

F1 IN SCHOOLS CAR MANUFACTURING GUIDE R-Type

Solid Works / Solid Edge / Pro/DESKTOP 8



QuickCAM 3D or QuickCAM Pro



VR CNC Milling V5

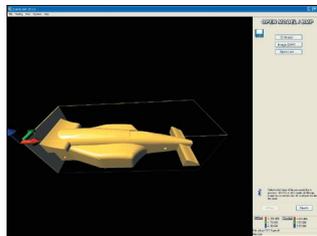
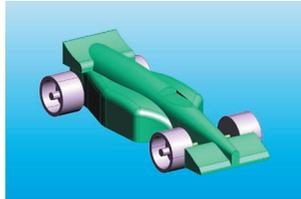


Table of contents

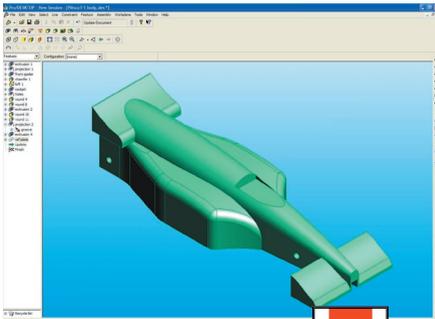
Introduction.....	3
Stage 1 - Create your car design.....	4
Adding a reference plate to your finished car design:	5
Using Solid Works:.....	6
Using Solid Edge:	8
Using Pro/DESKTOP:	10
Stage 2 - QuickCAM 3D:.....	12
Stage 3 - Using Denford VR CNC Milling:	30



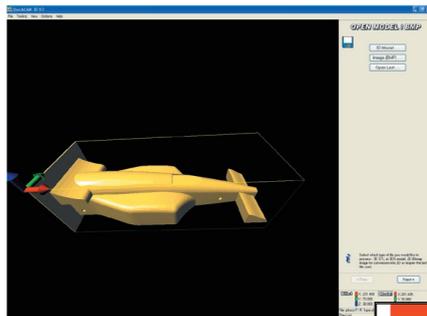
Introduction

This tutorial guide leads you through the process of converting a 3D model file describing the shape of an F1 in Schools, R-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 at least 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. Some designs may need to be manufactured from 3 or 4 sides to produce all the detail on the design



Stage 1:
Solid Works / Solid Edge /
Pro/DESKTOP



Stage 2:
QuickCAM



Stage 3:
Denford VR CNC Milling V5

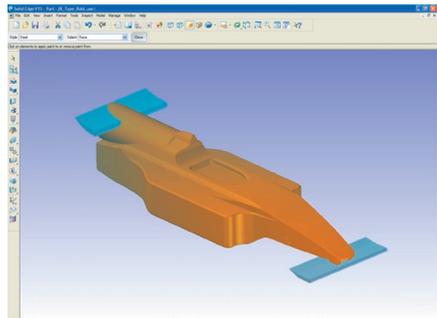


Stage 1 - Create your car design

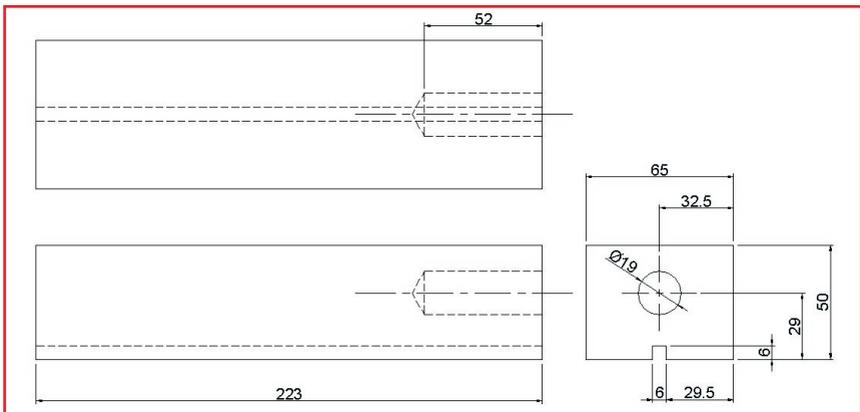
1.1 - Create your car design

Your car design is created using a CAD software package. There are two basic ways to create your design:

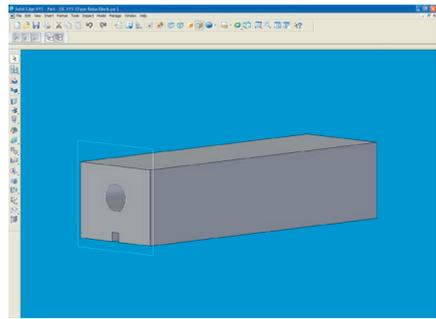
- 1) You can create your car design from a blank starting screen. If you use this method, you must make sure your final 3D model can fit inside the dimensions of the balsa wood billet. If you make your car design larger than the balsa wood, you may end up machining the jig used to hold the billet in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



Although the balsa wood billet is 223mm long, the car must be no longer than 210mm. The excess material is used for holding the billet in the machine. If the car is made longer than 210mm there is a possibility of machining into the jig that holds the balsa wood billet.



- 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

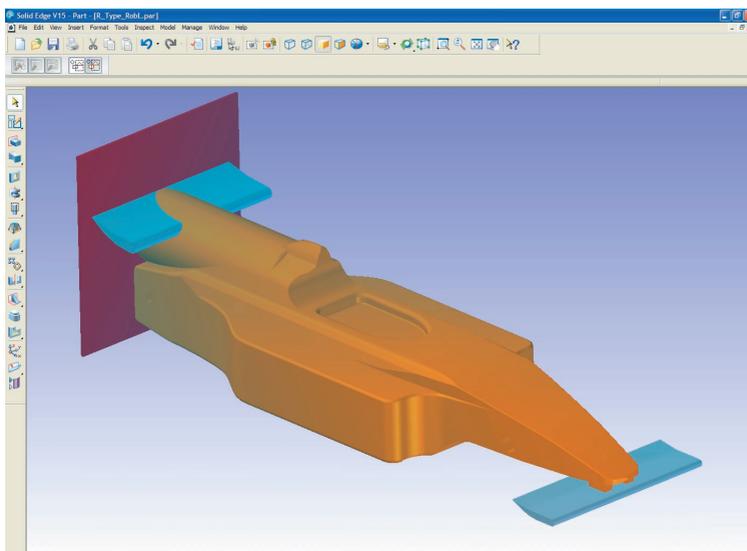


QuickCAM installed you can find the Balsa Billet template in 'My Documents\Denford\F1' (The templates for Solid Works refer to the www.f1inschools.com)

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.

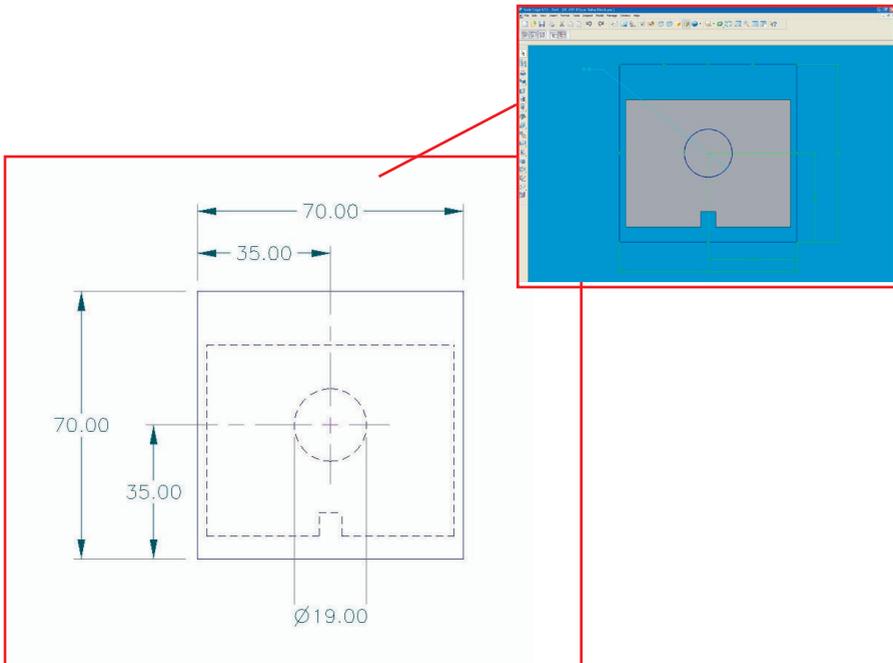
Add a reference plate to your finished car design

Before saving your car design as an STL file, you must add a 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 reference plate (Solid Works):

- 1) Create an “Extruded Boss/Base” feature and select the back face of your car design (the flat face around the CO2 cartridge hole). (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 70mm and a width dimension of 70mm 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 35mm. Attach a dimension between the centre of the hole and the bottom side of the rectangle, then redefine the dimension to 35mm.
- 4) Define the thickness of the extrusion as 1mm in a direction away from the car and name the extrusion “reference plate”.
- 5) Save a copy of the car design as “<your name> with reference plate”.



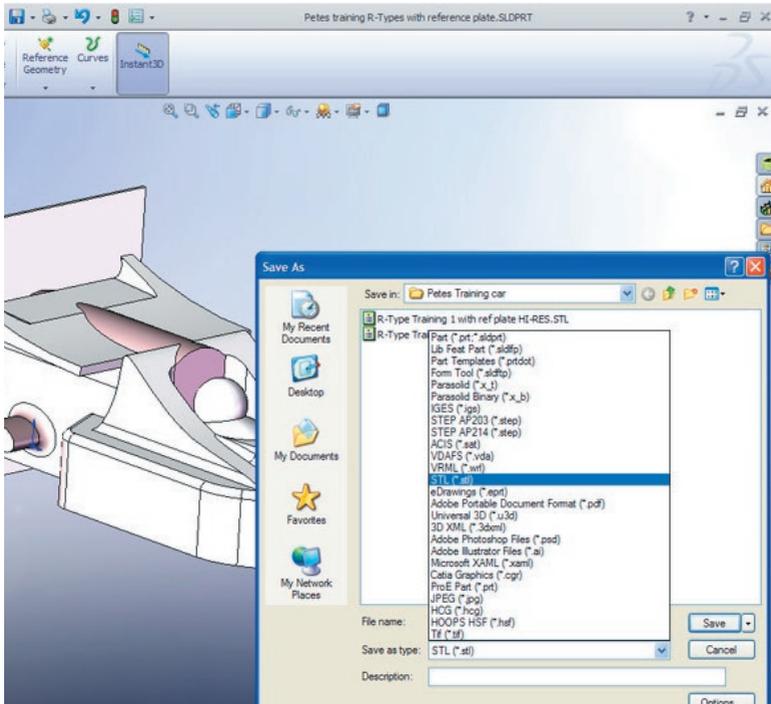
Save the design as a Stereo Lithography File (Solid Works)

From the drop down menus select 'File | Save as...'

