

Ladybird Project - Vacuum Mould



Prerequisite Mould drawn and saved as “STL” file from Solidworks

Focus of the Lesson On completion of this exercise you will have completed:

- Opening “STL” file
- Setting Machining Constraints
- Set up Tools
- Select Machining Plan
- Post Process
- Set Machining Parameters
- Set Tool Offsets
- Machine the Mould

PREPARATION FOR MACHINING (Denford)

Opening the Model

Start the QUICKCAM PRO programme.

Click on the 3D model button on the right of the screen (Figure 1).

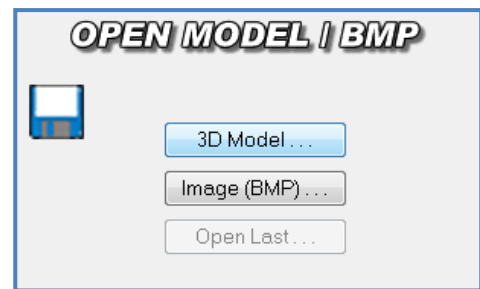


Figure 1

Locate the STL file from the saved location and click on Open (Figure 2).

QuickCam Pro will import the mould design (Figure 3).

There are number of steps which need to be undertaken before proceeding with the manufacture of the mould.

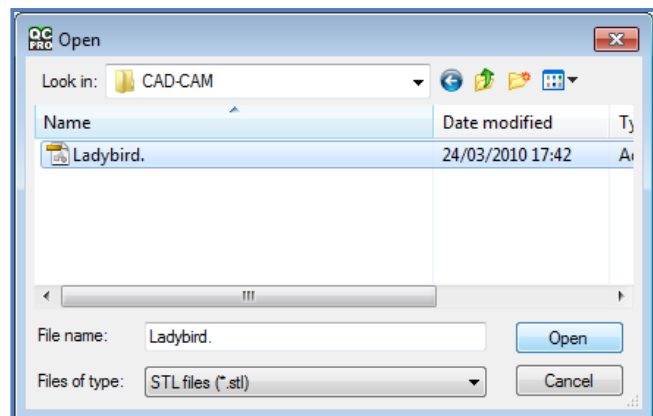


Figure 2

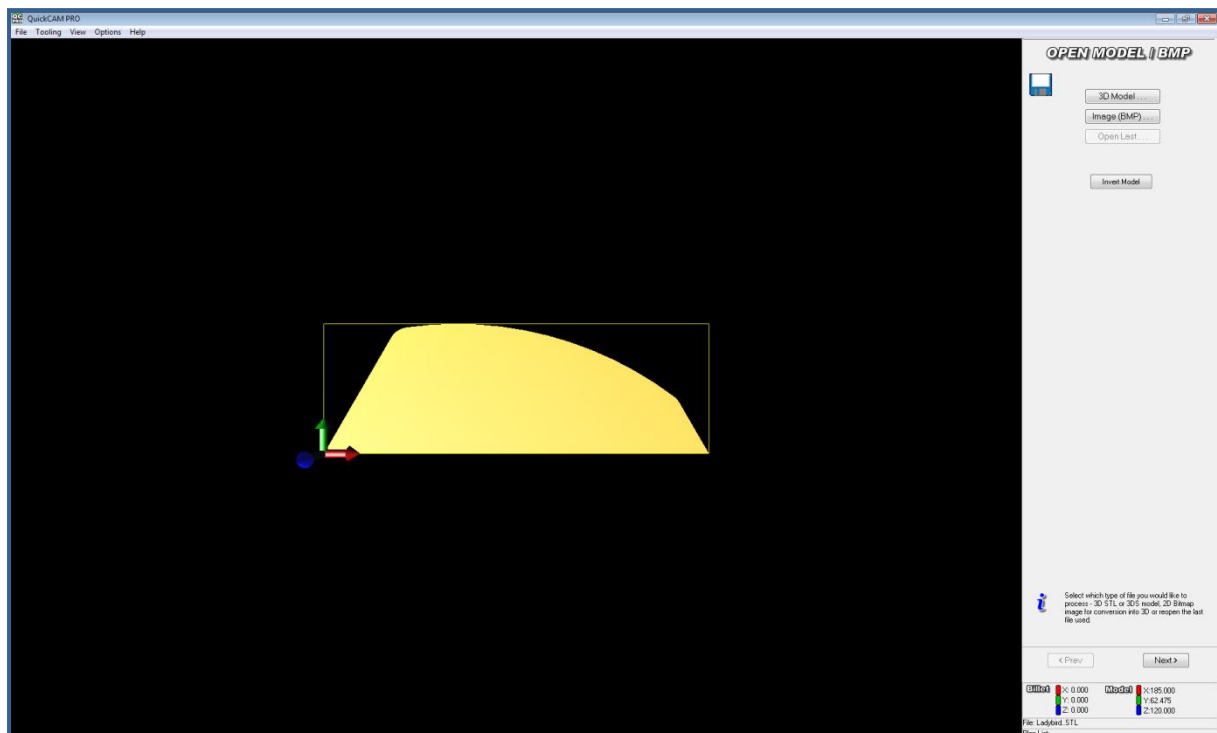


Figure 3

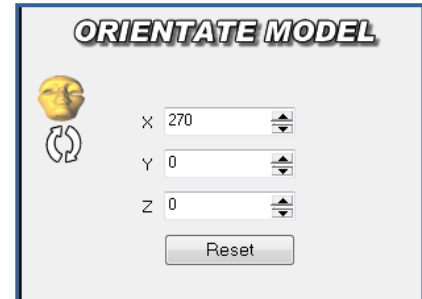
Click the Next button to continue.

