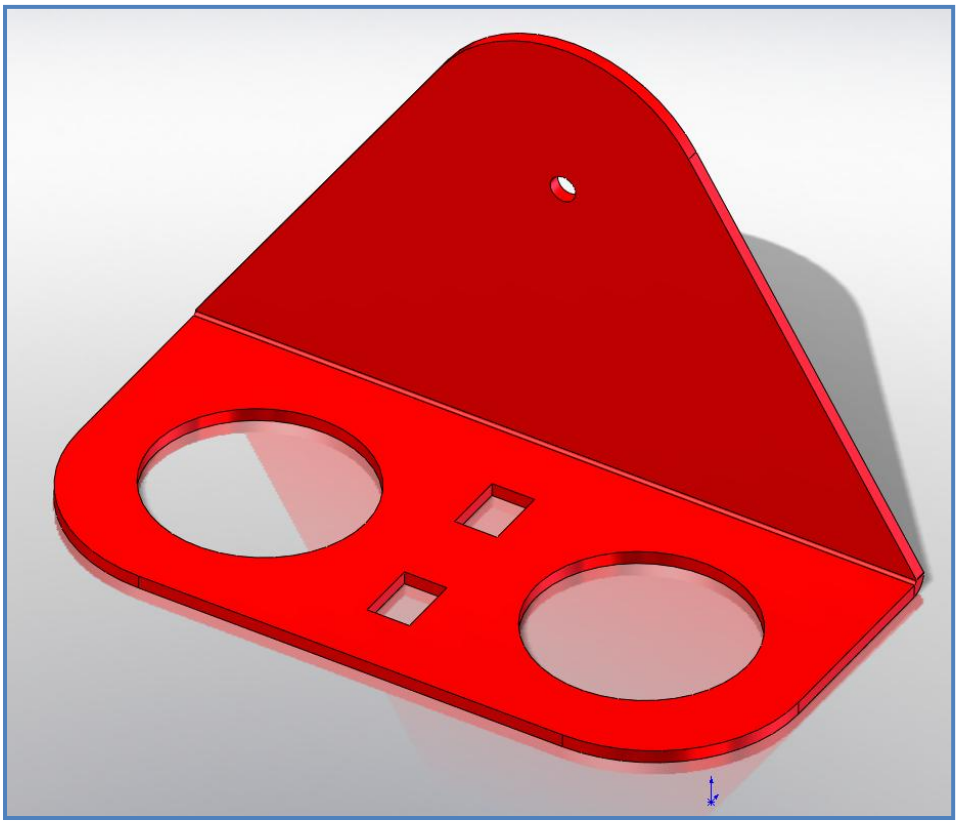
Toothbrush Holder Project – 2D Machining



Prerequisite Toothbrush Holder drawn and saved as a “DXF” file in SolidWorks

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

- Used the Denford Virtual Milling V5 software
- Imported DXF File format into VR Milling V5
- Set up Denford 1000 Pro for Machining
- Created a Toolpath
- Outputted a Toolpath
- Setup Tool Set Points
- Machined Component

Denford 1000 Pro- Virtual Milling V5

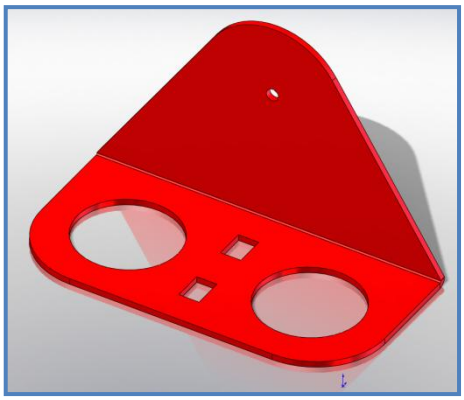


Figure 1

Toothbrush Holder

File Format

The Toothbrush Holder was generated as a 3D part file (Figure 1) in Solidworks and saved in the DXF file format.

Getting Started

Double Click on the VR Milling icon on the desktop (Figure 2) or from the Start menu select -> All Programs-> Denford > VR Milling V5 > VR Milling V5.

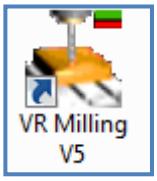


Figure 2

VR Milling Opening Screen (Figure 3).