Click on the drop-down menu bar labelled 'Save as type' and select 'STL'.

Important

The 'Student Design Kit' (SDK) version of Solid Works does not have the feature to save as an 'STL' file. If you have designed your car using the SDK, you must load your design into a version of Solid Works that can save as an STL file E.G. Solid Works - Student Edition or Solid Works - Education Edition.



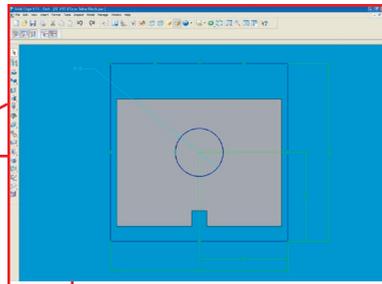
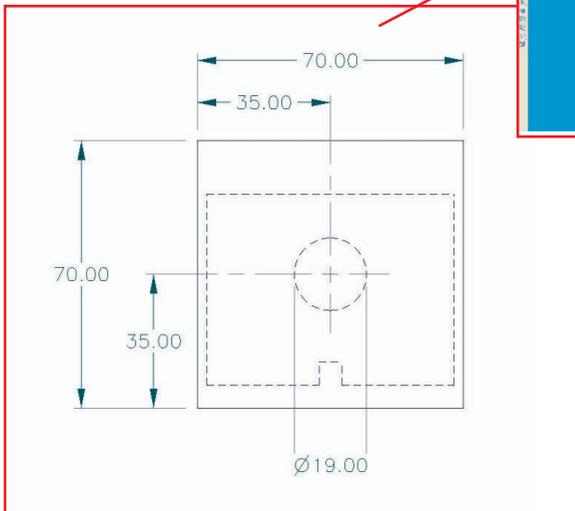
Click the [Save] button.

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

Creating the reference plate (Solid Edge):

(Note: The sketch is already created on the file "SE V15RType Balsa_Billet.par" available from Denford (website).

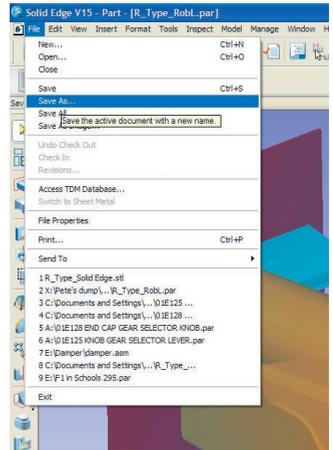
- 1) Select the back face of your car design (the flat face around the CO2 cartridge hole) and create a "protrusion" on the face of the object. (Your design must include a flat area around the hole at the back, this complies with F1 competition rules.)
- 2) Draw a rectangle around the car. Attach and define a height dimension of 70mm and a width dimension of 70mm 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 35mm. Attach a dimension between the centre of the hole and the bottom side of the rectangle, then redefine the dimension to 35mm.
- 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 protusion "reference plate" and click [Finish].
- 7) Save a copy of the car design as "<your name> with reference plate".



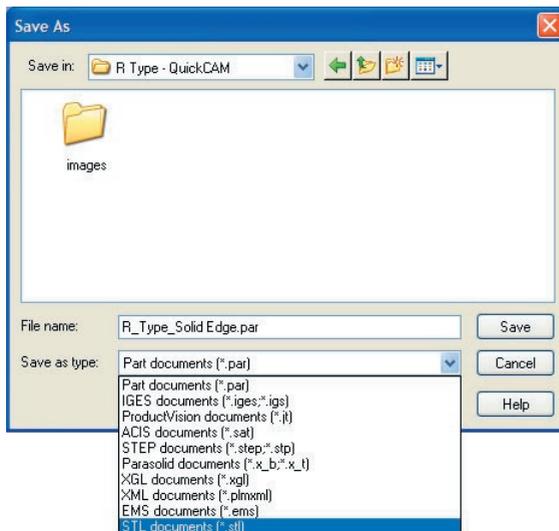
Save the design as a Stereo Lithography File (Solid Edge)

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.

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



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.

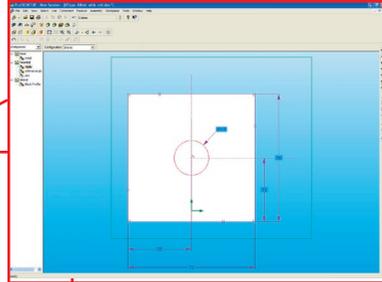
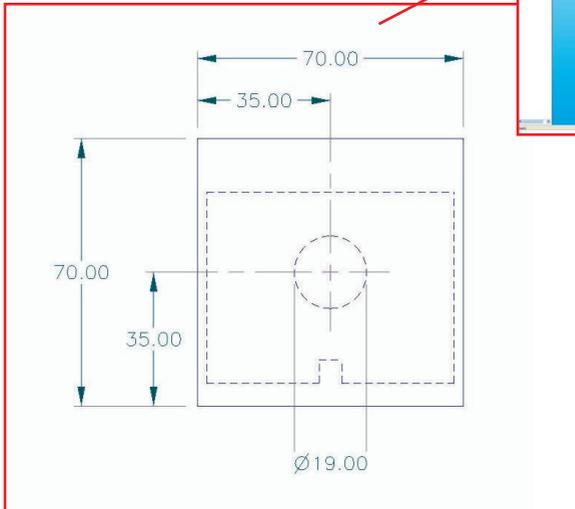
Click the [Save] button.

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

Creating a reference plate (Pro/DESKTOP)

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

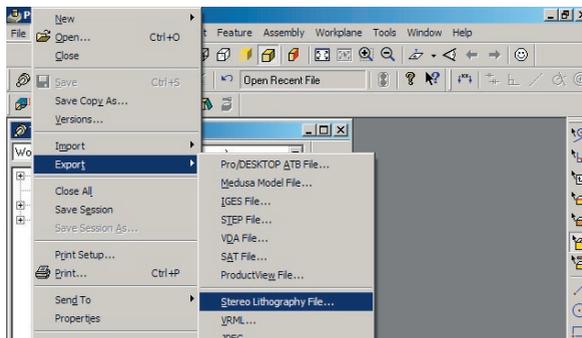
- 1) Select the back face of your car design (the flat face around the CO2 cartridge hole) and create a “New Sketch” on the face of the object. Create a “New Sketch” called “reference plate”, then view directly onto the workplane.
- 2) Select a drawing tool, then left click in a blank area of the screen. 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 70mm and a width dimension of 70mm 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 35mm. Attach a dimension between the horizontal construction line and the bottom side of the rectangle, then redefine the dimension to 35mm.
- 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.

Exporting your design as a Stereo Lithography File (Pro/DESKTOP)

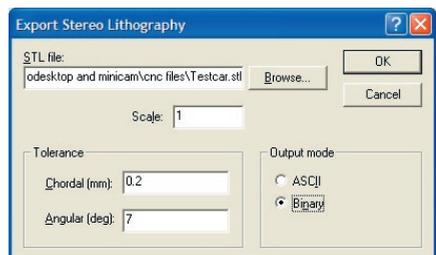
With your design loaded in Pro/DESKTOP, select 'File | Export | Stereo Lithography File...'



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

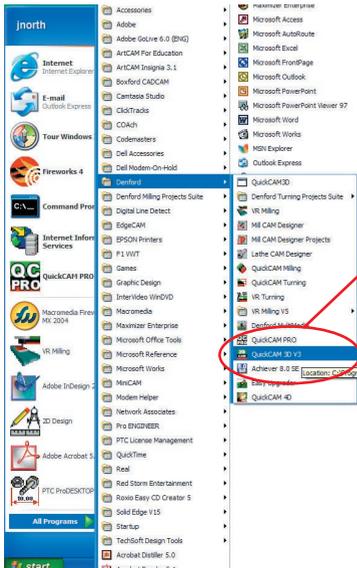
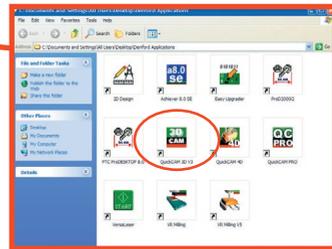
The Denford QuickCAM series are simple to operate, wizard based CAM software packages used to convert your STL (*.stl) file into a CNC program that can be run on a CNC milling machine or router.



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.

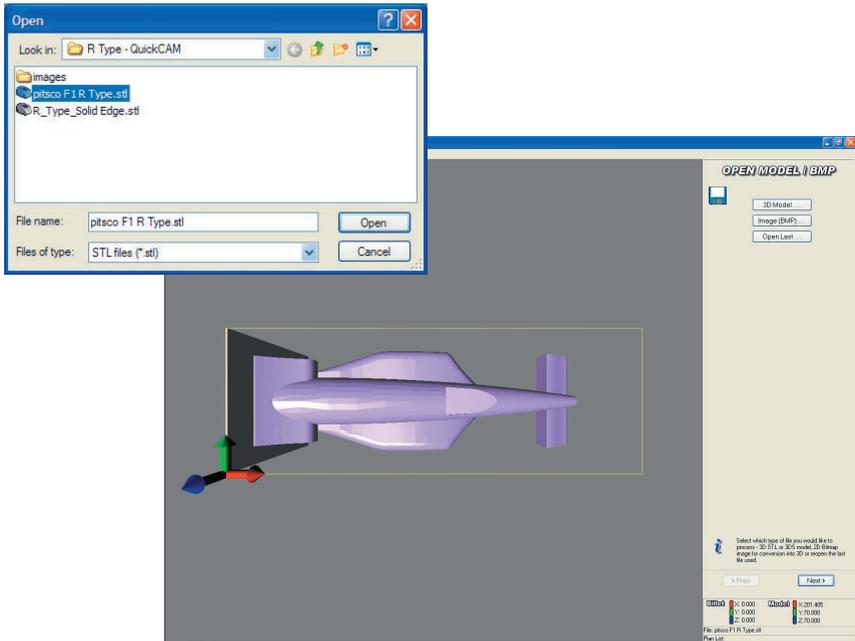


QuickCAM menu icon

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 70mm (the dimensions of the reference plate). The X dimension will vary depending on the length of the car (should not exceed 210mm). 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.

