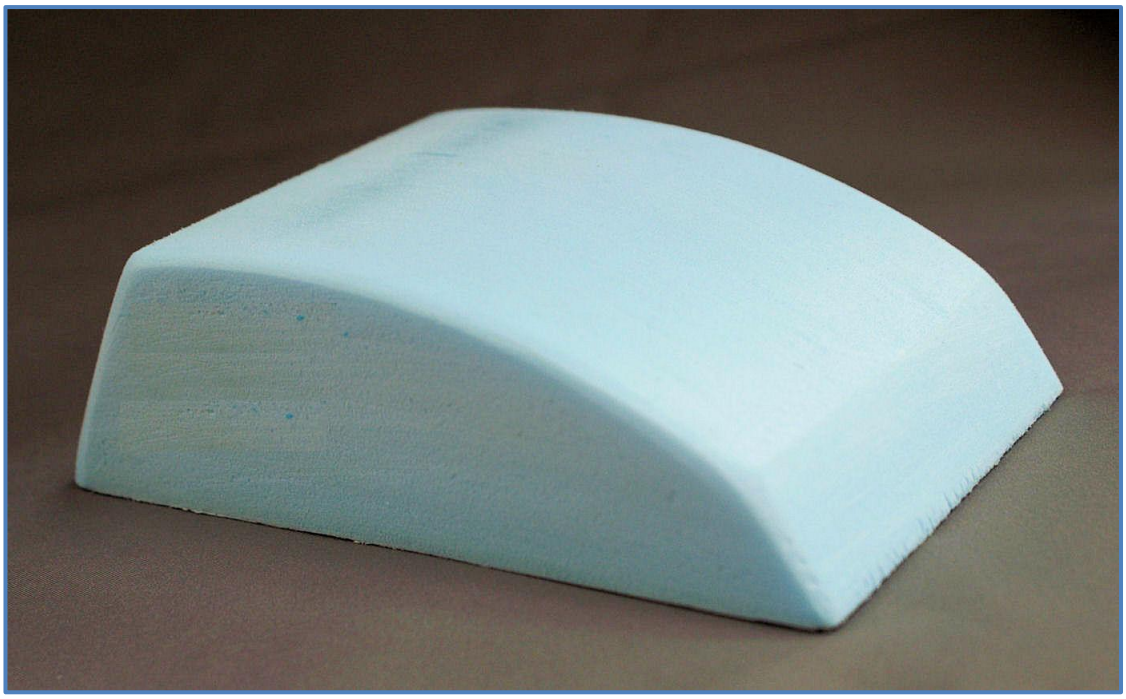
Ladybird Project - Vacuum Mould



Prerequisite Mould drawn and saved as an “STL” file in SolidWorks

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

- Opened an “STL” file
- Set Machining Constraints
- Set up Tools
- Selected a Machining Plan
- Post Process the Program
- Set Machining Parameters
- Set Tool Offsets
- Machined the Mould

TECHSOFT MDX-40- VISUAL TOOLPATH

Step 1: Orientate and Model Size

Open the Techsoft Visual Toolpath programme through Start - All Programs- Visual Tool Path or by double clicking on the icon on the Computer Desktop (Figure 1).

Figure 1

The initial window allows the loading of the 3D file or access to recently opened files.

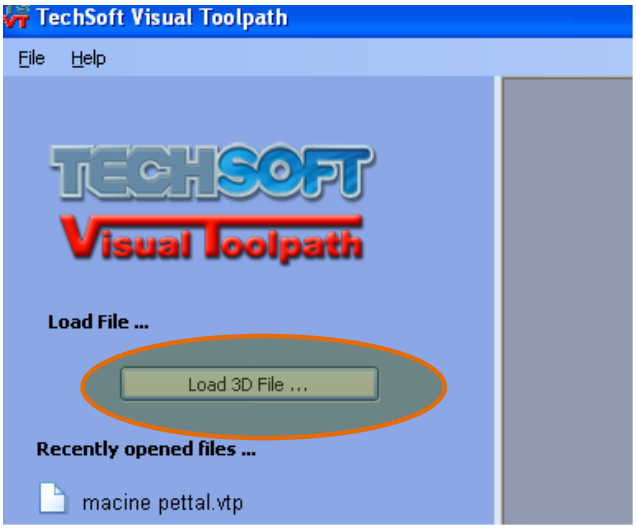


Figure 2

Click on the Load 3D file icon (Figure 2) and browse to find the correct file. The file format must be saved in STL format, a SolidWorks part file will not be recognised.

The file for this exercise is called **Ladybird.STL**.

The model to machine is displayed within the main Visual Toolpath window, but the initial orientation of the model may not always be correct for machining (Figure 3).

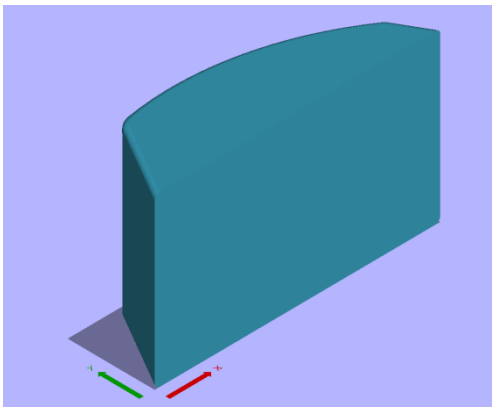


Figure 3

If the orientation of the model is not correct the required top surface can be rotated into position by selecting a different view of the model (Figure 5).