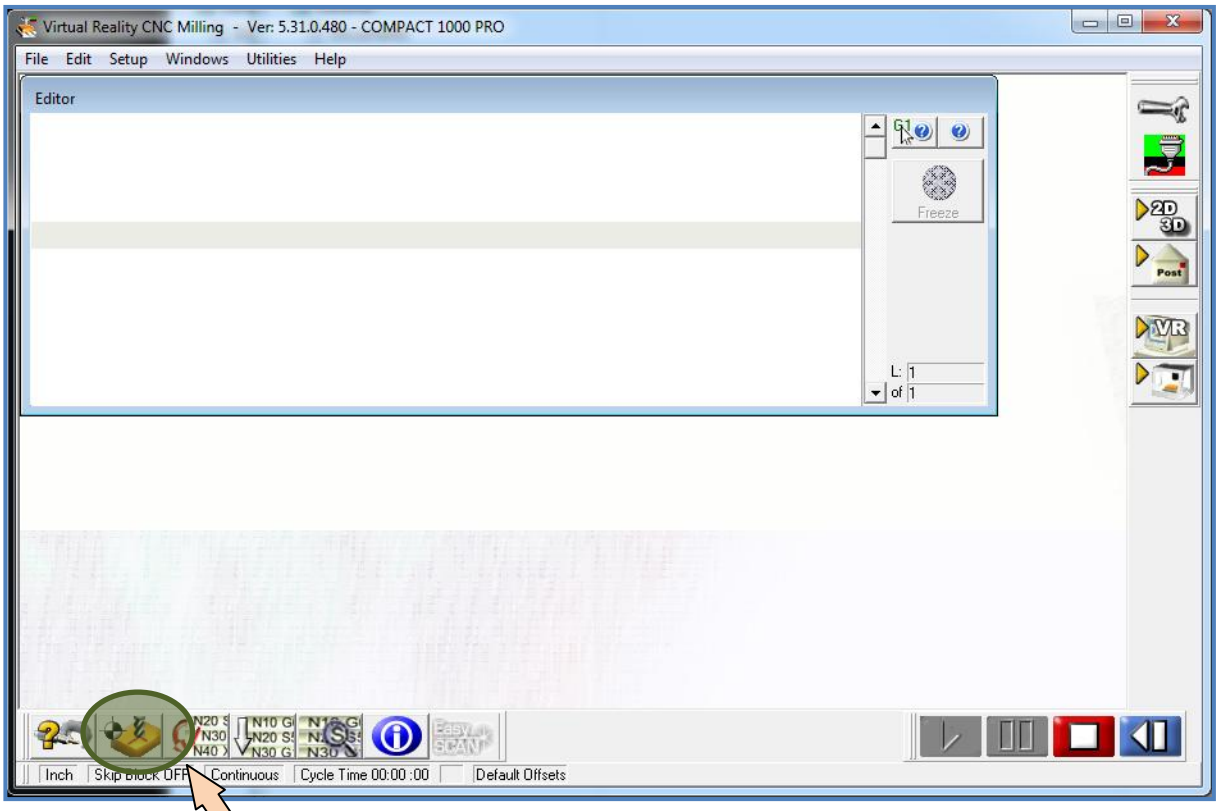


Figure 3

Figure 4

Selecting the Required Cutter

Before opening the machining program select the required cutter or cutters.

- In the bottom left of the VR Milling Window select the Show Tool & Offset Editor Window icon (Figure 3 & Figure 4).
- In the Tool and Offset Editor window select the "Tooling Data" tab by clicking on it (Figure 5).
- Click on the Load a Predefined Tool from the Tool Library icon (Figure 5).
- In the Library Tool Selector window select the 3mm Slot Drill (Figure 6) and click OK.
- Click on the Tool & Offset Editor Window icon (Figure 4) to close the window.