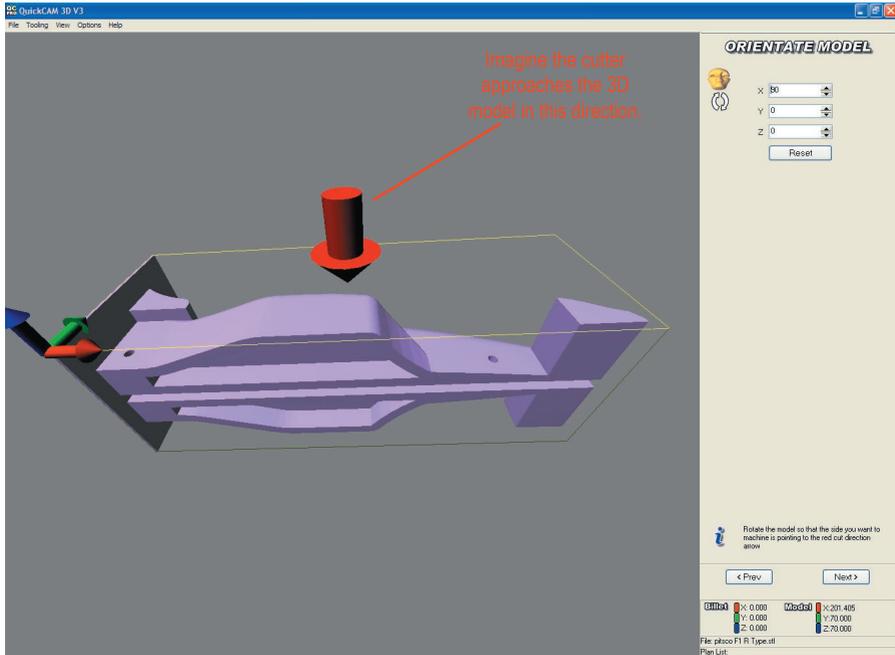
Click and hold the right mouse button whilst moving the mouse to zoom in or out.

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.

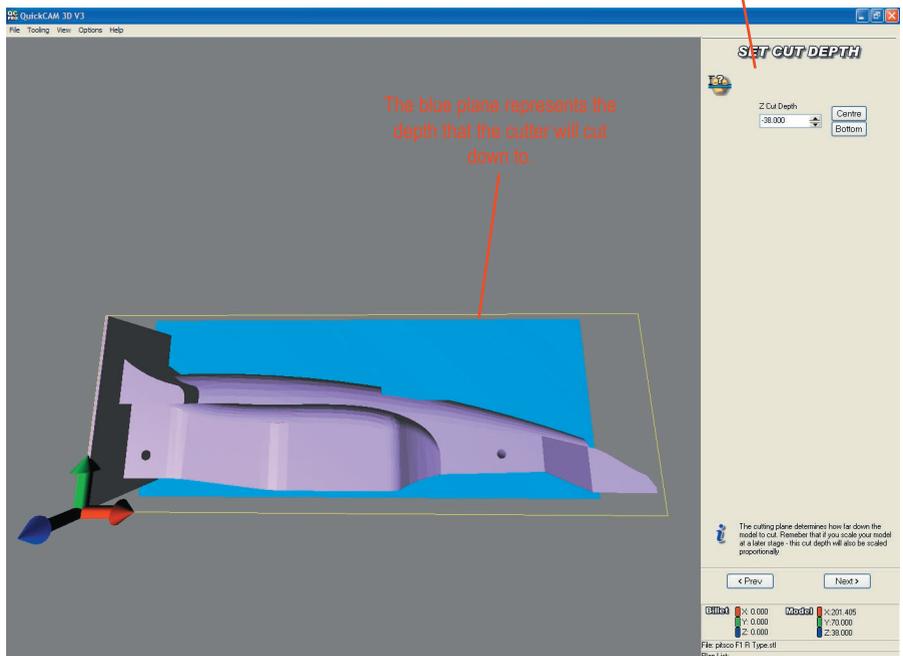
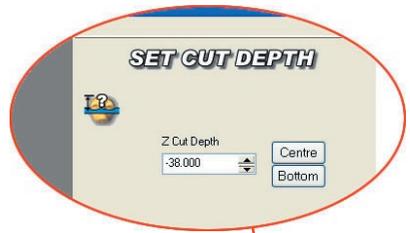
The default option on this screen is to cut to the centre of the model, you should find that the "z cut depth" is at -35. We are going to lower this value by the radius of the cutter to prevent a ridge being left around the car.

Set the value to -38

The standard 1/4" long series cutter, provided with Denford CNC Routers can cut 31mm deep.

If you are using this cutter the model will have to be cut in 2 passes.

An extra long series cutter is available which will cut in a single pass. Contact Denford for details.

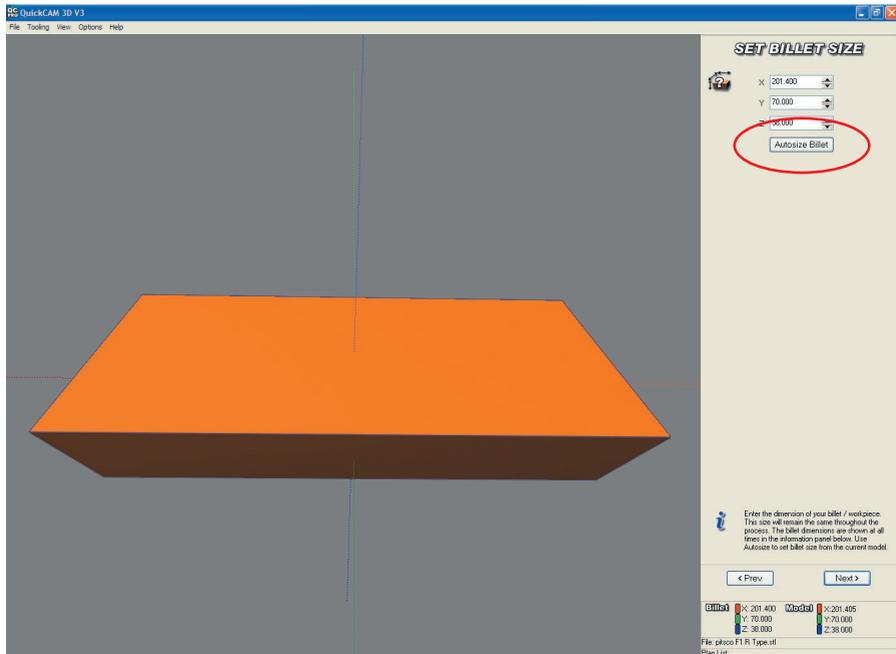


Click the [Next>] button to move to the next screen.

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.



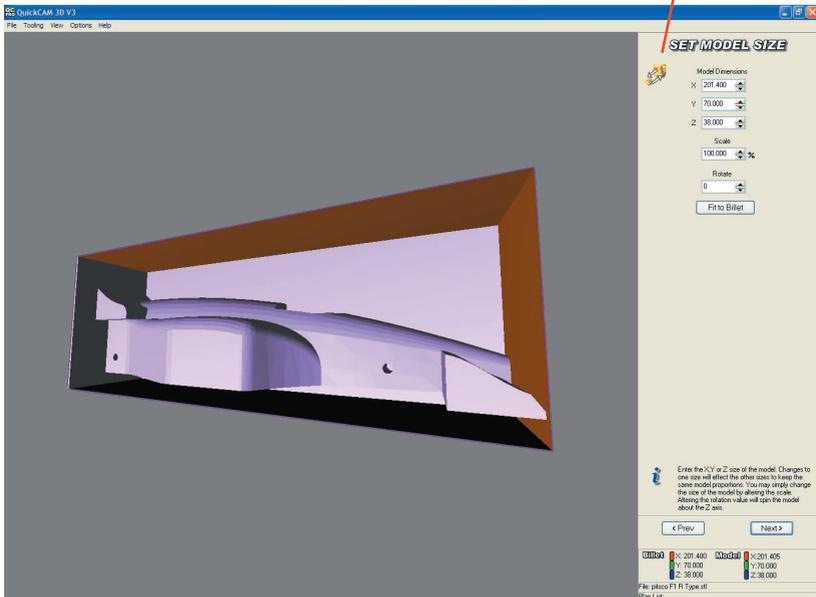
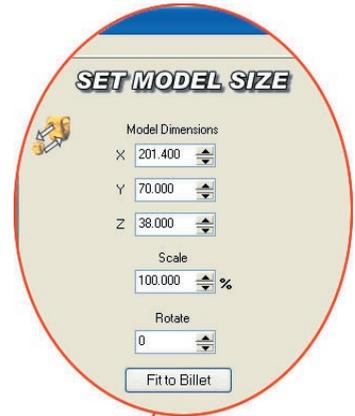
Click the [Next>] button to move to the next stage of the process.

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.)

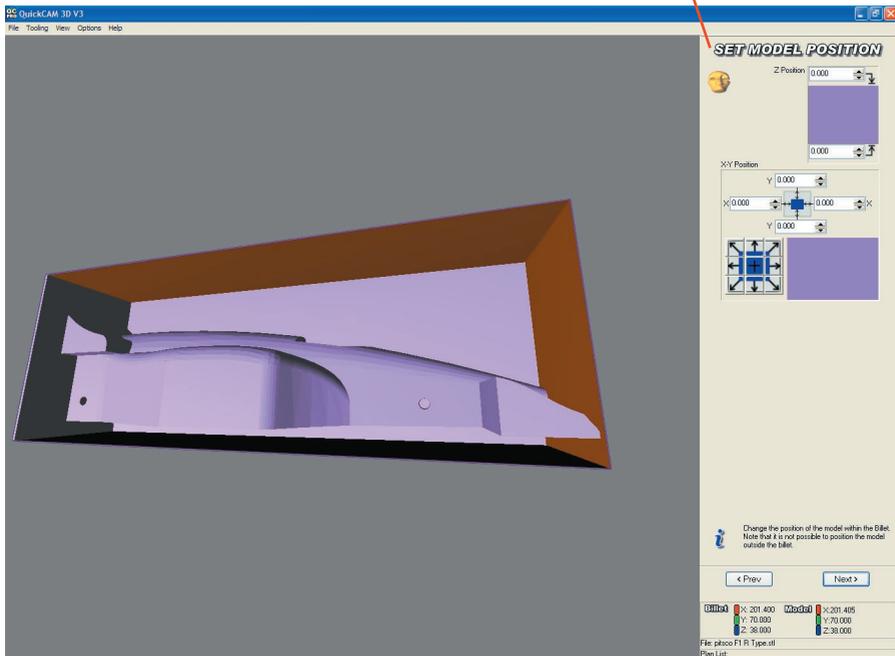
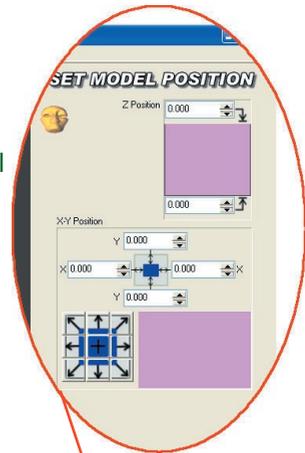


Click the [Next>] button to move to the next stage of the process.