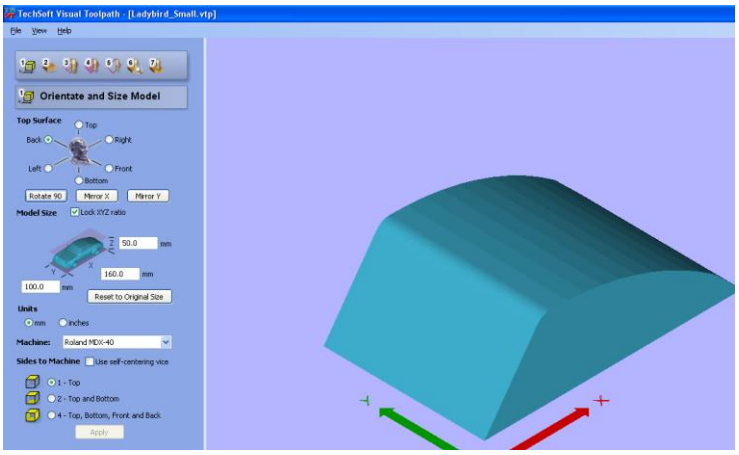


Figure 4

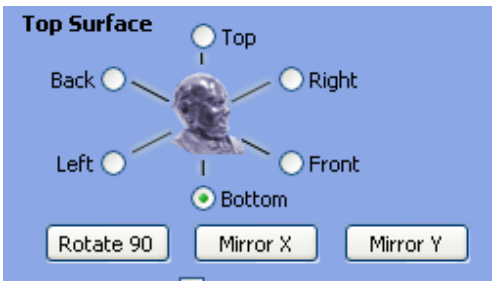


Figure 5

The default size of the model will be the same as the SolidWorks sketch (Figure 6).

If the Lock XYZ ratio is selected then changing any one of the sizes will automatically change the remaining sides in order to maintain the scale of the model.

Un-ticking the box will allow the changing of individual dimensions without affecting the other sides.

- Ensure that the correct machine is selected from the drop down menu (Roland MDX-40).
- Do not select the self-centering vice.

For this exercise the material to be machined will be secured to the front corner of the bed using double-sided tape.

The model will only require machining on one side therefore under the Sides to Machine Options select – Top (Figure 6).

- Click on the Apply button
- Select Next to continue to Step 2.

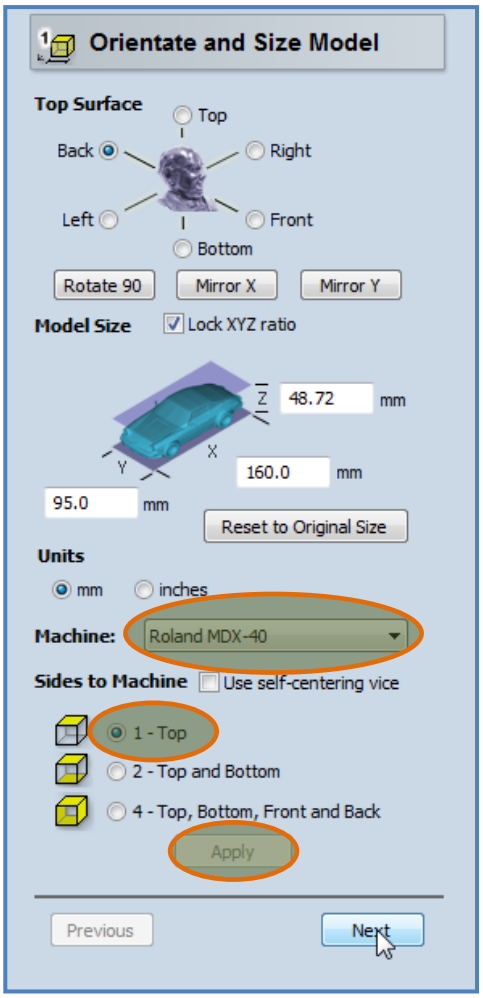


Figure 6

Step 2- MATERIAL SIZE AND MARGINS

The material size is set as the model size and a preview of the model will appear in the main window (Figure 7).

For this exercise it is acceptable to use the dimensions of the model size.

If the model had to be machined on both sides or machined using the self-centering vice then the material size should always be set greater than the model to avoid collision with the jaws of the vice or to create space for tabs.

- Select Next to continue to Step 3.