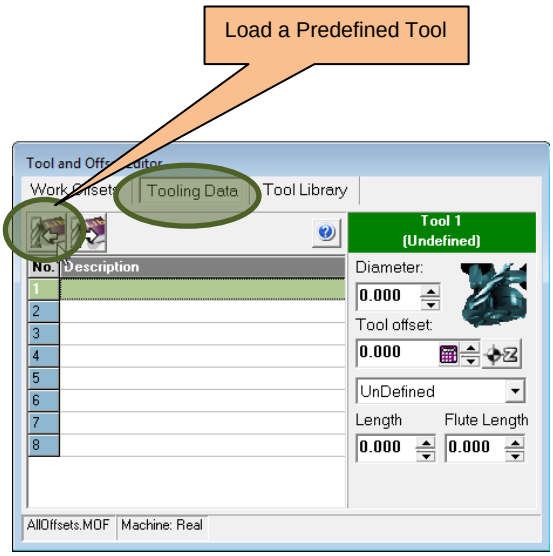


Figure 5

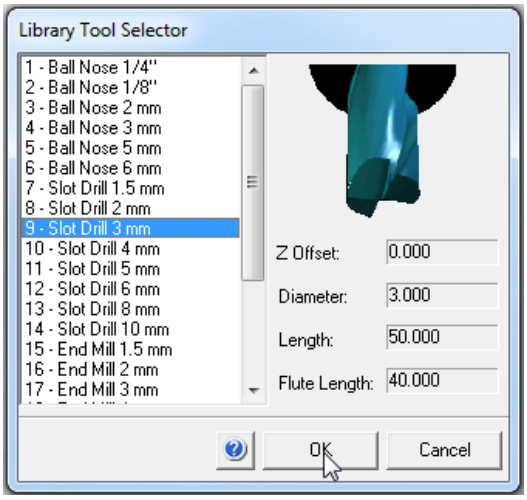


Figure 6

Program Setup

Before opening the machining program set the units to metric (Figure 7)

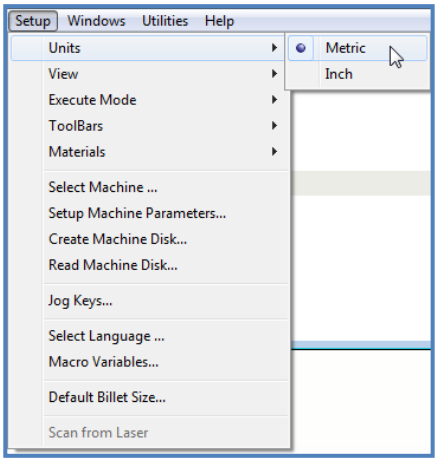


Figure 7

Opening the File

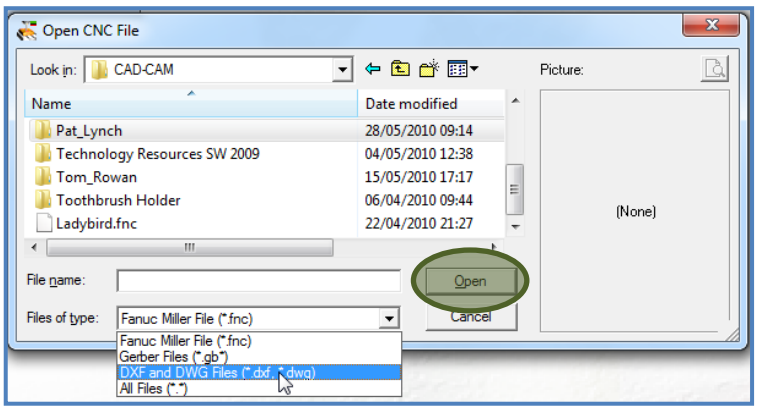


Figure 8

These instructions refer to a part file modelled and saved as a "DXF" file in Solidworks.

Note: A Solidworks "prt" file cannot be loaded into the VR Milling V5 program.

- Choose: File > Open.
- Select the correct file type (Figure 8)

- Locate the "**Toothbrush_Holder**" file.
- Click on the Open button to open the file.
- Choose the required material (Figure 9).
- Click the OK button to proceed.

The "Drawing Import CAM" window will open (Figure 10).

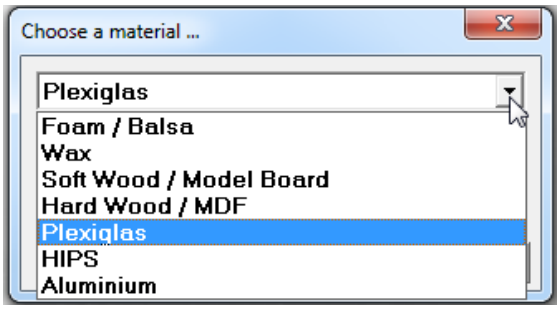


Figure 9

- Check that the selected Import File Units are set to metric (Figure 10).

Note: The scale of the piece may be distorted if metric is not selected.