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)



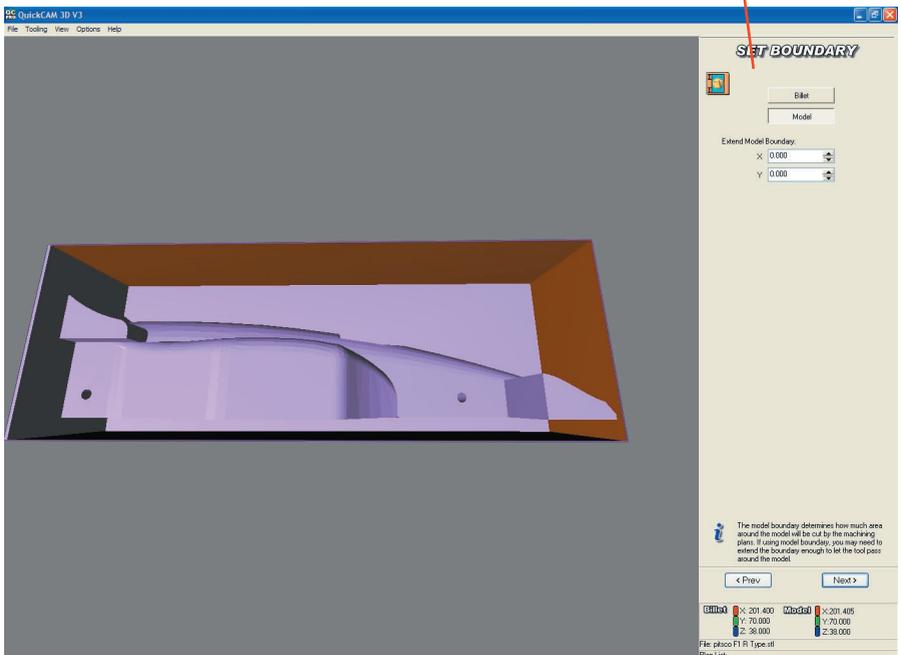
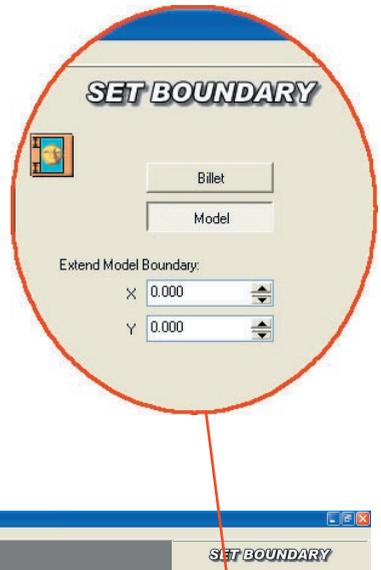
Click the [Next>] button to move to the next stage of the process.

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.

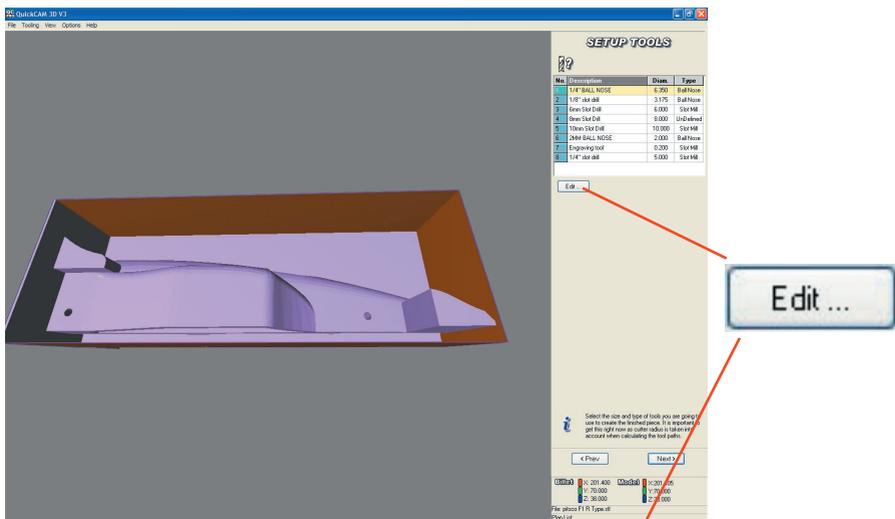


2.9 - Setup Tools

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

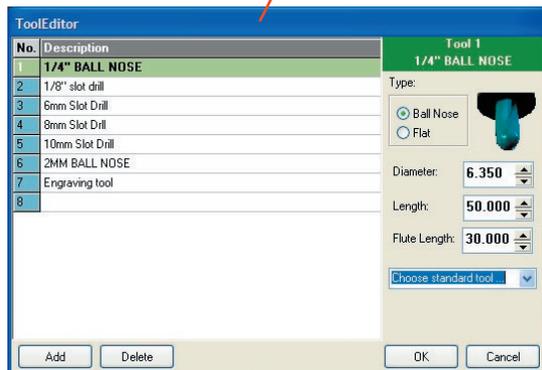
For the F1 Team in Schools car we recommend using the 1/4" Ball Nose Cutter which is supplied with every Denford CNC Router.

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



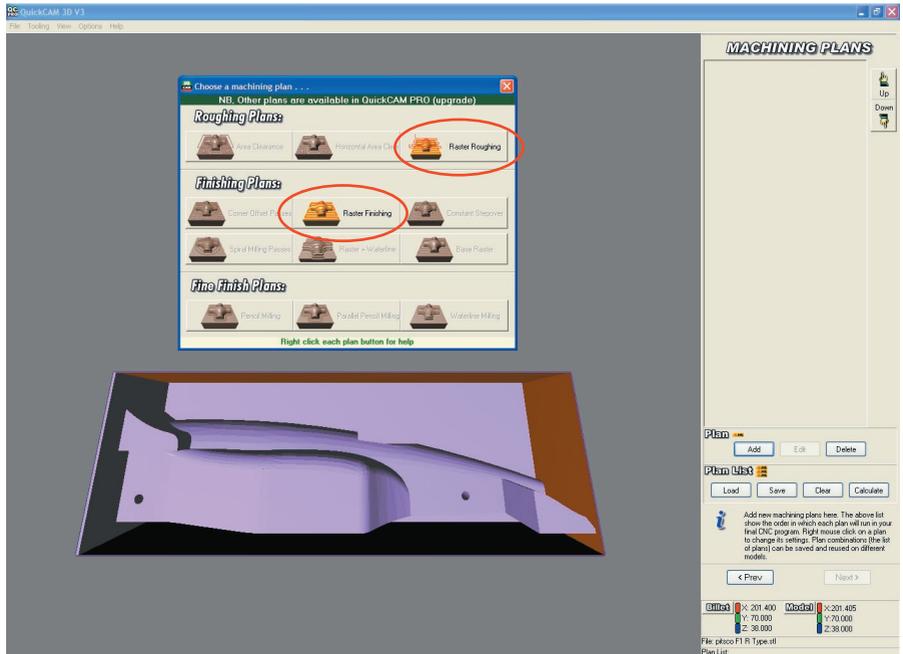
Choose the 1/4" Ball Nose Cutter from the list of standard tools and click [OK].

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.



If you are using an Extra Long series Ball Nose Cutter (has a 38mm flute length) you can choose the 'Raster Finishing' Plan.

Choose the 'Raster Finishing' Plan.



If you are using the Long Series Ball Nose Cutter (has a 31mm flute length) you will need to choose the 'Raster Roughing' Plan. This cutting tool is supplied standard with a Denford CNC Router.

Choose the 'Raster Roughing' Plan



2.11 - Edit Plan Parameters - Raster Finishing

This information applies to those using the Extra Long Series 1/4" Cutter. (38mm flute)

Edit the plan parameters so they appear as below.

Edit parameters for the new plan

Description: Raster Finishing

Tool Data		Machining Boundary	
Tool:	T:1 - D:6.350mm - 1/4" BALL NO:	Minimum	Maximum
Step Over:	1.270 mm <-> 20.000 %	X	0.000 201.400
<input checked="" type="checkbox"/> Create vertical step overs		Y	0.000 70.000
Step Down:	10.000 <input checked="" type="checkbox"/> Adaptive Stepdown	Z	-38.000 0.000
Feedrate:	5000.000	<input type="button" value="Custom Boundary ..."/>	
Spindle Speed:	23000		

General Machining

Safe Height:	5.000	Raster Angle:	90	Cut Direction <input type="radio"/> One Way <input checked="" type="radio"/> Bi-Directional <input type="radio"/> Down Mill <input type="radio"/> Up Mill
Finishing Amount:	0.000	Ramp In Radius:	3.000	
<input type="checkbox"/> Use contact area only		Parallel pencil count:	5	

NB, Other options are available in QuickCAM PRO (upgrade)

- The Tool chosen should be the 1/4" (6.35mm) Ball Nose cutter.
- 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 'Finishing Amount' needs to be at 0.000
- The Feedrate when using a Denford Microrouter should be 5000mm/min
- The Spindle Speed should be 23,000 rpm
- The 'Raster Angle' is 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.

2.11 - Edit Plan Parameters - Raster Roughing

This information applies to those using the standard Long Series 1/4" Cutter. (31mm flute)

Edit the plan parameters so they appear as below

Machining Boundary		
	Minimum	Maximum
X	0.000	201.400
Y	0.000	70.000
Z	-38.000	0.000

Custom Boundary . . .

General Machining

Safe Height: 5,000 Raster Angle: 90
Finishing Amount: 0,000 Ramp In Radius: 3,000
Use contact area only Parallel pencil count: 5
Cut Direction:
 One Way
 Bi-Directional
 Down Mill
 Up Mill

NB. Other options are available in QuickCAM PRO (upgrade)

OK Cancel

- The Tool chosen should be the 1/4" (6.35mm) Ball Nose cutter.

- 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 'Step Down' should be set to 20mm, this means that the tool will take 2 passes to cut to the required 38mm depth.

- The 'Finishing Amount' needs to be at 0.000

- The Feedrate when using a Denford Microrouter should be 5000mm/min

- The Spindle Speed should be 23,000 rpm

Click [OK] and the toolpaths will be calculated.