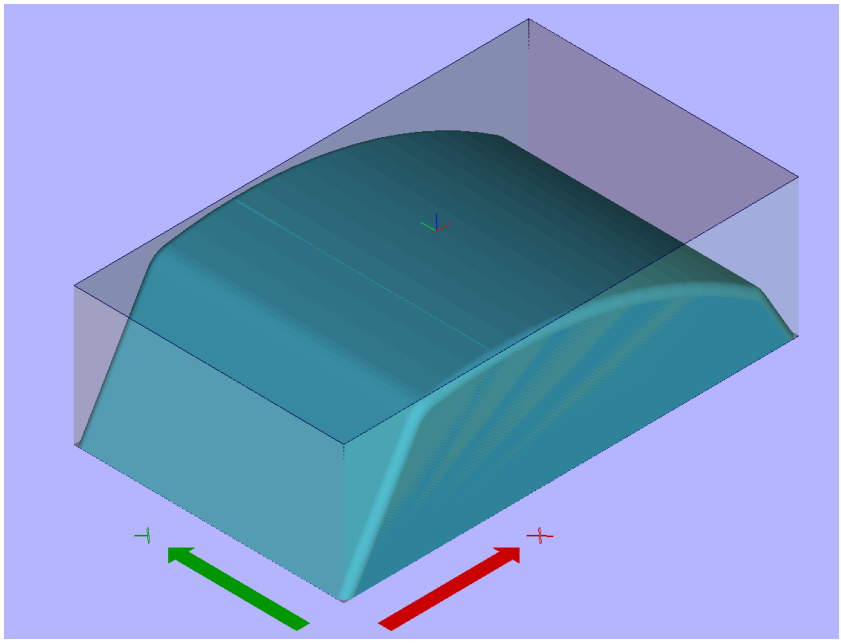


Figure 7

Ensure that the XY origin position is set to the corner of the material (Figure 8). This is important for setting the tool XYZ origin.

- Tick ✓ the Symmetrical & Use Model Silhouette check box (Figure 9).
- Set the width of cut around the model at 4mm.

Figure 8

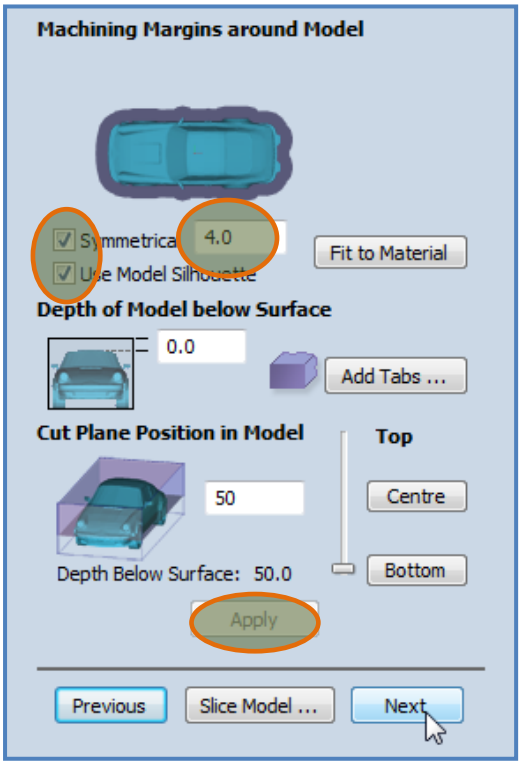


Figure 9

As the model is machined from one side the cutting plane will be to the bottom of the model at 50mm. There are no tabs required for this exercise (Figure 9).

- Click Apply and Next.

A warning message will appear (Figure 10) indicating that the machining margin's are outside the material size. This is a not a problem in this case as we are not using clamps to secure the work-piece.

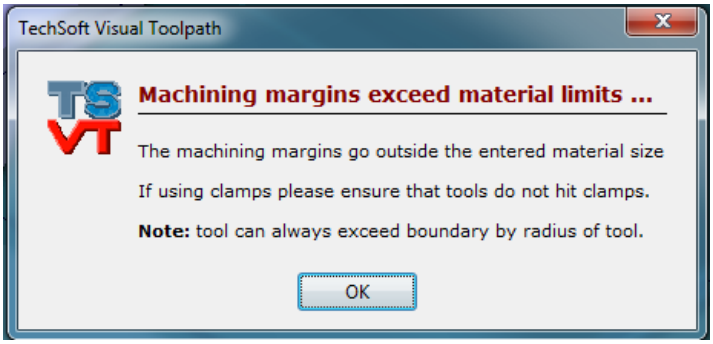


Figure 10

Step 3- ROUGHING TOOLPATH

The Techsoft MDX 40 generally processes the parts in two cutting stages, the roughing cut and the finishing cut.

The roughing cut is optional but aside from a very small model it is nearly always desirable to create one.

The finishing cut can use the same cutting tool as the roughing cut or can use an alternative tool.

The core tools available from Techsoft for the MDX40 are the 2mm, 3mm and 4mm diameter ball nose or slot drill cutters.

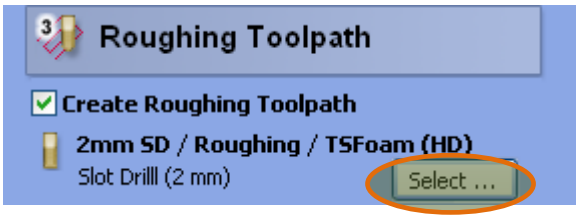


Figure 11

- The default roughing tool is a 2mm slot drill, but a very effective tool is the 4mm ball nose.
- Select Next to continue to Step 2.

To change the roughing tool click the "select" button (Figure 11).