SETTING THE MACHINING CONSTRAINTS

Orientate model

Align the direction of cut on screen, as shown by the red pointer, to match that of the CNC machine cutting tool.

This will allow you to achieve the desired cutting direction and is done by adjusting the X, Y and Z axes.



In this case the X axes is set to 270° , but the Y & Z are left at 0 (Figure 4). Figure 4

Click on the mould & hold down the left mouse button to rotate the mould to get a better view of the new cutting direction (Figure 5).

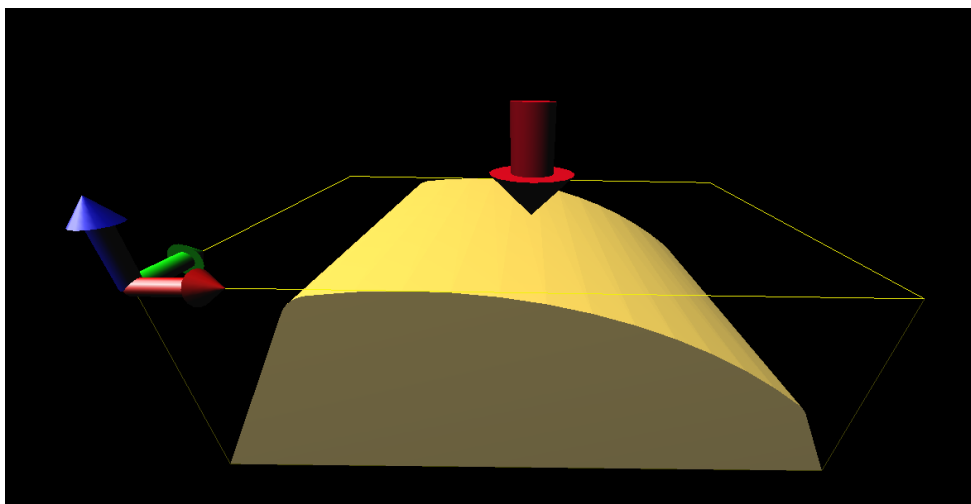


Figure 5

Click the Next button to move to the next screen.

Selecting Depth of Cut

There are two default settings:

- Centre
- Bottom

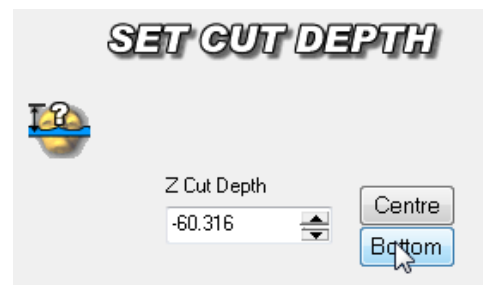


Figure 6

The depth of cut can also be set manually, by typing in the depth required or using the up/down arrow keys (Figure 6).

For this exercise the depth of cut is set to the Bottom.

Click the Next button to move to next setting.

Set billet size

The billet size is the dimensions of the material from which you intend to make the mould.

The billet size on this occasion is slightly larger than the finished mould and is set by adjusting the X, Y and Z dimensions of the billet (Figure 7).

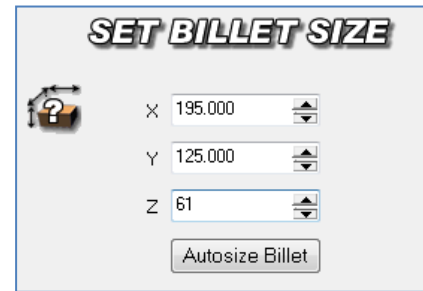


Figure 7

Alternatively, you can auto size the billet. If the billet is auto-sized the programme will set the billet to the same dimensions as the model.

The model and billet dimensions can be seen in the bottom right corner of the screen.

Click the Next button to continue.

Set Model Size

The model size will be set to the actual size of the model by default but can be adjusted by setting the X, Y and Z dimensions (Figure 8).

Selecting Fit to Billet means the model will be set to the same size as the Billet.

Leave the default settings and click Next to continue.

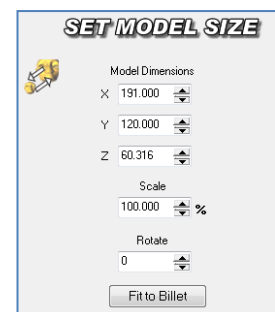


Figure 8

Set model position

If the model is smaller than the Billet, it can be moved within the billet.

This is done by manually changing the X, Y, position values or alternatively by selecting one of the nine positional squares to move the model to the desired location (Figure 9).

Click Next to continue.