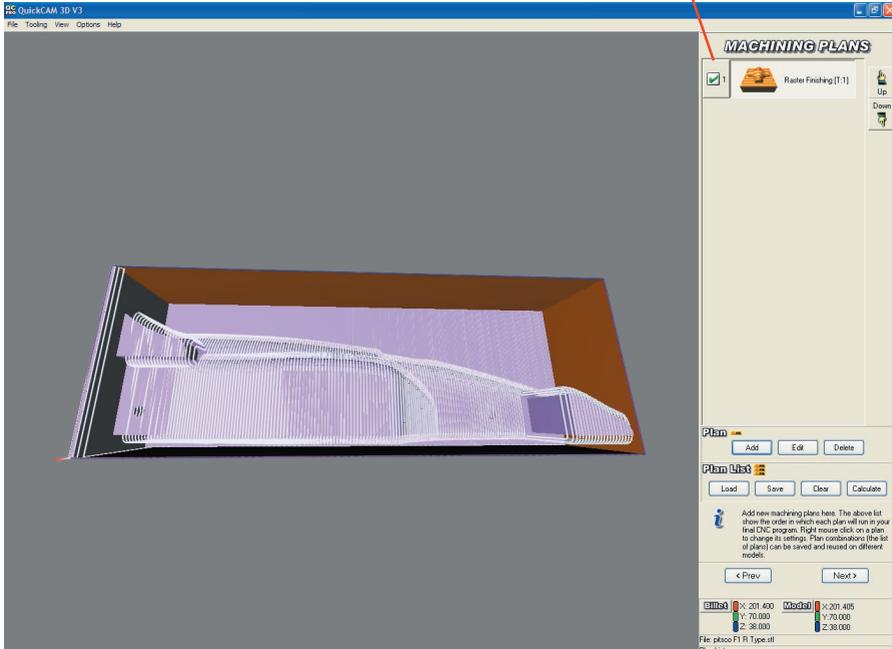
2.12 - Create the toolpath

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.

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



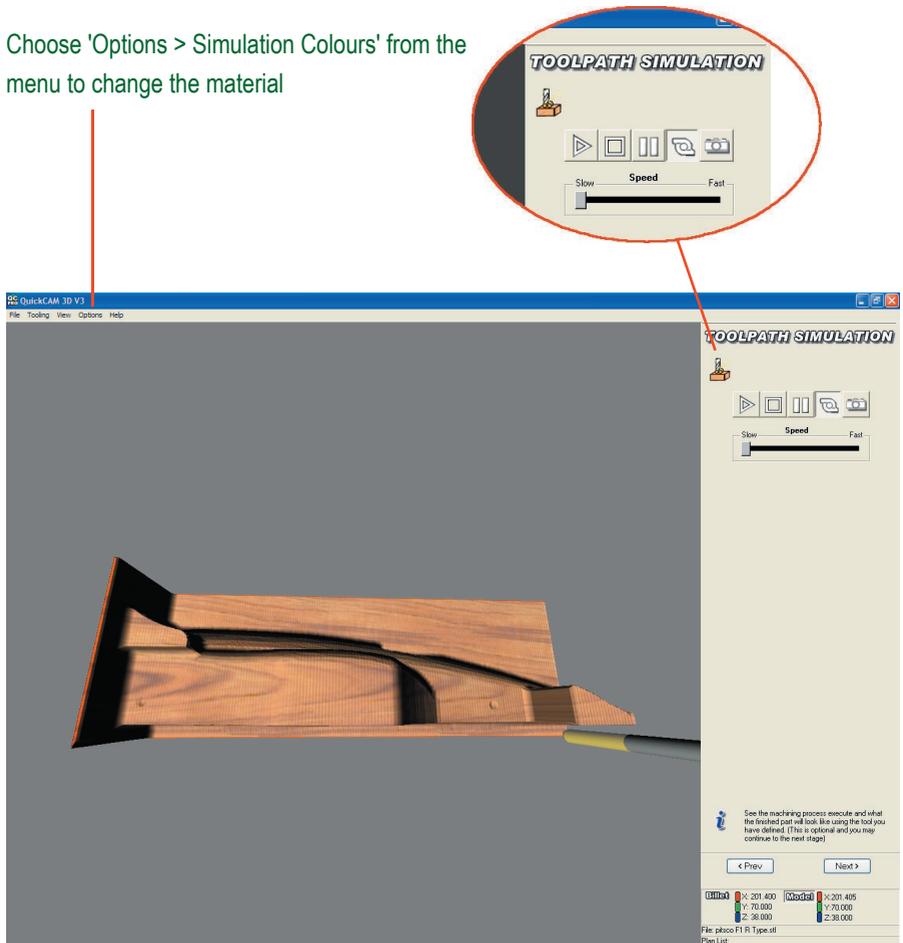
2.13 - Simulate the toolpath

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

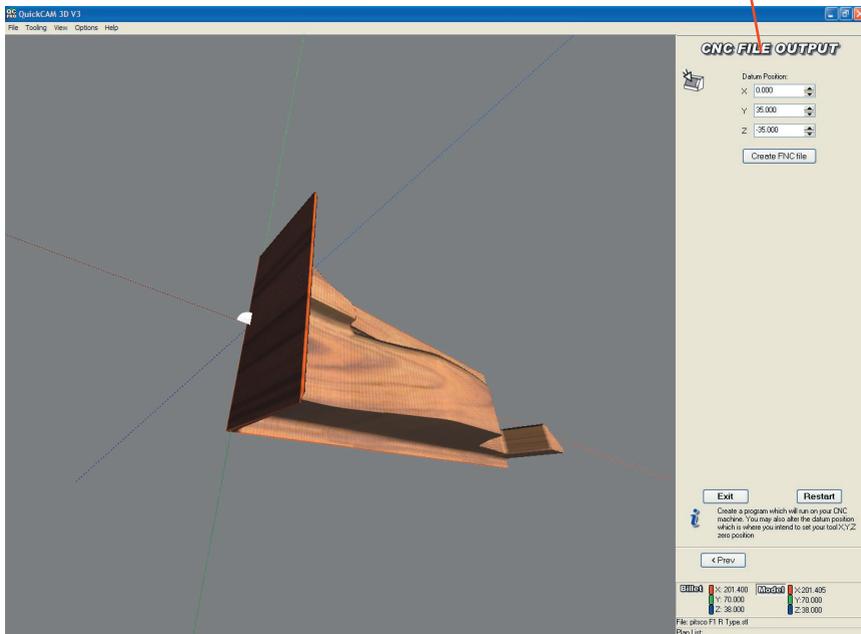
This is the final screen in QuickCAM and is where the CNC file is saved. Before we click the [Create FNC file] button, the 'datum' position will be changed. The 'datum' is the reference point for the program. We will move the datum to the centre of the hole in the Balsa Wood blank. Drawing the reference plate in the 3D CAD software makes the centre of the hole easy to find. The reference plate was 70mm x 70mm, and was centered around the hole. We therefore need to change the datum position to X = 0, Y = 35mm and Z = -35mm 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.



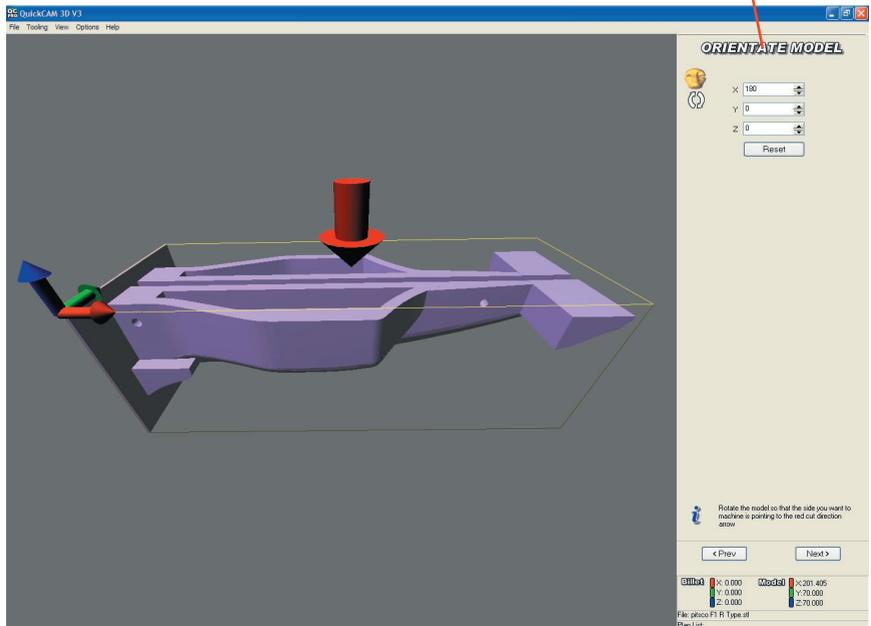
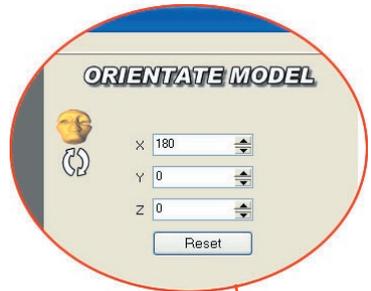
2.15 - Creating the CNC files for the other sides.

To create the 2 sides of the car we can use the same CNC file but input a simple code - 'M71' to mirror the program in the Y axis and produce the other side.

If your car has detail on the top or bottom which could not be produced by machining the car from the sides, you will need to produce another CNC file.

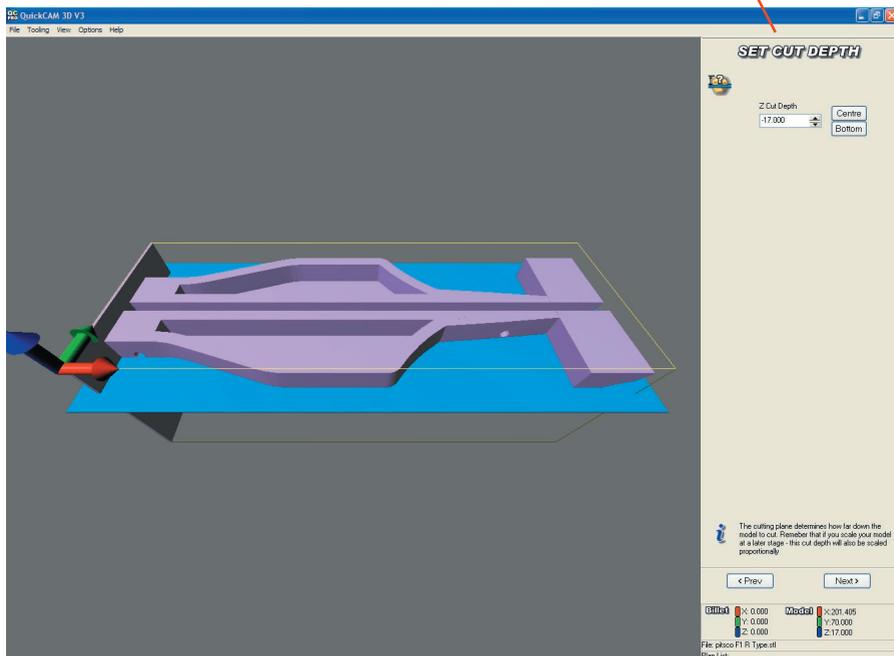
This example has some recesses on the underside which would not be produced by only machining the car from both sides. If this is the case run through QuickCAM again but re-orientate the model so that the detail can be produced.