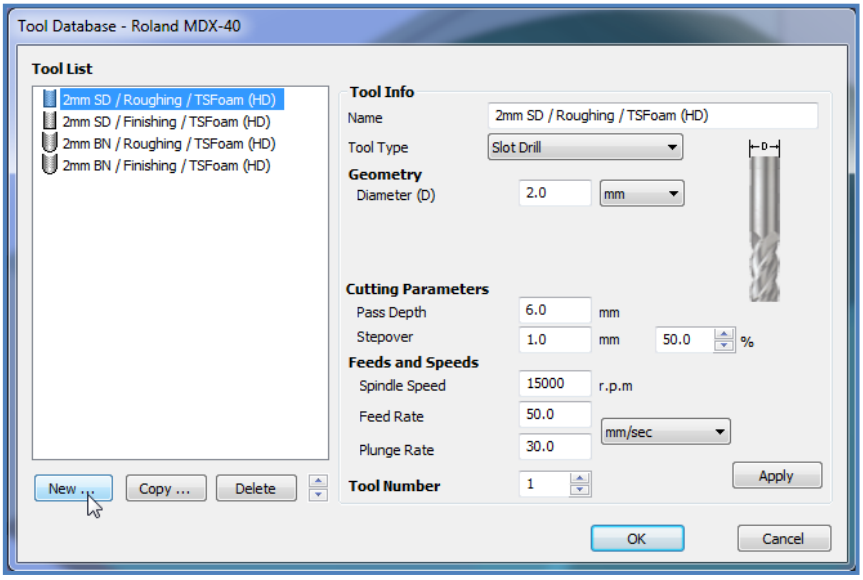


Figure 12

The 4mm ball nose can be selected from the Tool List on the left.

If tool is not present it can be added manually as follows:

- Click on New (Figure 12).
- Add the Tool information as shown in Figure 13.

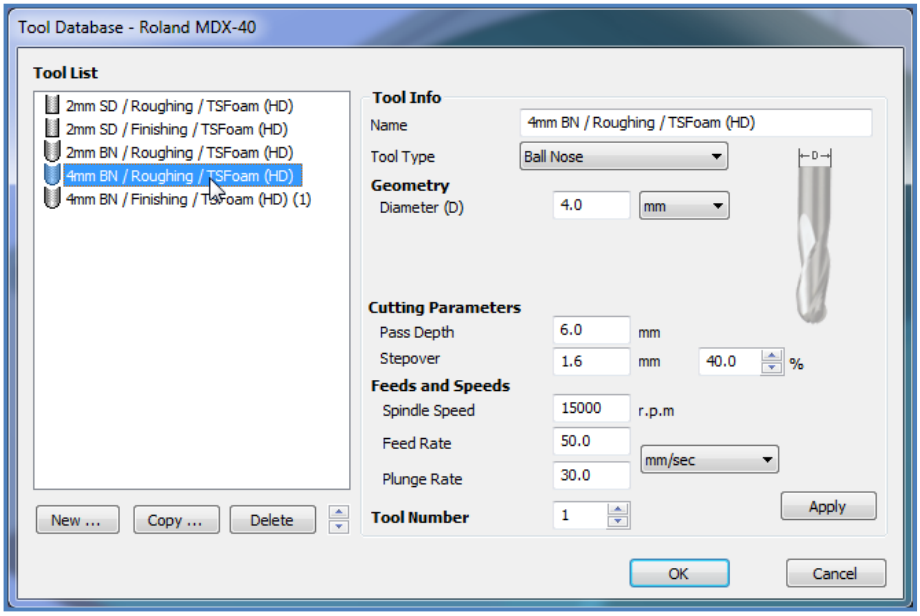


Figure 13

- Click on the Apply Button
- Click on the OK Button

Note: The cutting parameters will need to be changed to match the material to be manufactured. These feed rate, step over and pass depth settings are for a model made from soft foam.

Set the rapid clearance gap to 5mm for this exercise. If using the self-centering vice this value should be increased to avoid collision with the jaws of the vice as the tool hovers over the work piece.

The machining allowance is set to .2mm. This is how close the roughing cut is to the finish cut. It is important to also select the step strategy for the fastest machining time (Figure 14).

- Finally click Calculate.

The toolpath will be generated and an estimated time for machining displayed (Figure 15 & 16).

- Click Next.