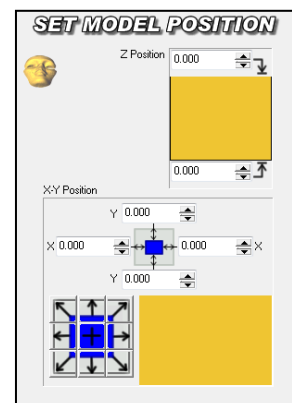


Figure 9

Set boundary

The boundary can be pre-set to the size of the:

- billet or
- model.

Figure 10 shows the model boundary set to the size of the billet.

Click the Next button to move to the next setting.

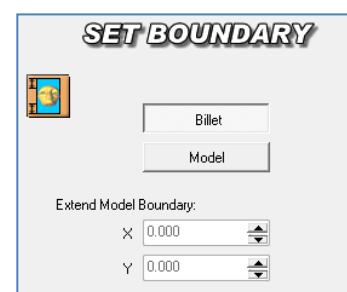


Figure 10

SET UP TOOLS

The model and billet parameters are now set, but the correct cutting tool must also be selected before machining begins.

Selecting the required tool

Click on the "Edit" button (Figure 11).

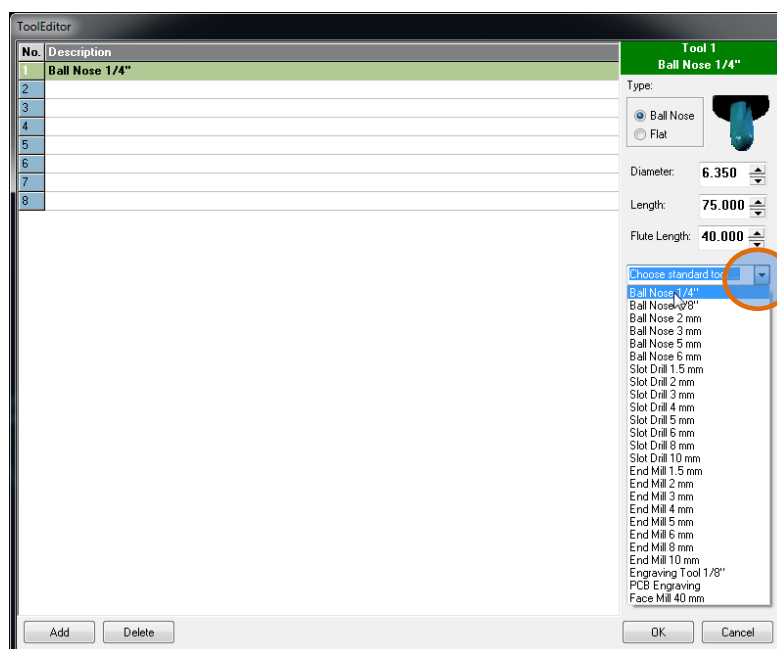


Figure 12

should be checked and altered if necessary. This includes tool diameter, length and flute length. This is vitally important as the tool diameter is taken into consideration when calculating the tool path.

Click OK to confirm selection and to close the Tool Editor Window. The selected tool now appears in the 1st slot of the Setup Tools window (Figure 13).

Be extremely careful when setting tool parameters as the wrong length of tool or wrong flute length inputted could lead to the collision of the cutting tool holding device with the billet

Click Next to continue.

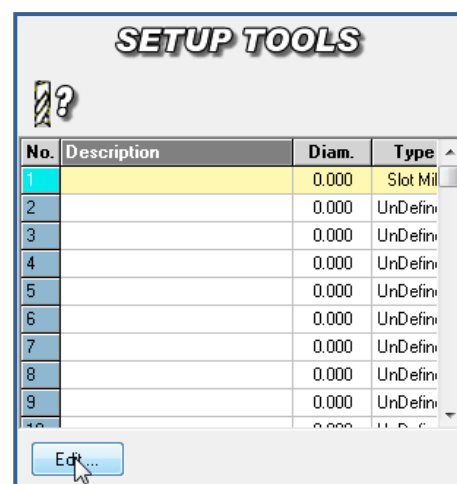


Figure 11

In the tool Editor window click on the down arrow to display the tool library (Figure 12)

From the list of available tools choose the Ball Nose 1/4" (Figure 12).

All relevant data concerning the tool

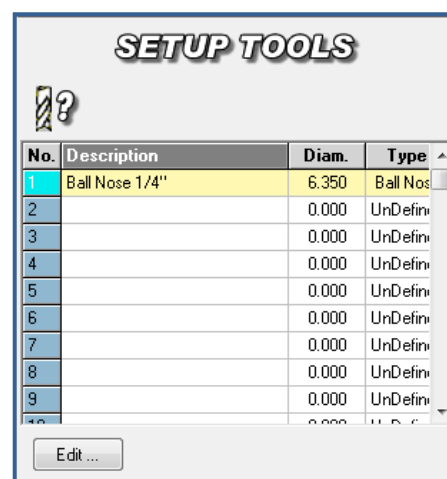


Figure 13

Machining plan

The machining plan is where the type of cut required is selected. There are 3 types of machining plans:

- rough cut,
- finishing cut
- fine finishing cut.

Click the Add button to begin selecting the required plan (Figure 14).

The type of plan required will often depend on the geometry of the mould. As a plan is selected it will be added to a plan list.