- Click the Next Button to proceed.

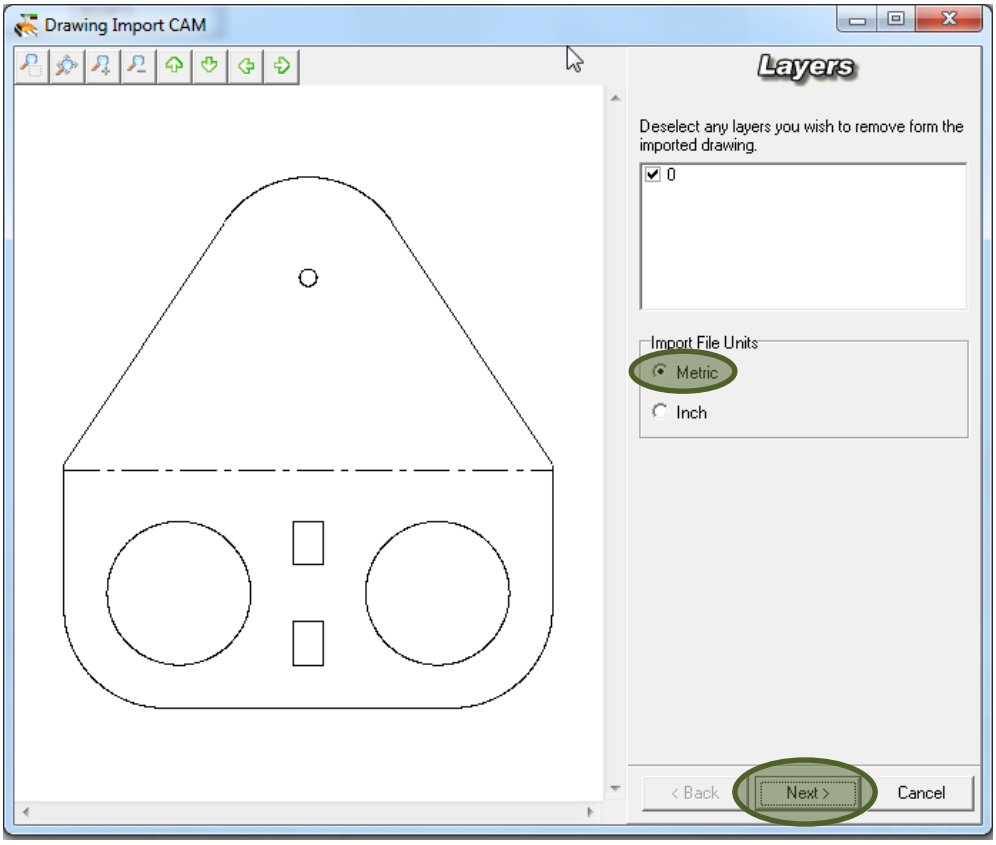


Figure 10

Defining the Billet

- In the Define Billet Window un-tick the "Use Drawing Origin" box (Figure 11).
- Click on the Auto Size Button (Figure 12).