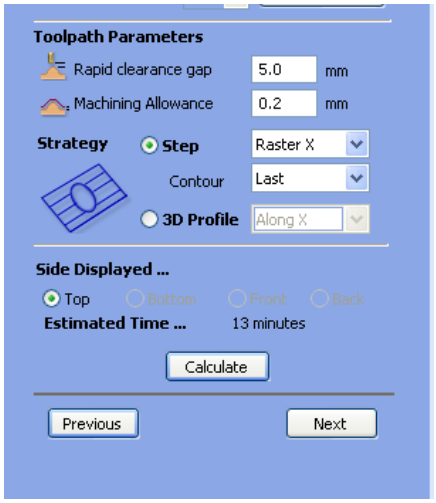


Figure 14

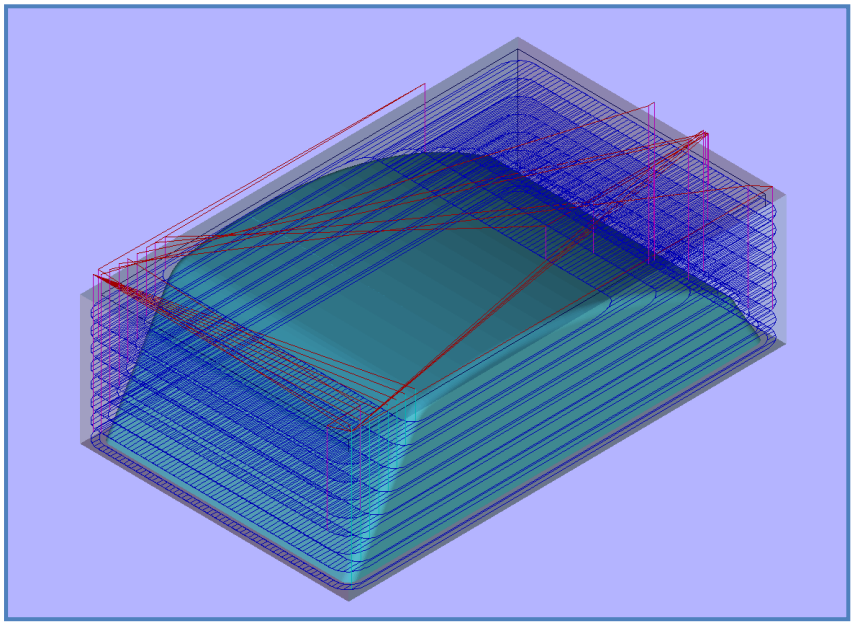


Figure 15

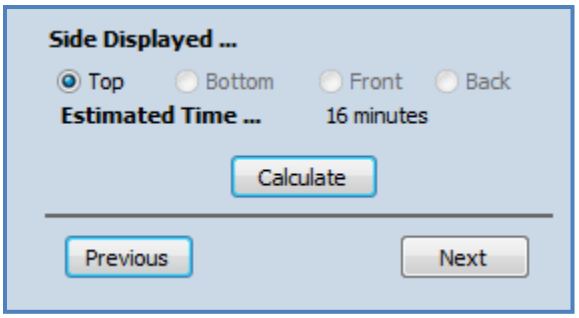


Figure 16

Step 4- FINISHING CUT

In this case the finishing cut will be performed using the same cutting tool. This will speed up the machining time and eliminate the need for a tool change.

Similar to the roughing cut the 4mm ball nose tool is selected with a fast feed rate as the material used is soft and easy to machine. A slightly smaller stepover of 1.2 mm is used to generate a better finish (Figure 17).

The rapid clearance is the same as for the rough cut, allowing the tool to hover 5mm above the piece.

The raster angle is the path of the cutting tool relative to the x axis. Using an angle of 45^0 achieves a better finish on the piece. To achieve an even better finish a second pass can be created at 90^0 to the first, but this would have little benefit in this exercise because of the material being used.

- Click Calculate to generate the toolpath and an estimated time for the finish cut will displayed.
- Click Next to move to step 5.

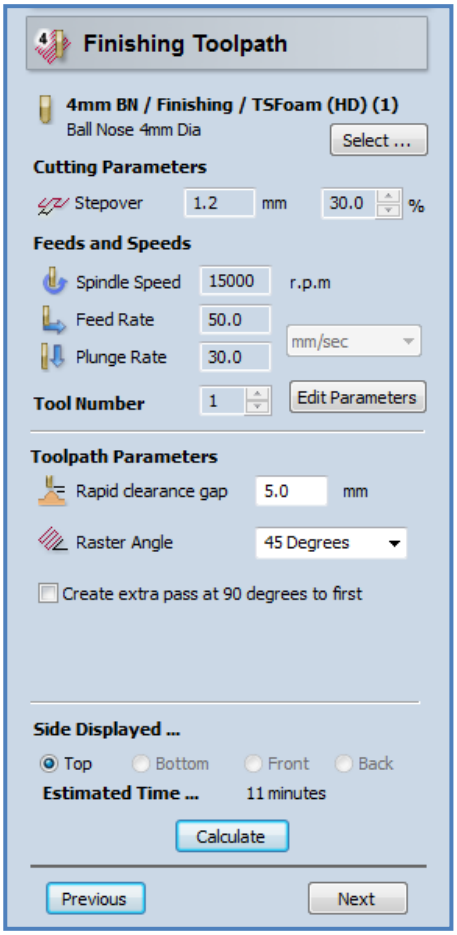


Figure 17

Step 5- CUT-OUT TOOLPATH

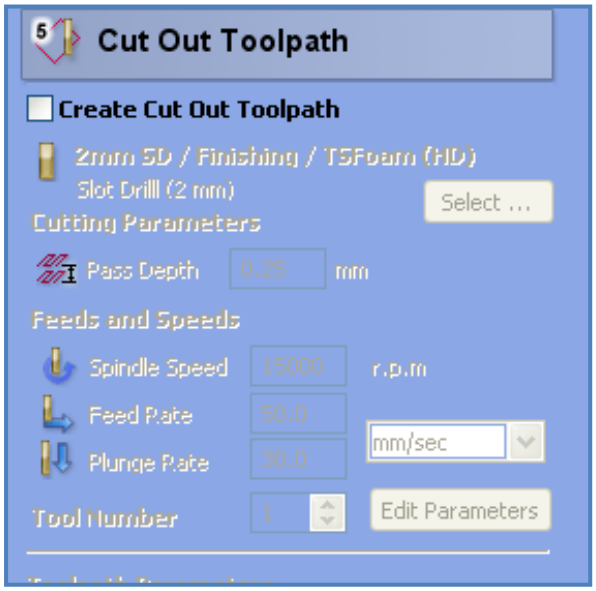


Figure 18

This optional step is to remove the tabs that may have been added to a double sided model. It is an additional cutting step that runs after the finishing cut.

This option is rarely used as there is a danger using this toolpath of the piece coming into contact with the cutting tools as the tabs are being removed.

It is best therefore to remove the tabs manually with a band saw or knife.

- Click Next to proceed to step 6.