The plan list shows the order in which each plan will run in the final CNC programme and can be saved for reuse if required.

When you select a plan you have the opportunity to edit the parameters for that plan (Figure 16). The parameters are preset to the default values according to the tool data you have already input. This includes information such as spindle speed, feed rate and step down distance. However, you can adjust these values to suit your particular needs.

Select the Raster Roughing Cut and change the Step Down value to 4mm as shown in figure 16.

Click Ok to continue.

When a plan is selected the software calculates the machining paths (Figure 17).

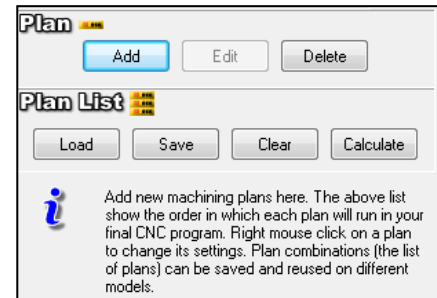


Figure 14



Figure 15

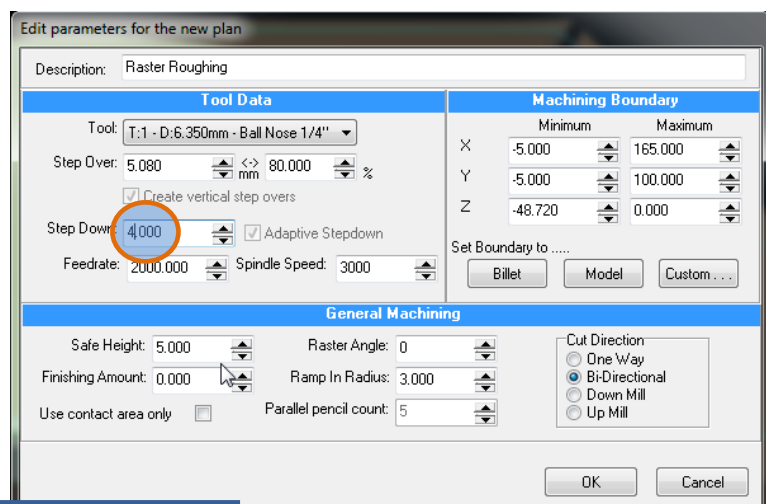


Figure 16

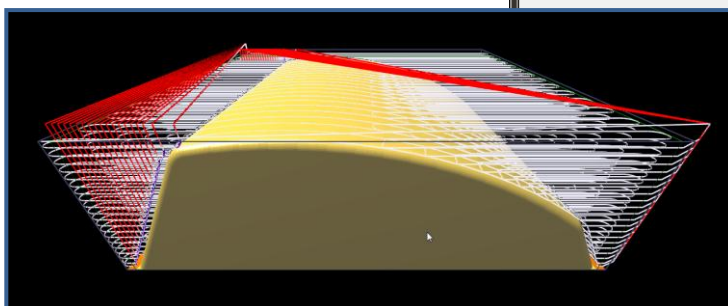


Figure 17

The plan is added to the Machining Plans list.

Select Raster Finishing as the finishing cut and the plan is added to the list (Figure 18).

Click on the save button to save the machining plan.



Figure 18

Having completed the machining plan it is important to check for any unidentified problems and that the machining plan will give the desired outcome. This is done by through the use of a tool path simulation.

Click the (Play) Run Simulation Button (Figure 19).

The speed of the simulation can be adjusted from slow to very fast by moving the slider to the right (Figure 19).

The software will simulate the machining cycle (Figure 20)

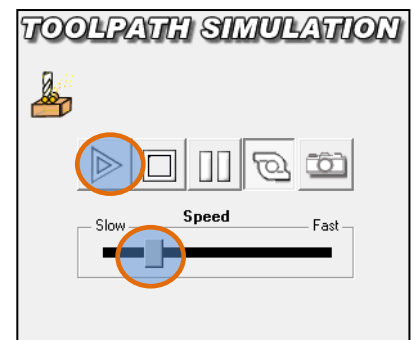


Figure 19

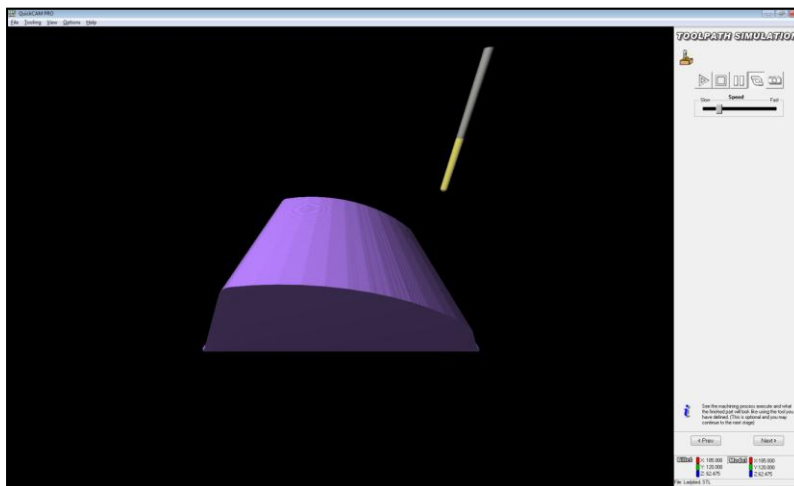


Figure 20

Post Process

File output

When the simulation is complete and you are satisfied with the predicted outcome the file is sent to the VR Milling programme.