Figure 11

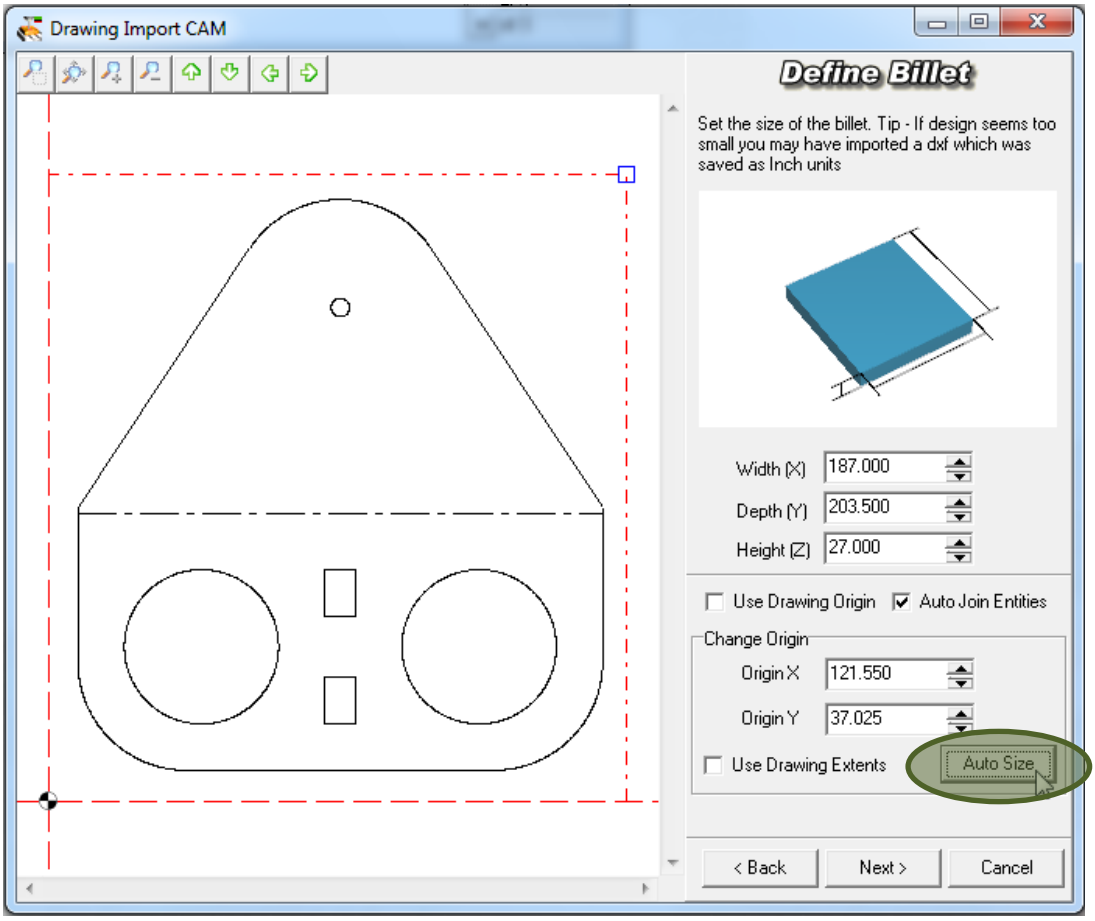


Figure 12

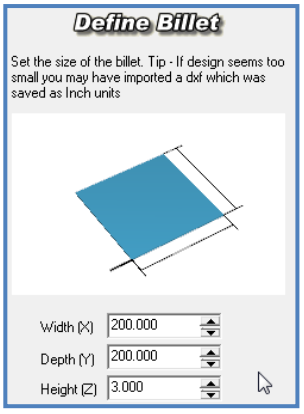


Figure 13

- Set the Billet size to 200 x 200 x 3 (Figure 13.
- Click the Next Button to Proceed.

The Machining Plan window will appear. (Page 6 - Figure 14)

- Click on the Outside Offset Cut tab.
- Check Cut settings as shown in Figure 15 and change if necessary.
- Click anywhere on the outside profile lines of the Toothbrush holder.
- Click on the Apply button and if correct click on the OK button.

The selected Machining Plan will be added to list of Machining Plans (Figure 15a).

