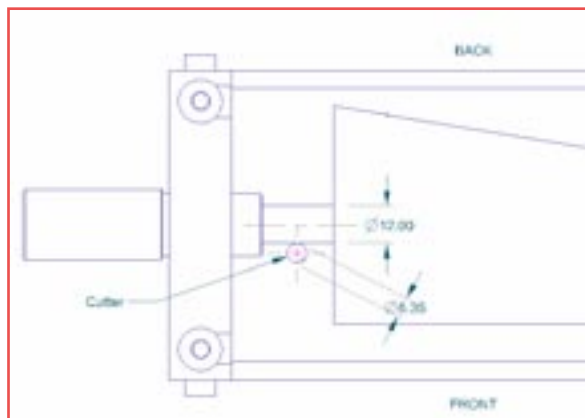
"activate offset"

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

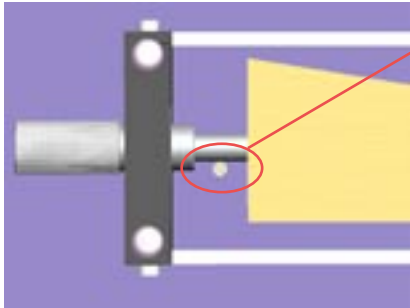
In 'jog continuous mode', move the tool a few millimetres away from the fork. We are going to touch onto the fork at its smallest diameter. (With the jig shown here it is 12mm, but other varieties may have a different diameter.)



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



3.9 - Setting the Y axis offset

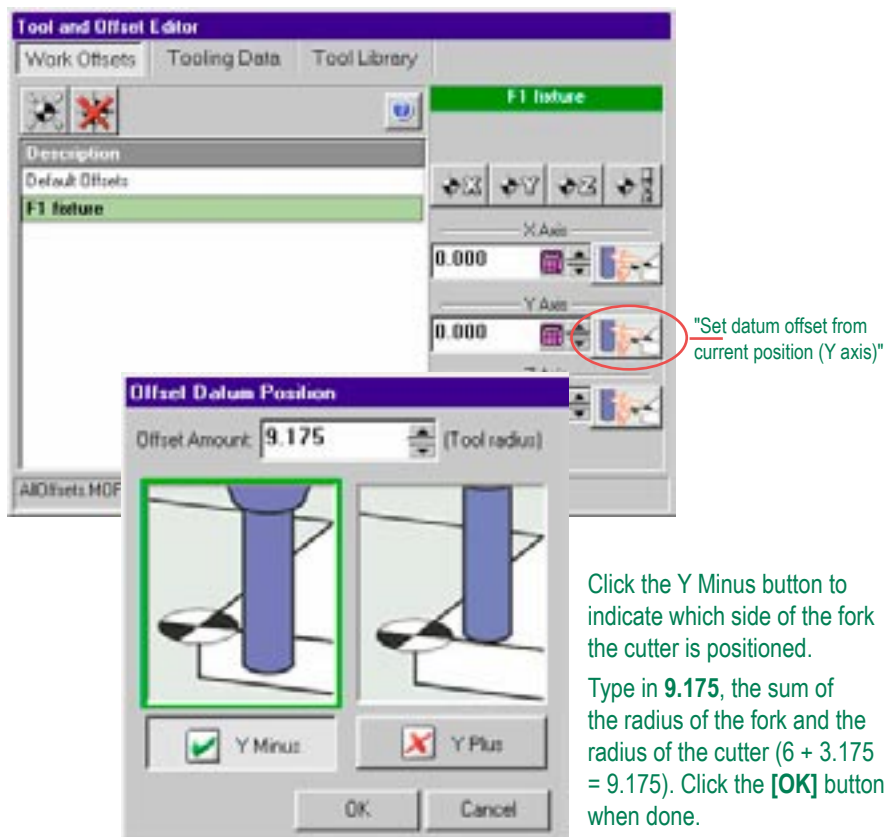


With the cutter in this position we will find the centre of the fork.

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

The radius of the fork is 6mm and the radius of the cutter is 3.175mm.
($6 + 3.175 = 9.175\text{mm}$).

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.



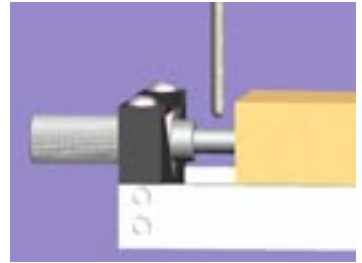
Click the Y Minus button to indicate which side of the fork the cutter is positioned.

Type in **9.175**, the sum of the radius of the fork and the radius of the cutter ($6 + 3.175 = 9.175$). 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 fork. Use the 'Page Up' key to lift the tool above the fork, ensure that the tool is above the fork before proceeding to the next step.