Click on the Next button to proceed.

Check that the correct machine is selected (Figure 21) and then click on the Post Process button.

At this point the file must be saved as an "fnc" milling file in order to be recognised by the Miller/Router machine. Name the file "Ladybird" and save it to a location of your choice (Figure 22).

When the file is saved the QuickCAM PRO software will automatically open the VR milling programme and the posted FNC file will be loaded and ready for machining (Figure 23).

If for any reason the VR Milling Programme does not open automatically it can be manually opened and the relevant file selected.

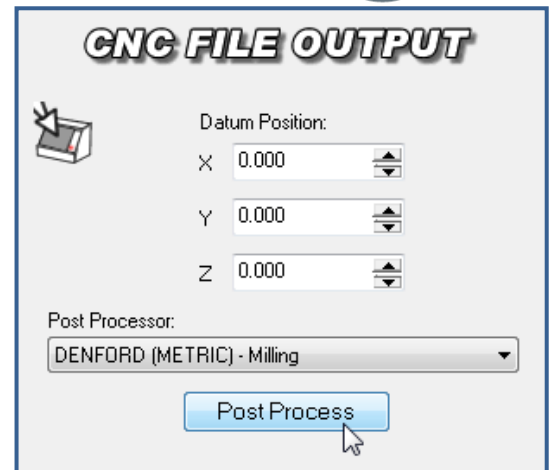


Figure 21

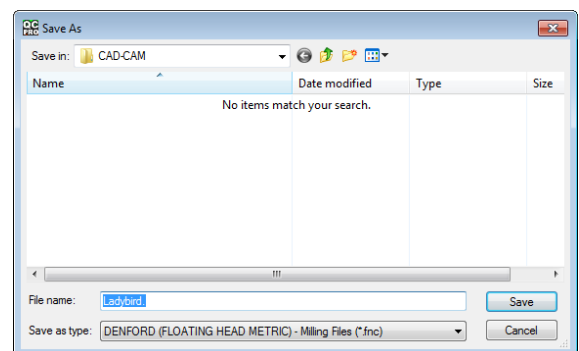


Figure 22

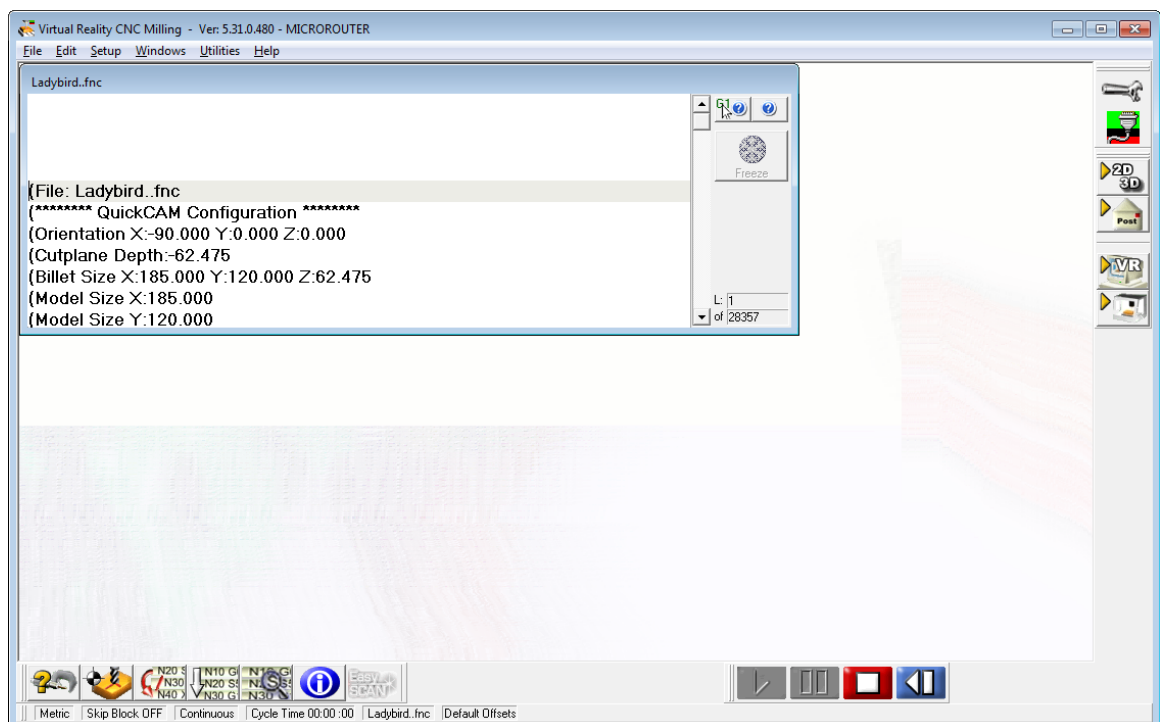


Figure 23

Running the Denford CNC Router

Securing the work

The billet to be machined should be secured to a waste piece of material which in turn is secured on the table by the locking clamp.