To capture the detail underneath, this particular model has been rotated through 180 degrees.



2.15 - Setting the Cut Depth for the other sides.

When setting the Cut Depth for the other sides, adjust the 'Z cut depth' to capture the additional detail only. By default the cut plane will be set to the centre of the model (-35mm). The additional detail on the underside of this example is 17mm deep so there is no point in machining any deeper than that.

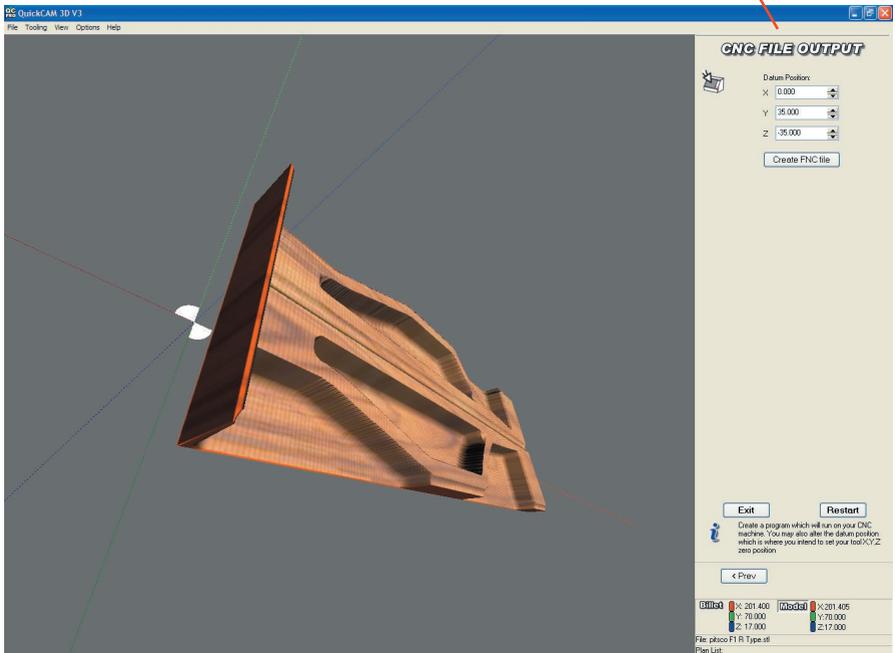
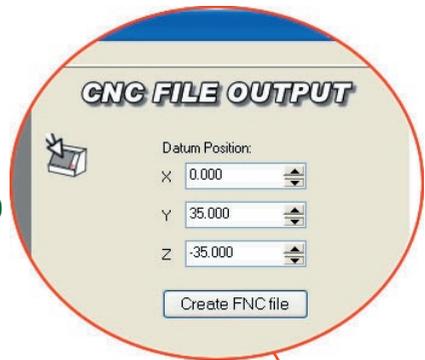


2.15 - Creating the CNC files for the other sides.

The rest of the pages in QuickCAM should have exactly the same settings as detailed previously.

Make sure that you still have the Datum position set at X=0.000, Y=-35.000, Z=-35.000

Setting the Datum in this position means we only need to set one start position at the machine for all CNC files.



Click the [Create FNC file] button. Save the file with a relevant name.

You are now ready for the next stage.

Stage 3 - Using Denford VR Milling V5

Denford VR CNC Milling is a simple to operate software package used to control your

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.



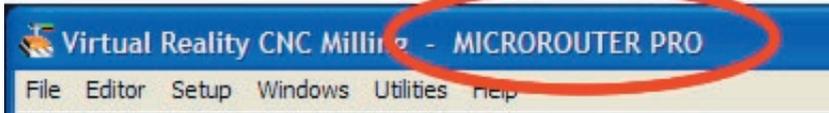
If the shortcut is not available, click "Start" on your "Windows" Start button followed by the "Programs" option, the program group "Denford" and finally the "VR Milling V5" icon.



3.2 - Configure the software for the machine

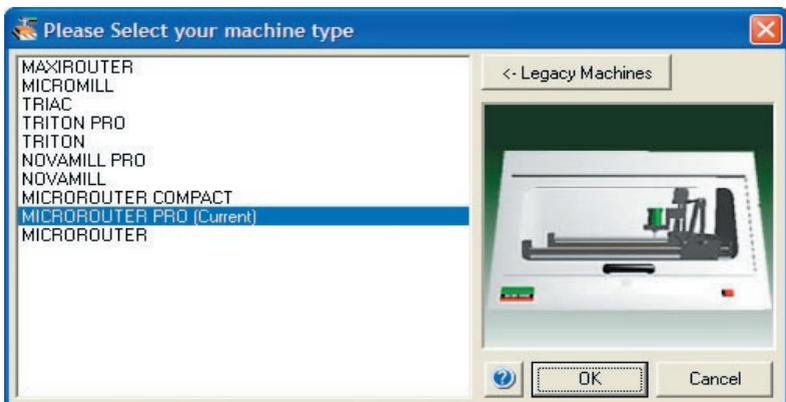
Ensure that the software is configured for the machine you are going to use.

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

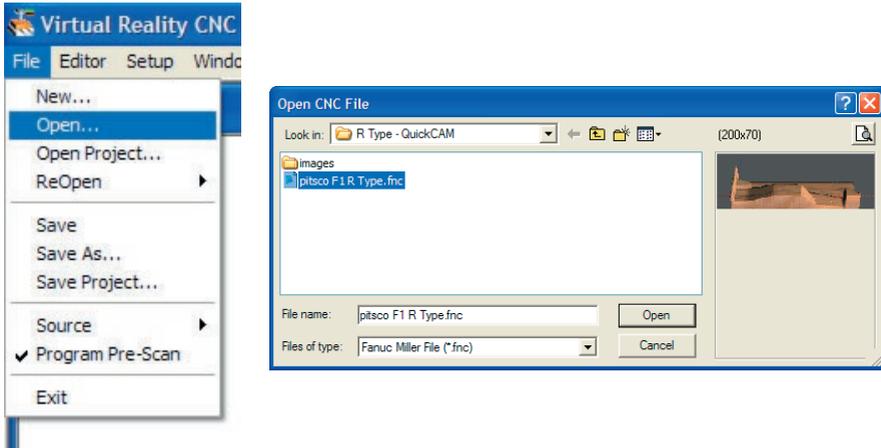


Legacy machines: These are older machines that are no longer in production but are still compatible with VR CNC Milling V5 software. Click the [Legacy Machines] button to list these types of machine.

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:

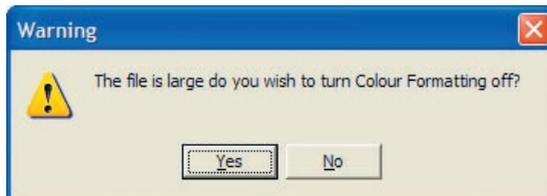
Colour Formatting

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

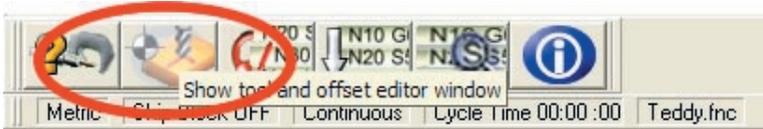
When this facility is activated, any subsequent CNC programs are scanned before they are loaded into the editor. During the scan they:

- 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:

- 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”
- 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”



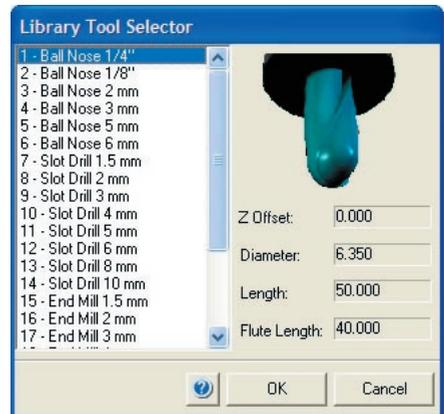
“Insert Library Tool”

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.



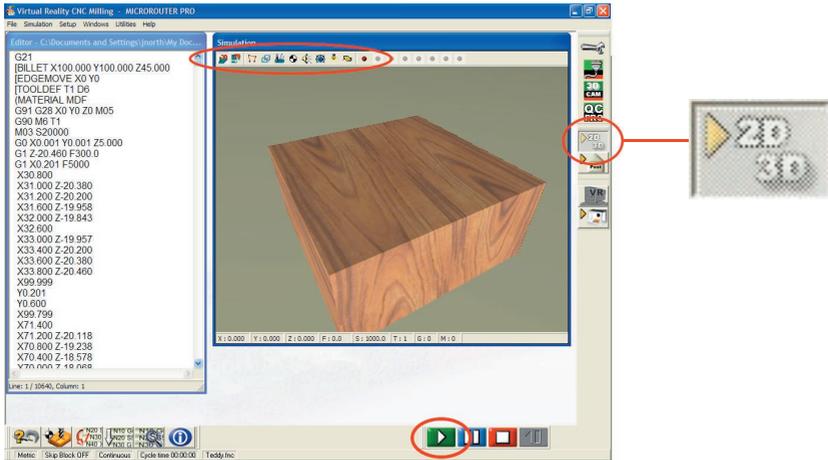
Choose the tool from the “Library Tool Selector”, click [OK]. The ¼” Ball Nose cutter should appear in tool position number 1.



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

3.5 - 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.

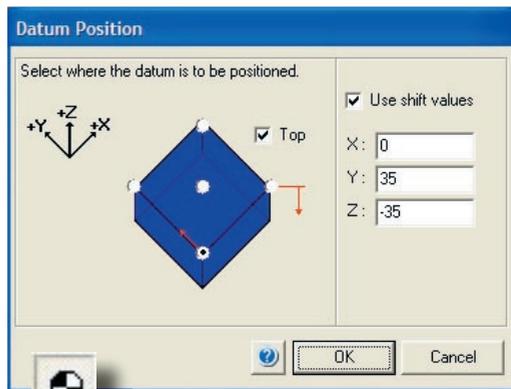
Under the “Simulation” menu choose “Billet Materials ...” to change the appearance of the simulation including the material with which the model is rendered.

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 = 35 and Z = -35, 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.