Step 6 PREVIEW

This allows for an animated view of the machining process.

Both the roughing and finishing cuts can be observed to ensure that the desired outcomes will be achieved.

There is a list of supplied materials that can be selected to give a more realistic image (Figure 19).

This step is optional and does not have to be viewed in order to progress to Step 7.

- Click Next to proceed to step 7.

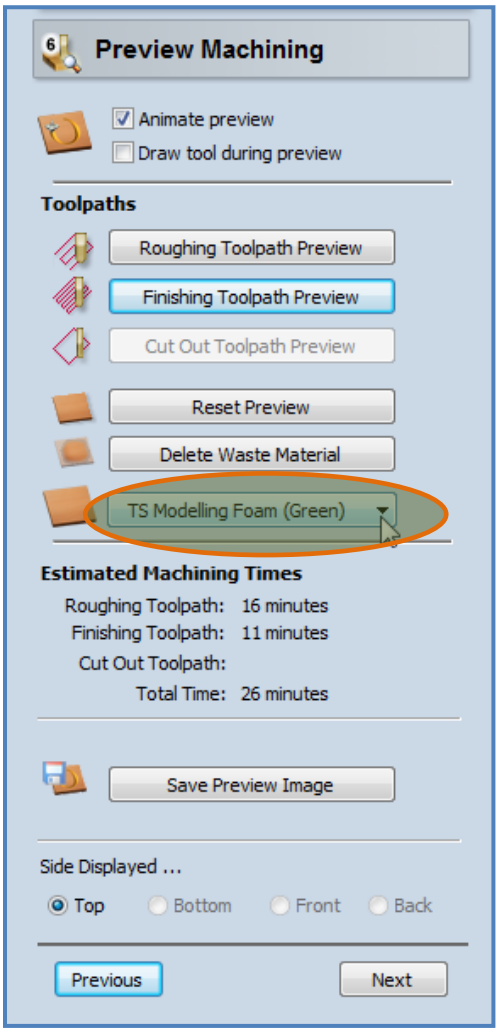


Figure 19

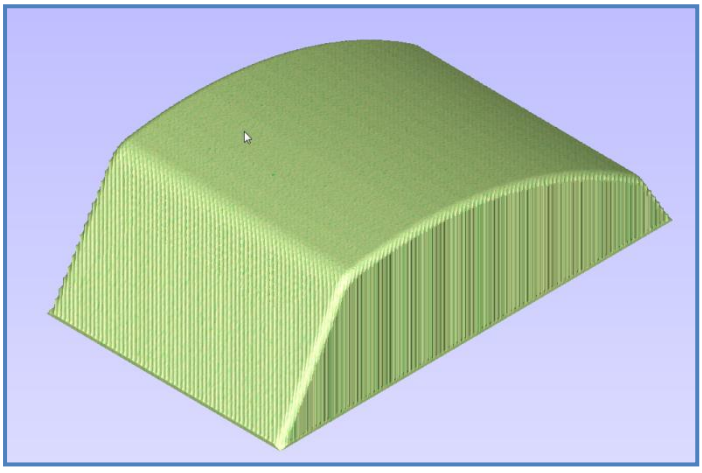


Figure 20

Step 7- OUTPUT TOOLPATHS

The final stage is the manufacturing of the product.

However before it is possible to begin manufacture a number of issues must be addressed.

Firstly it is necessary to select the correct print driver.

- Ensure that the Roland MDX-40 is the default Print Driver (Figure 21). The Print Driver must be selected and must match the Machine. If this is not the case click the "change" button and select the correct driver from the menu list.

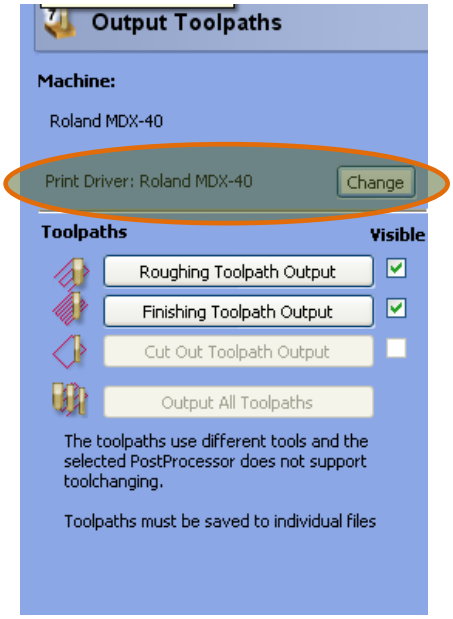


Figure 21

The two Toolpath options available in step 7 are outputting of the Roughing Toolpath Output and Finishing toolpath Output (Figure 21).

Care must be taken when beginning the manufacture of the model. The Techsoft machine is not intuitive and will not detect incorrect tooling or the wrong sequencing of cuts. Therefore it is possible to output the finishing toolpath before the roughing toolpath, which would have serious implications for the cutting tool, machine and material.