For safe operation of the Router the billet is attached to the top of the sacrificial waste piece to give the workpiece extra height and position it within the safe operating parameters of the machine tool.

Tip: In order to stop lifting of the piece when locking in place the clamped side of the sacrificial piece should have a 45 degree bevel machined on the edge.

Setting the machine parameters

Before running the CNC Router there is certain information which must be input.

Referencing the machine position

Once the FNC file is open connect the computer to the CNC machine by clicking on the relevant icon (right menu) as shown in Figure 24.

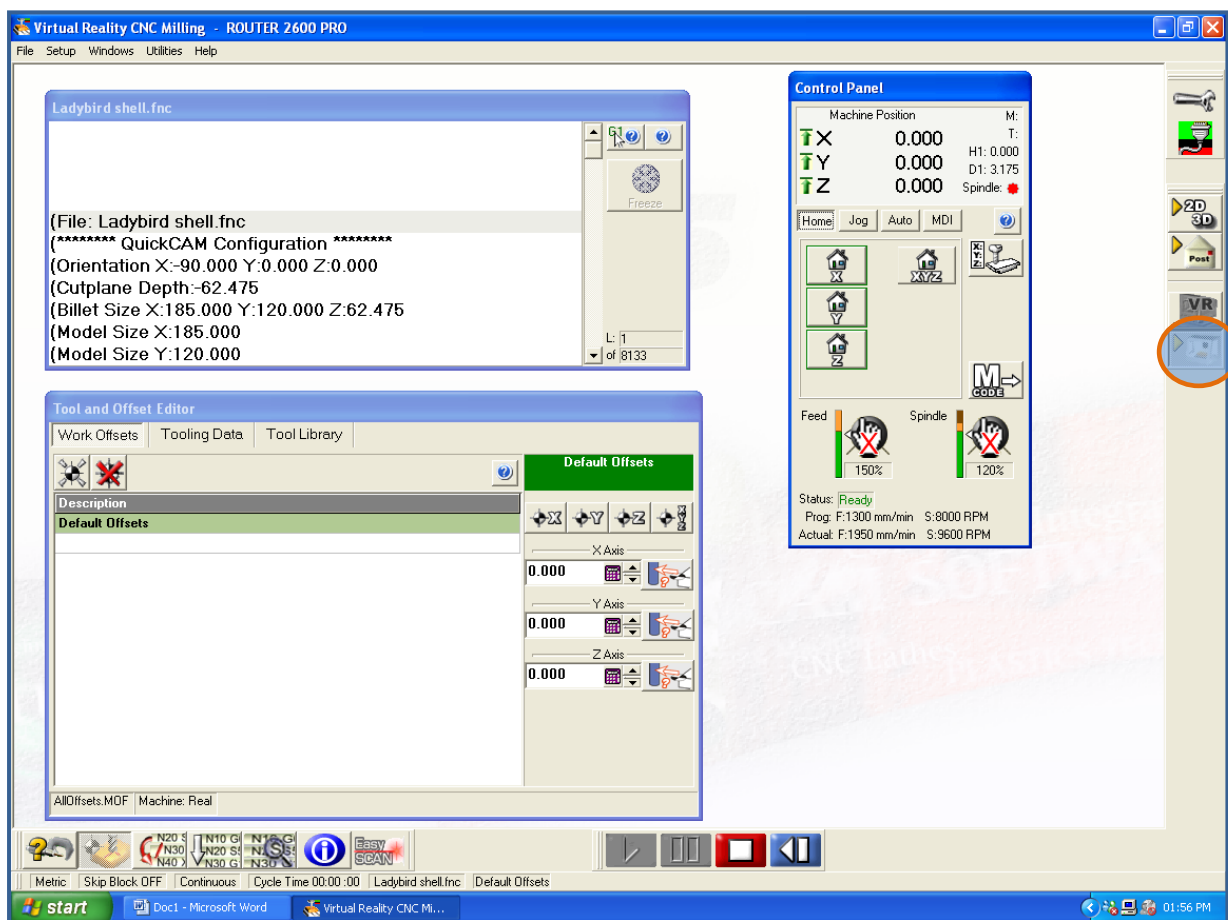


Figure 24

When the link is made with the CNC Router the software will automatically display the machine position relevant to the home position. The home position is the position the machine tool will automatically return to on completion of an operation. By clicking on the home command icon you can set the machine to the home position (Figure 25). The home position is the back top right corner of the work space.

Setting the default offset position

The offset position is the position where you want the cutter positioned before machining begins. This is normally the front left corner on the top of the billet.

To Edit the Offset position first click on the tool and offset editor icon (Figure 25 – Bottom left).