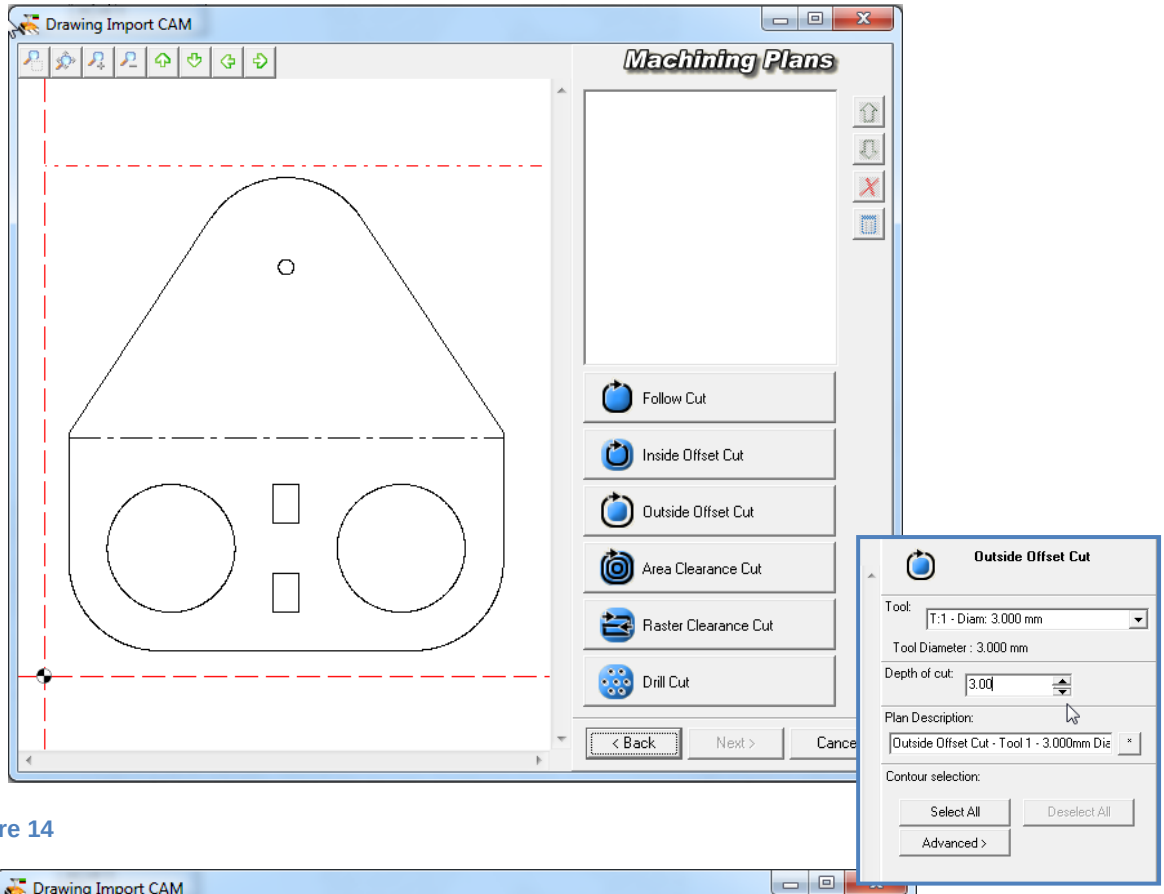


Figure 14

Figure 15

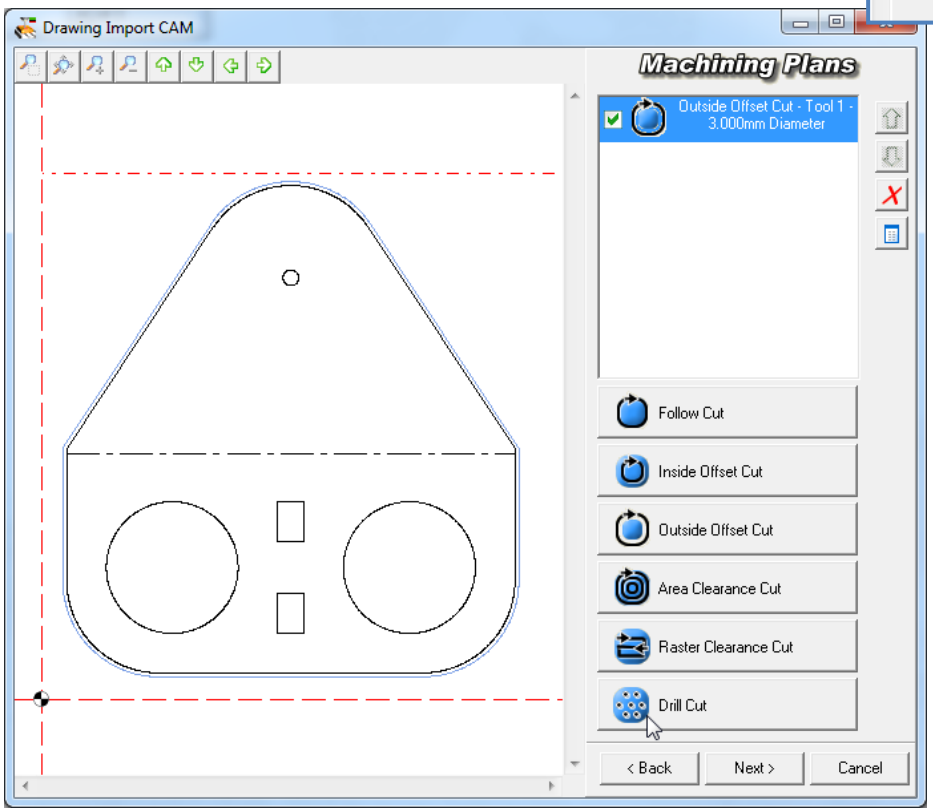


Figure 15a

- Repeat the procedure to select the Inside Offset Cut in order to machine the 3 circles and 2 internal slots (Figure 16).