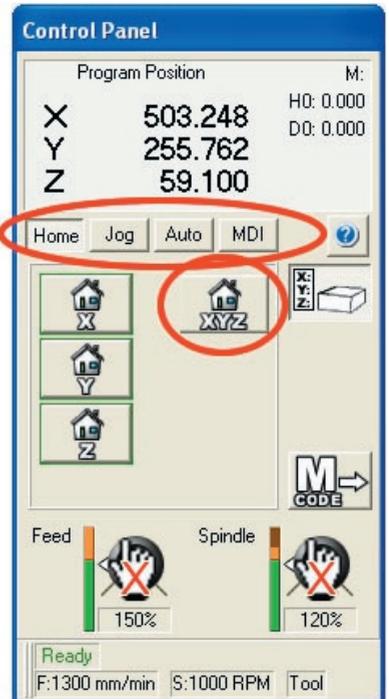
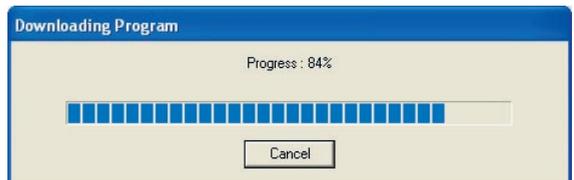


3.6 – Connect to, and Home the CNC machine



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.

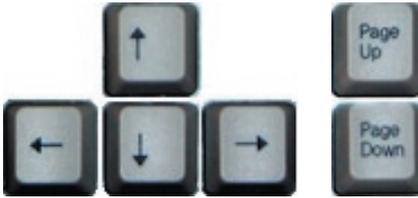
3.7 - Move the machine head and fit the cutting tool.

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.

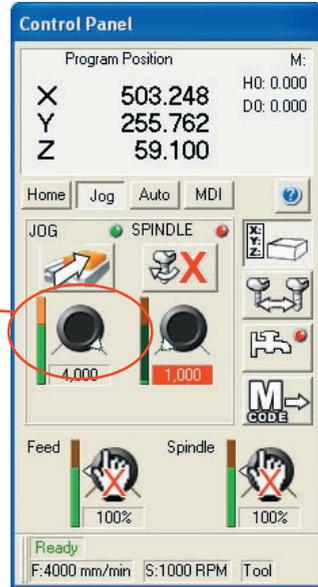


To change the position of the machine head incrementally, click the Jog button until the image changes from a single arrow to three small, stepped arrows, signifying Jog Step Mode.

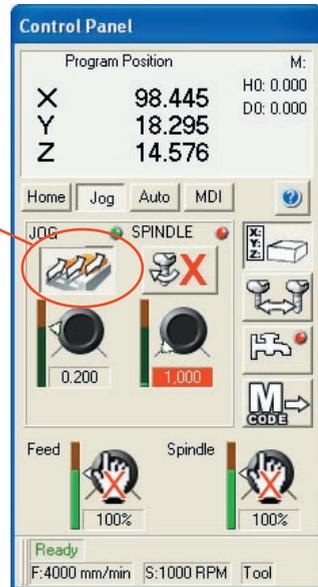
Click and drag the Jog Feed control knob to adjust the increment. When you press the cursor keys the cutter will move by the amount set.

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.

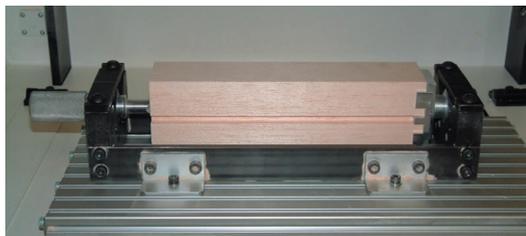
The circular fork on the left of the jig fits into the CO2 cartridge hole on the back face of the balsa wood billet. The taper end of the billet is tightened against the bracket on the right side of the jig.



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).



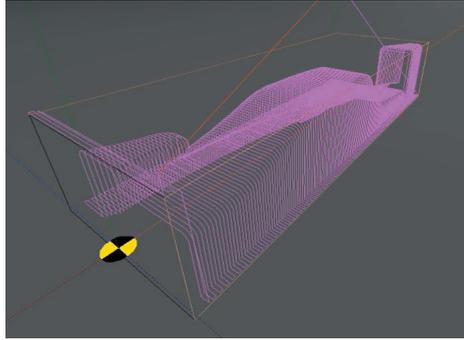
Fit the balsa wood billet in the jig as shown.

3.9 – Set the work offsets

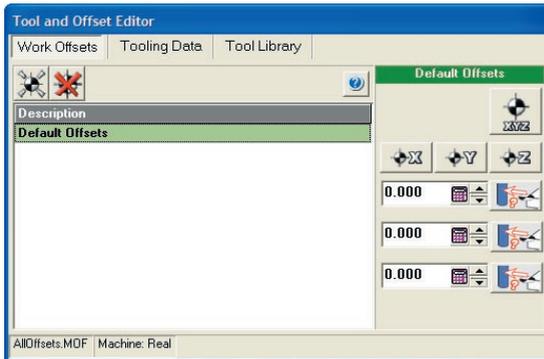
What are offsets?

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

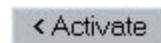
It is possible to store a number of offsets and swap between them for different jobs. Use this facility to create a new offset and add it to the list.

- 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.



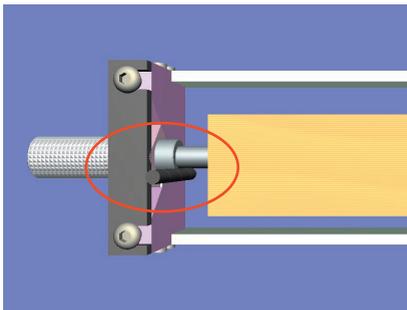
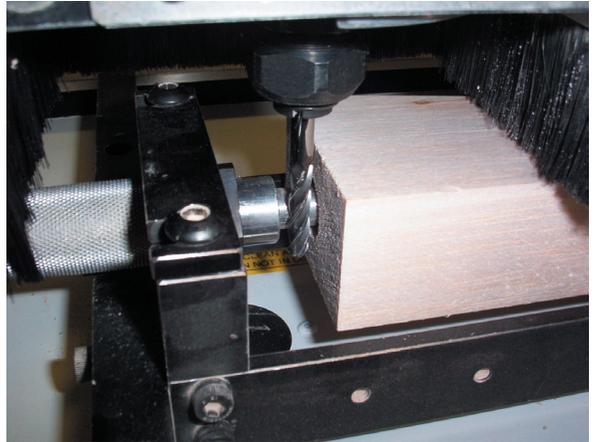
"New work offsets"



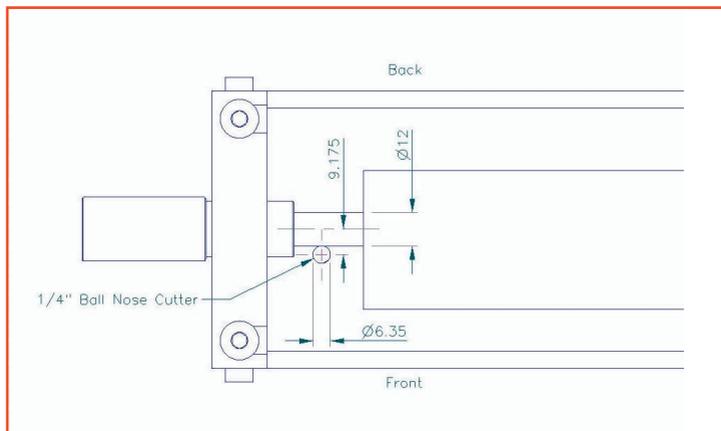
"activate offset"

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

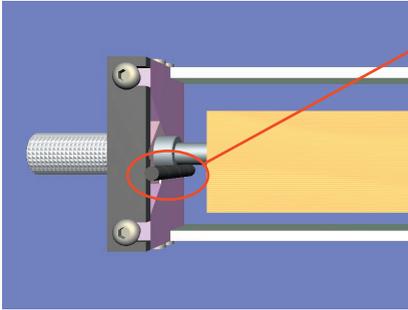
In 'jog continuous mode', move the tool a few millimetres away from the fork. We are going to touch onto the fork at its smallest diameter. (With the jig shown here it is 12mm, but other varieties may have a different diameter.)



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

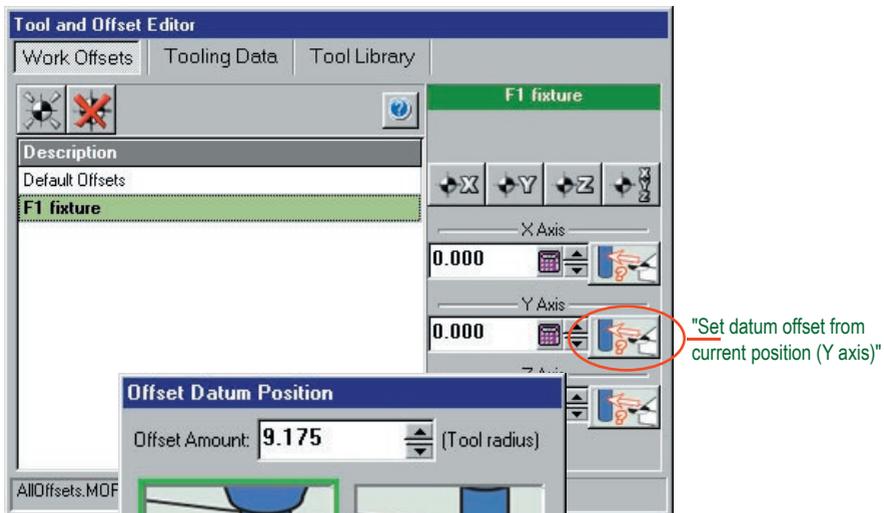


With the cutter in this position we will find the centre of the fork.

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.



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

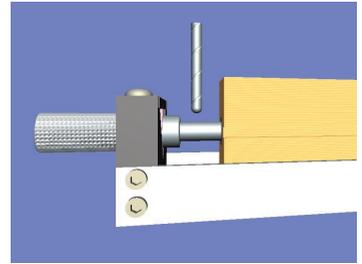
Click the Y Minus button to indicate which side of the fork the cutter is positioned.

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

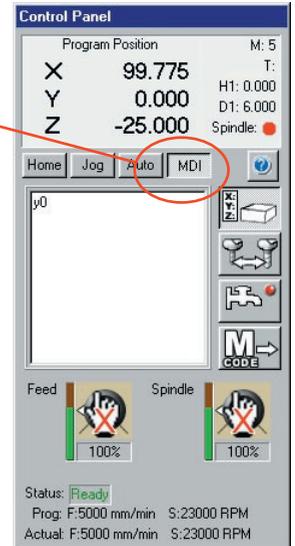
Click the 'Jog' tab to go back to the jog mode.

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.