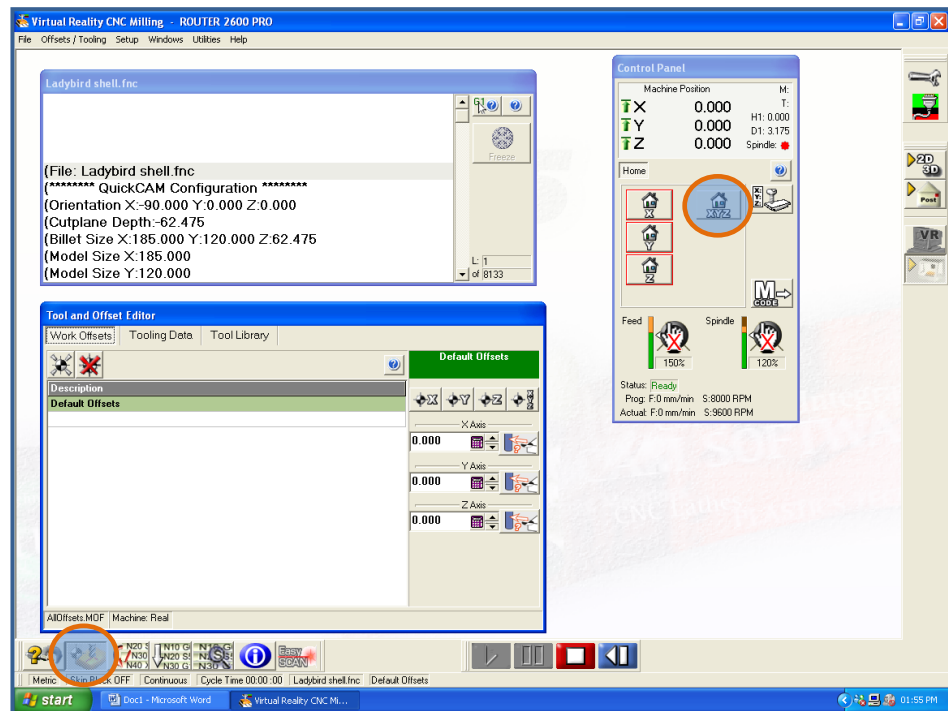


Figure 25

Put the machine in jog mode by clicking on the jog tab in the Control Panel (Figure 26).

This allows the operator to manually move the machine tool to the required offset position by using the left, right, up and down arrows on the keyboard for movement in the horizontal planes and the Page Up and Page Down keys for movement in the vertical plane.



Figure 28

It is important to remember that all CNC Router operations begin at the top front left corner of the work piece and this should now be the new offset position.

Make sure that the central axis of the machine tool is in line with the edge of the work piece as shown (Figure 27 & Figure 27a).

After 'jogging' the machine to the new

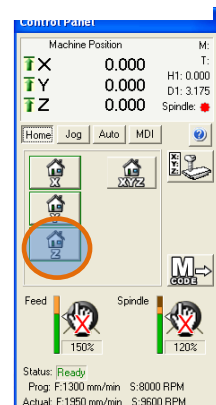


Figure 26

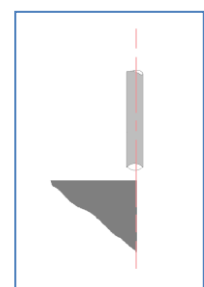


Figure 27a

offset position, the new offset values is set by clicking on the XYZ offset position icon (Figure 28).

These offsets can be saved for repeated use if required.

Selecting the correct tool

It is important that the tool chosen when setting the machine parameters is the same as that in the tool holding device. If it is not, it must be changed now.

To change the tool click on the Tooling Data tab in the Tool and Offset Editor (Figure 29).

If the tool in position one is not the one you wish to use you can select an alternative one from the tool library.

Running the file.

Before running the programme and starting the CNC Router turn on the Turbo option.

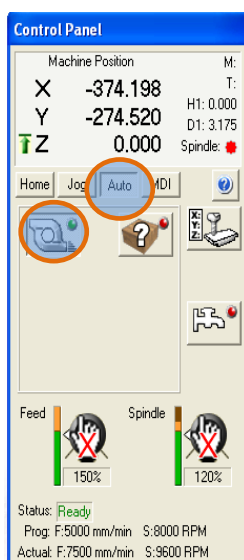


Figure 30

This is done by clicking on the Auto tab and then on the Turbo icon (Figure 30). This will help to automatically vary the speeds of the cutting tool.