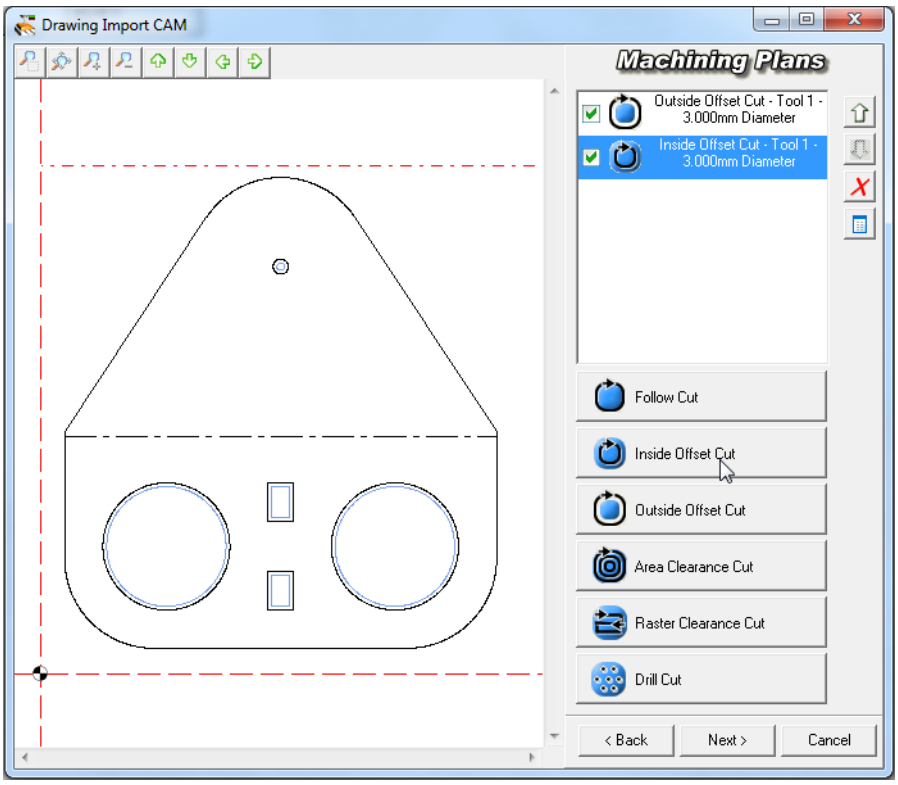


Figure 16

- Click on the Next button to continue.

Machining instructions are now ready for post processing (Figure 17.

The instructions will be output as an "fnc" file which will be recognised by the VR Milling program.

Saving the file.

- Select the location for the file by clicking on the button to the right of the file name (Figure17).
- Include the file name.
- Click on the Save button.
- Click the OK button to continue.

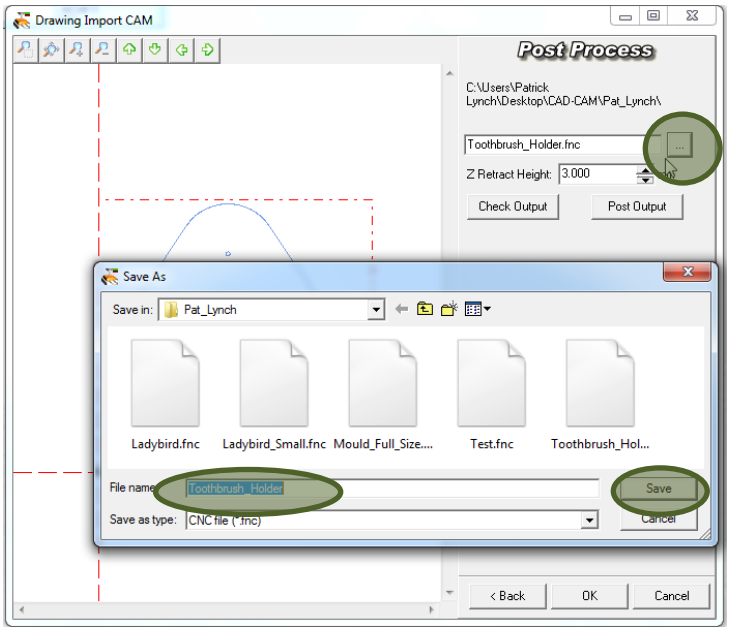


Figure 17

The Machine code will be loaded automatically into the VR Milling Program.