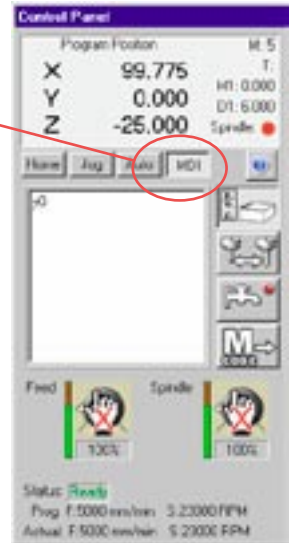


Click on the MDI tab in the control panel and type Y0 (zero not the letter O)

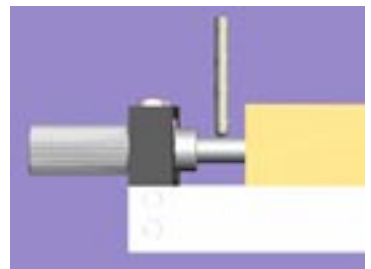
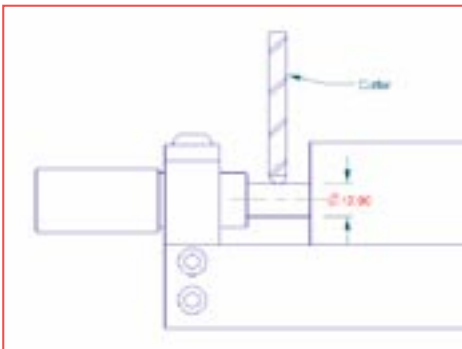
Press Play on the file control.



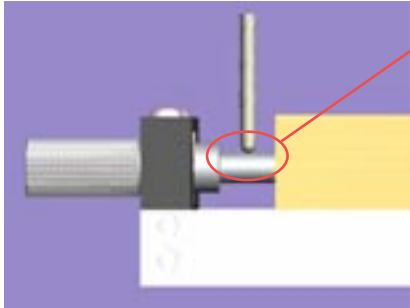
The cutter will move to the Y position set previously. This should be directly over the centre of the location fork, if not the Y offset is incorrect, repeat the previous operation.



Using the jog keys in 'jog step mode', bring the nose of the cutter down so it just touches the section of the fork where the diameter is 12mm as shown.



3.9 - Setting the Z axis offset

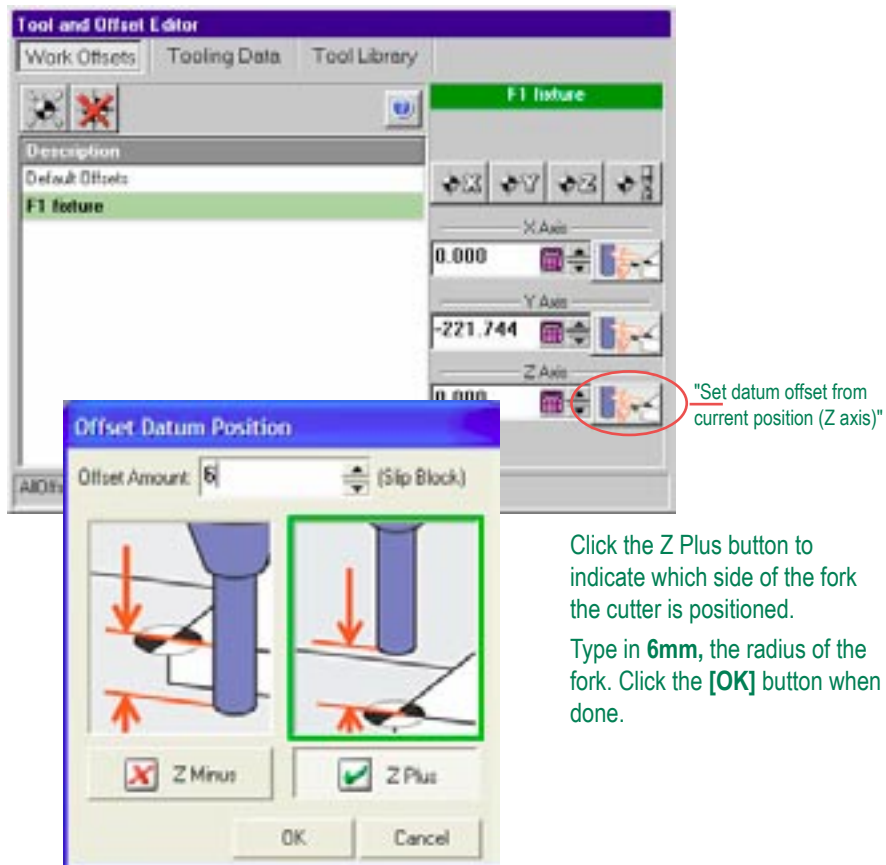


With the cutter in this position we will find the centre of the fork.

To find the centre of the fork we are going to include the radius of the fork. We will tell the machine to travel an extra - 6mm

The radius of the fork is 6mm.

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.



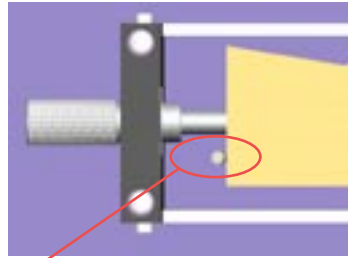
Click the Z Plus button to indicate which side of the fork the cutter is positioned.

Type in **6mm**, the radius of the fork. 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.