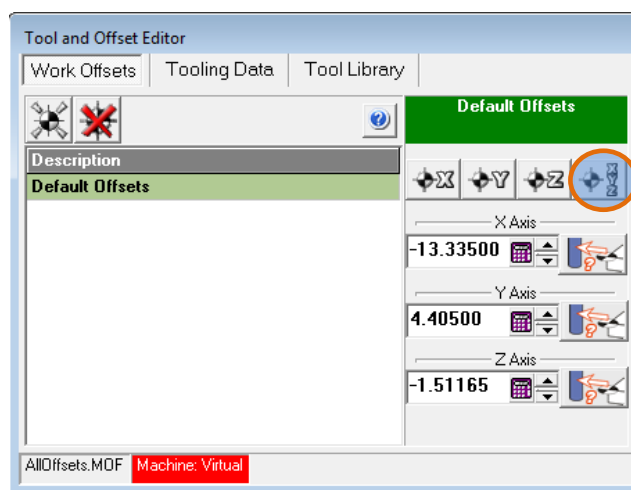


Figure 29

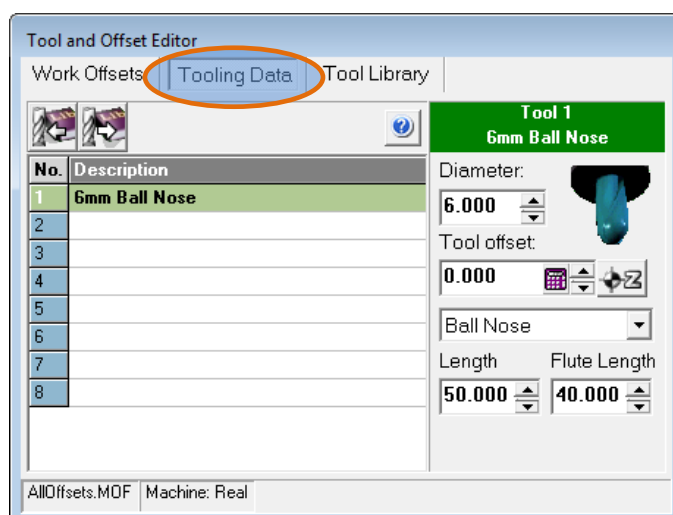


Figure 31

To begin the cutting sequence (Figure 31).

- Click the rewind control button.
- Click on the start button



Figure 32

At this point a warning box will appear to indicate that this is not a simulation but a real file execution (Figure 32).

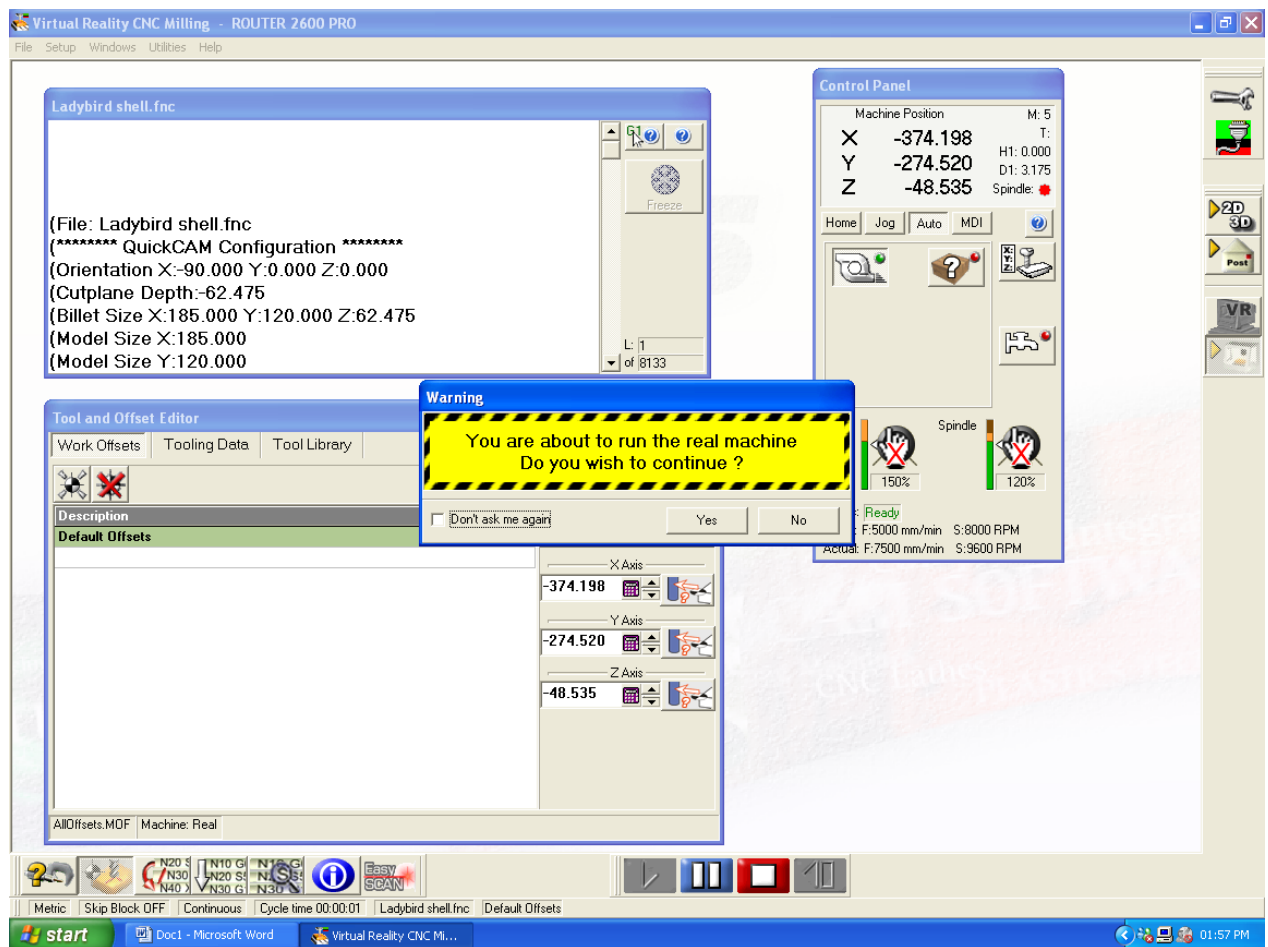


Figure 33

Click "Yes" to begin machining.