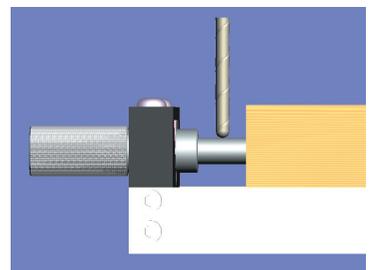
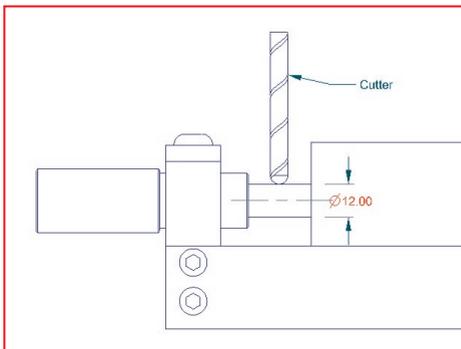
Click on the MDI tab in the control panel and type Y0 (zero not the letter O)

Press Play on the file control.

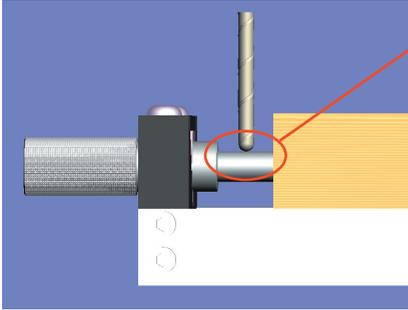


The cutter will move to the Y position set previously. This should be directly over the centre of the location fork, if not the Y offset is incorrect, repeat the previous operation.

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

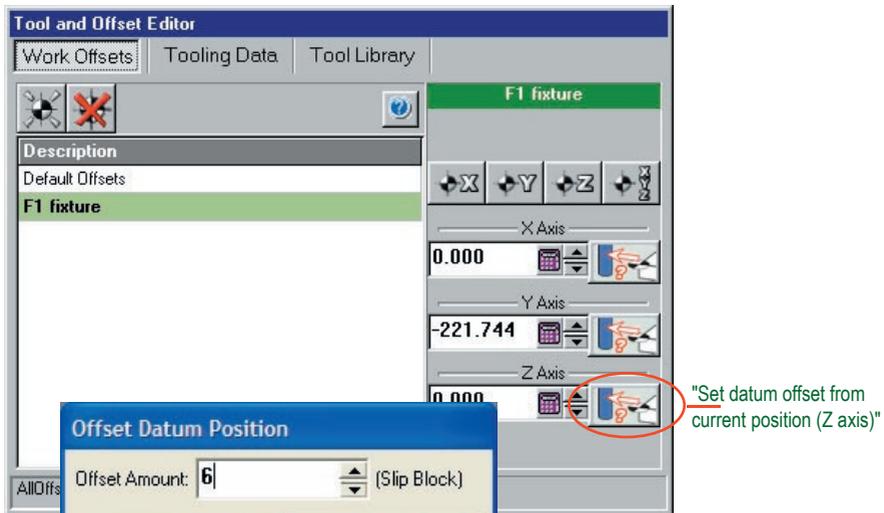


With the cutter in this position we will find the centre of the fork.

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.



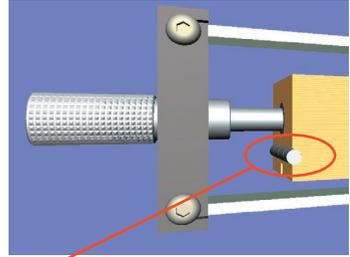
Click the Z Plus button to indicate which side of the fork the cutter is positioned.

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

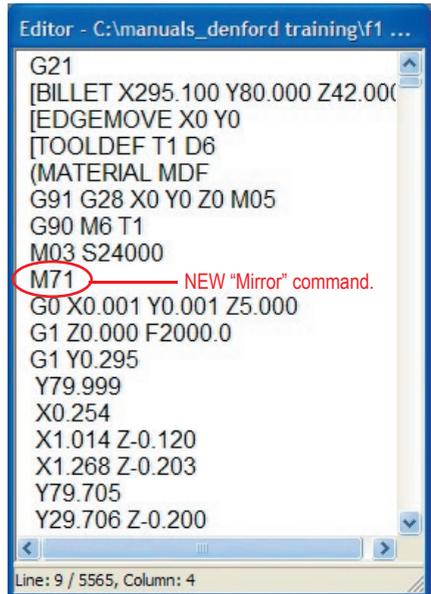
Click the [Turbo Mode] button to switch Turbo Mode on. This can be done at any time, even when the program is running. The 'turbo mode' feature has been developed to reduce the machining times of large 3D programs and complex 2D programs. For larger programs E.G. more than 100 lines, turbo mode on will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

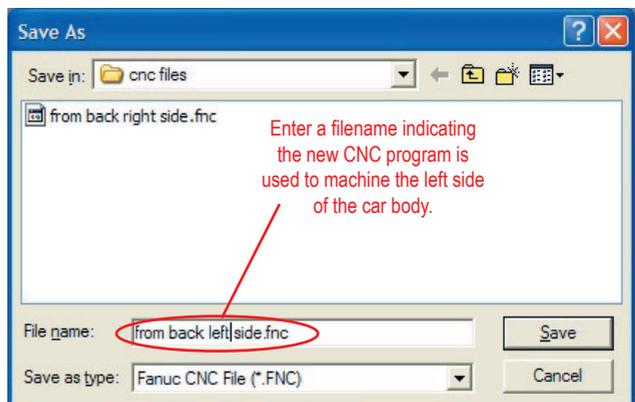


```
G21
[BILLET X295.100 Y80.000 Z42.000
[EDGEMOVE X0 Y0
[TOOLDEF T1 D6
(MATERIAL MDF
G91 G28 X0 Y0 Z0 M05
G90 M6 T1
M03 S24000
M71
G0 X0.001 Y0.001 Z5.000
G1 Z0.000 F2000.0
G1 Y0.295
Y79.999
X0.254
X1.014 Z-0.120
X1.268 Z-0.203
Y79.705
Y29.706 Z-0.200
```

- this is the code defining the “mirror in the Y axis” command.

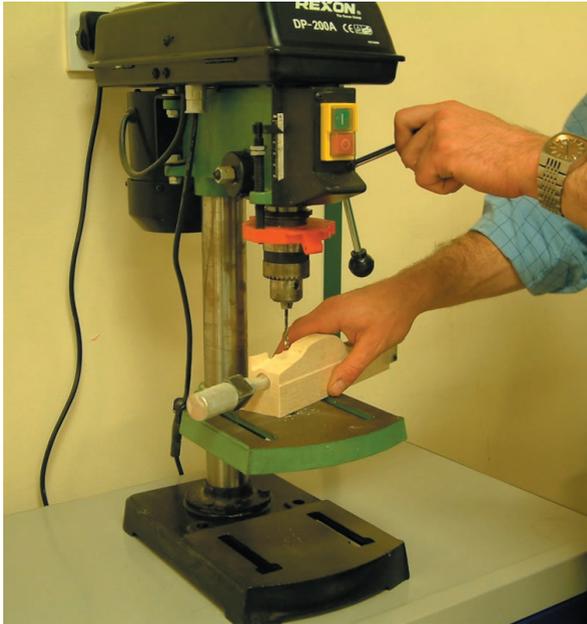
Click the “File” menu and select the “Save As” option. DO NOT click “save” by mistake, since this will overwrite your original CNC file!

Enter a name describing the outcome of your new CNC file - in our example



3.12 - Drilling the axle holes in the model

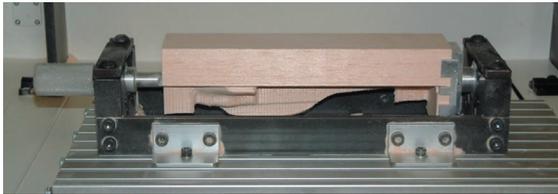
Before machining the second or third sides it is a good idea to drill the axle holes. Drilling before machining the other sides ensures that there is a flat reference, a pillar drill can be used to drill the two axle holes. Use a 4mm diameter drill.



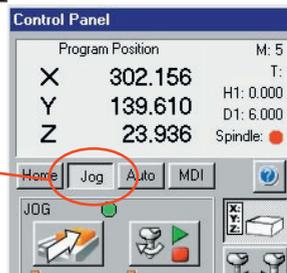
Alternatively, you could place the 4mm drill into the CNC machine, place the model back into the jig and manually drill the holes. Ensure the drill is positioned in the spindle so it can machine completely through the full width of the model. Use "Jog Mode" to move the CNC machine head to align with 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.

3.13 - 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.



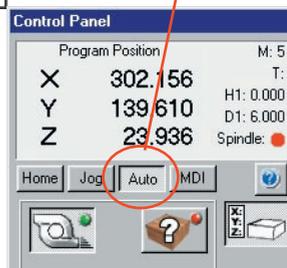
```
Editor - C:\from back left side.fnc
|
|  G31
|  [BILLET X295.100 Y80.000 Z42.000
|  [EDGEMOVE X0 Y0
|  [TOOLDEF T1 D6
|  (MATERIAL MDF
|  G91 G28 X0 Y0 Z0 M05
|  G90 M6 T1
|  M03 S24000
|  M71
|  G0 X0.001 Y0.001 Z5.000
|  G1 Z0.000 F2000.0
|  G1 Y0.295
```

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

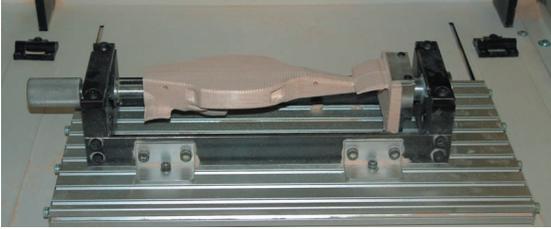
Left click on "Auto" to enter auto mode.



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



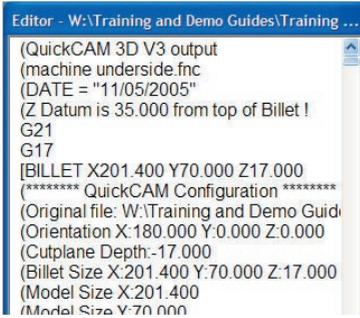
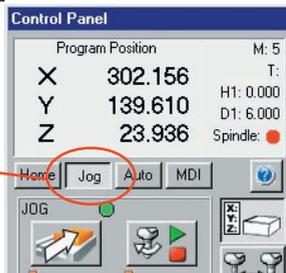
3.14 - Machining any subsequent sides of the car



Physically rotate the Balsa Wood in the fixture by the required 90 degree increment.



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

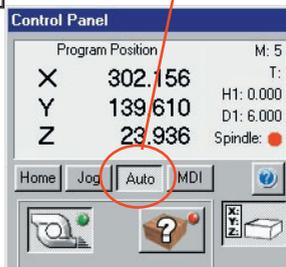


You must now make sure you have opened the required CNC file

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 728080

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,
Savoy Place,
London,
WC2R 0BL,
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, 2005. 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. Solid Edge is a registered trademark of EDS. 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.