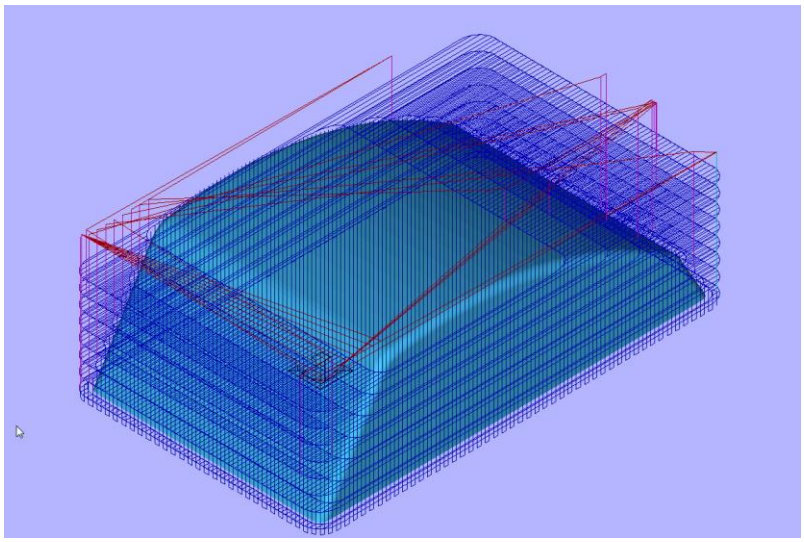


Figure 22

MATERIAL PREPARATION AND HOLDING

At this stage the billet should be fixed to the front of the sacrificial bed using double sided tape (Figure 23).

Ensure that enough tape is applied to prevent the billet lifting and crashing into the tool during machining.

See page 12 for information on Cutting Tools.

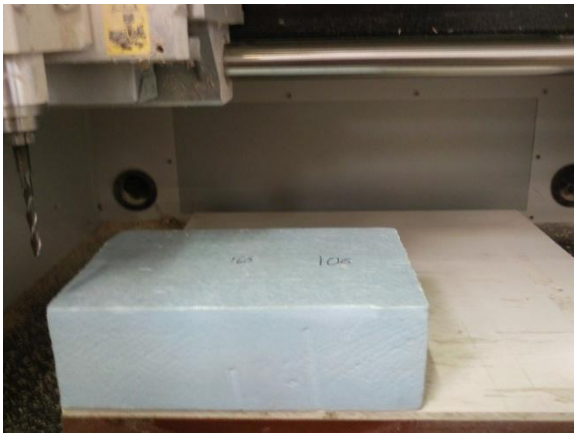


Figure 23

MACHINE CALIBRATION

Once the MDX 40 machine is turned on it automatically checks if it's in Engraving (Modelling) or Scanning mode. The relevant mode will become illuminated on the display (Figure 24).

The machine then completes a standard procedure, moving the bed into the home position and bringing the bed fully out to the View position.

As the bed moves the view button will flash and will remains on when the bed has come to a complete stop.

- Press the View button to get the bed to return to the machining position.

Figure 24

ZEROING THE AXES

It is vital that the X,Y and Z axis are all set to zero in their correct positions.

These settings can be viewed and reset using the Roland Control Panel (Figure 25).

Figure 25

Installing a shortcut for the Roland Control Panel on the Computer Desktop (Figure 26).

- Click start followed by "Printers and Faxes".
- Right click on the "Roland MXD 40" and select "Properties".
- Under the "General" Tab select "Printing Preference".
- Select the "Options" Tag.
- Click on the "Create Shortcut" icon.

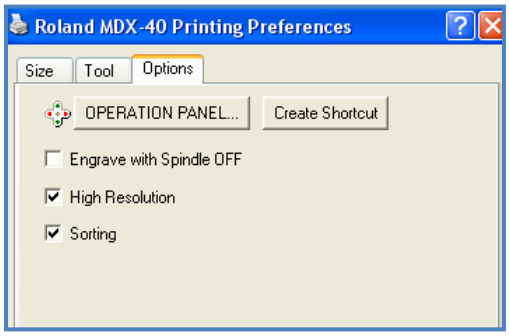


Figure 26

The control panel icon will now appear on the Desktop.

- Open the Roland Control Panel (Figure 25) by double clicking on its Desktop shortcut.

The origin of the X &Y axis is when the head is moved completely to the left and to the front of the bed (Figure 27).

Figure 27

It is important to check these settings prior to operating the machine.

- Move the head and the bed using the X &Y controls on the Roland Panel (Figure 28).

When the head is left and at the front corner of the bed the X & Y read out should be 0.00, and 0.00

If this is not the case reset the X and Y Origin to 0.00 at this position.

The Z origin is always set to the top surface of the work piece.

- Using the control panel move the cutting tool over the piece.
- Drop the tool down until it touches the top of the work piece.

The Z axis can be controlled from the computer screen or by using the Z axis buttons on the machine.

- When the tip of the tool is in contact with the work piece, select "set the Z origin here" and click "Set".

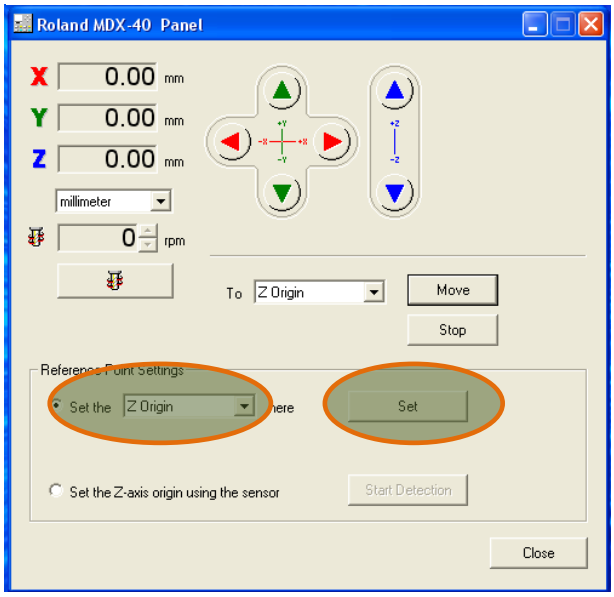


Figure 28

The Z readout should now be 0.00.