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

Click the X Minus 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

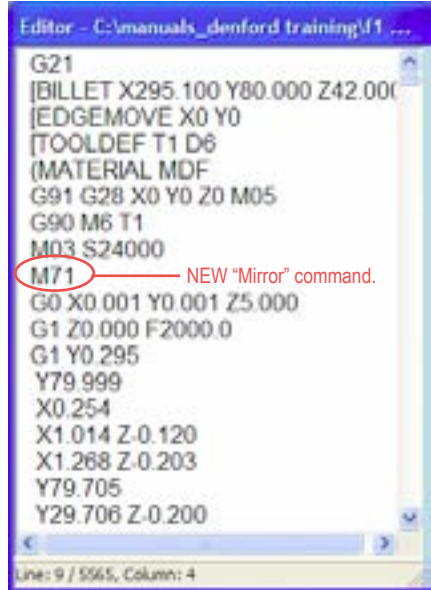
Click the [Turbo Mode] button to switch Turbo Mode on. This can be done at any time, even when the program is running. The 'turbo mode' feature has been developed to reduce the machining times of large 3D programs and complex 2D programs. For larger programs E.G. more than 100 lines, turbo mode on will 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.

3.11 - Creating the CNC file to machine the opposite side

Provided that your car design is symmetrical, the CNC file used to machine the right side of the body can also be used to machine the left side. In order to do this, you must manually edit the original CNC program, inserting an extra line that activates the “mirror” command.

Find the program line referring to the spindle speed. In the example screenshot right, this is the line “M03 S24000”. Remember that the “S” value depends on the type of CNC machine being used, so this value may be different in your program.

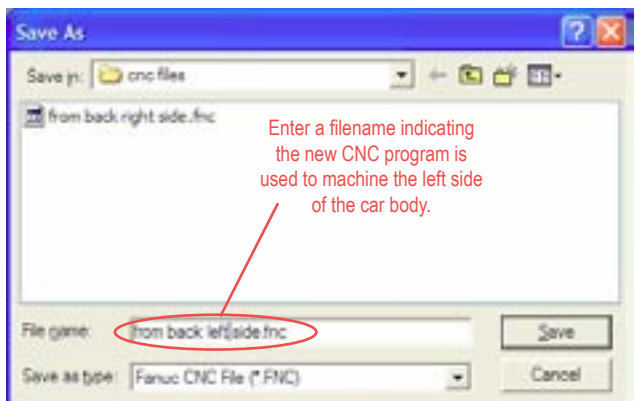
Click the cursor at the end of this line, then press **[Enter]** on your computer keyboard - a new program line is created. Type **M71** - this is the code defining the “mirror in the Y axis” command.



```
G21
[BILLET X295.100 Y80.000 Z42.000
[EDGEMOVE X0 Y0
[TOOLDEF T1 D6
[MATERIAL MDF
G91 G28 X0 Y0 Z0 M05
G90 M6 T1
M03 S24000
M71
G0 X0.001 Y0.001 Z5.000
G1 Z0.000 F2000.0
G1 Y0.295
Y79.999
X0.254
X1.014 Z-0.120
X1.268 Z-0.203
Y79.705
Y29.706 Z-0.200
```

Click the “File” menu and select the “Save As” option. DO NOT click “save” by mistake, since this will overwrite your original CNC file!

Enter a name describing the outcome of your new CNC file - in our example we have used “from back left side”.



3.13 - 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.

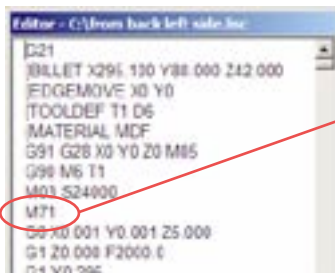
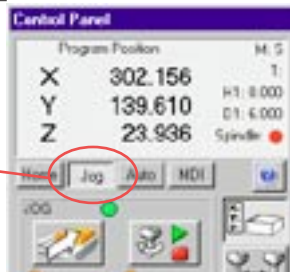
3.12 - Machining the 'Left' side of the car body



Physically rotate the Balsa Wood in the fixture by 180 degrees.



Left click on "Jog" in order to enable the menus needed to change the CNC program.

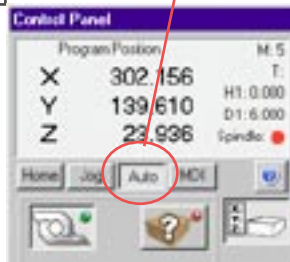


You must now make sure you have opened the modified file with the M71 added.

Left click on "Auto" to enter auto mode.



Using the file control press Stop, Rewind and finally Play to begin machining the program.



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 728000
Fax: 01484 728100
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. No part of this manual or its accompanying documents may be reproduced or transmitted in any form or by any means, electronic or mechanical, for any purpose, without the express written permission of Denford Limited. Copyright Denford Limited - Version 04-03. MiniCAM is a registered trademark of Delcam. Pro/DESKTOP is a registered trademark of PTC. All other brands and products are trademarks or registered trademarks of their respective companies. All rights reserved.

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 728000 and ask to speak to a member of our Technical Authoring Team.