This Z origin reset must always take place when a tool is changed or a different size billet is placed on the worktable or in the vice.

Check List Before machining:

- Material Billet/ Work piece is the correct size and shape
- Billet is correctly positioned and securely attached to table
- The correct cutting tool for the Roughing Cut is in position and tightened
- The XY Origin is set correctly
- The Z origin is set to the top of the work piece

When checklist is complete return to Step 7

Step 7 continued

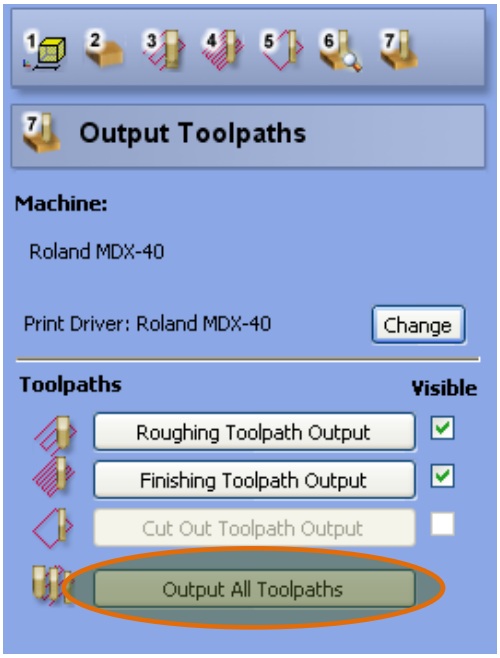


Figure 29

The 4mm ball nose cutter is now in position and zeroed to the work piece.

The two output options are the Roughing and Finishing toolpaths, as both cuts are useing the same cutting tool the "Output all Toolpaths" will be active, allowing a single command to complete the entire process.

- Click on the "Output All Toolpaths" button.

The spindle will start and the head will move to the XY origin and drop to the Rapid Clearance distance.

(At this point always be ready to hit the emergency stop; this is where most tool crashes occur).

The router will now proceed with the rough cut.

Figure 30

The work can be stopped and viewed at any time by pressing the "View" button (Figure 30). The spindle will stop and the bed will move forward to the front of the ma **extremely important to wait for the bed and the cutting head to stop moving before opening the guard.** The view button should have a constant green light before opening the guard.

Note: If the guard is opened before the constant green light is on the program will have to be restarted.

- When the view light remains on, open the guard and to view the work or remove the dust.
- Close the guard and press the "View" button again to restart the program.

When the "Output All Toolpaths" is selected the program will proceed to complete both cuts seamlessly (unless the program is paused manually).

When the program stops, press the view button to bring the table forward and remove the finished artefact (Figure 31).

Figure 31

TOOLING

The standard tools that come with the MDX40 machine are the 2mm slot drill and 2mm ball nose cutter (Figure 32). These have a shank length of 20mm and an overall diameter of 4.5 mm.

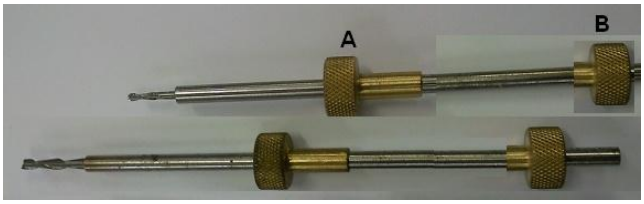


Figure 32

Figure 34

These cutting tools do not fit directly into the 6mm collet on the machine head but instead have brass adaptors (A&B) as shown in Figure 32. The larger brass adaptor A (Figure 33) is inserted into the collet and locked in place on the machine head (figure 34). The cutting tool is then placed down through the head and locked with the grub screw. Adaptor (B) is then placed on top and locked to stabilise the tool and eliminate vibration (Figure 34). Adaptor (B) is only attached to the tool and is a way of maintaining equal lengths between tools. For example when changing between tools for a rough and finished cut there would be no need to reset the Z axis on the machine if both tools were equal length from the tip of the tool to adaptor (B).

Figure 33

Figure 35

These initial cutting tools supplied can be restrictive due to their shank length and diameter. Techsoft offer a range of additional tooling from 2mm to 4mm, with both long and short shanks which offer a wider range of machining options (Figure 35).

A 4mm long reach slot drill and ball nose cutter and a 2mm long reach ball nose cutter is a good combination to have in stock. Cutters are available from Techsoft at:

http://www.techsoft.co.uk/materials_tools_milling_tools_mdx40.htm.

Figure 37

Unlike the standard tools long reach cutters are held directly in the machine head by the collet (Figure 36).

Figure 36

There is no requirement for any brass adaptors.

Tools are fixed by loosening the collet with the spanners provided and inserting the relevant tool and tightening the collet once the tool height is correct (Figure 37).

These tools are more robust and allow for quicker processing of more intricate parts.

The Z axis must be zeroed each time a tool is inserted.