Running the Denford CNC Router

Securing the work

The material to be machined is attached to the sacrificial bed by special doubled sided tape. The Sacrificial Bed is secured to the table by the locking clamp.

The material is attached to the top of the sacrificial bed to give the work-piece extra height and position it within the safe operating parameters of the machine tool.

Tip: In order to stop any lifting of the bed when locking the clamp the sacrificial bed should have a 45 degree bevel machined on the edge being clamped.

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 activate the link between the computer and the CNC Router by clicking on the relevant icon (right menu) as shown in Figure 18.

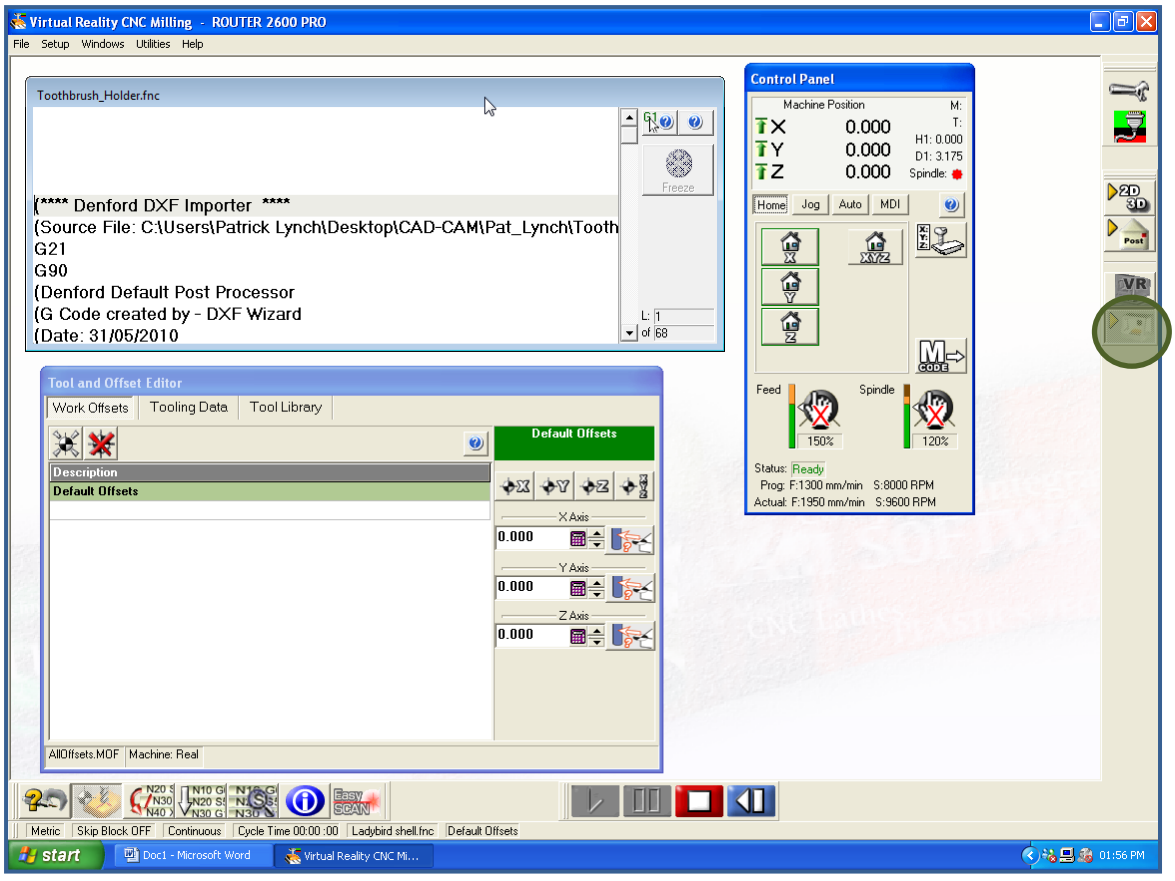


Figure 18

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 19).

Setting the default offset position

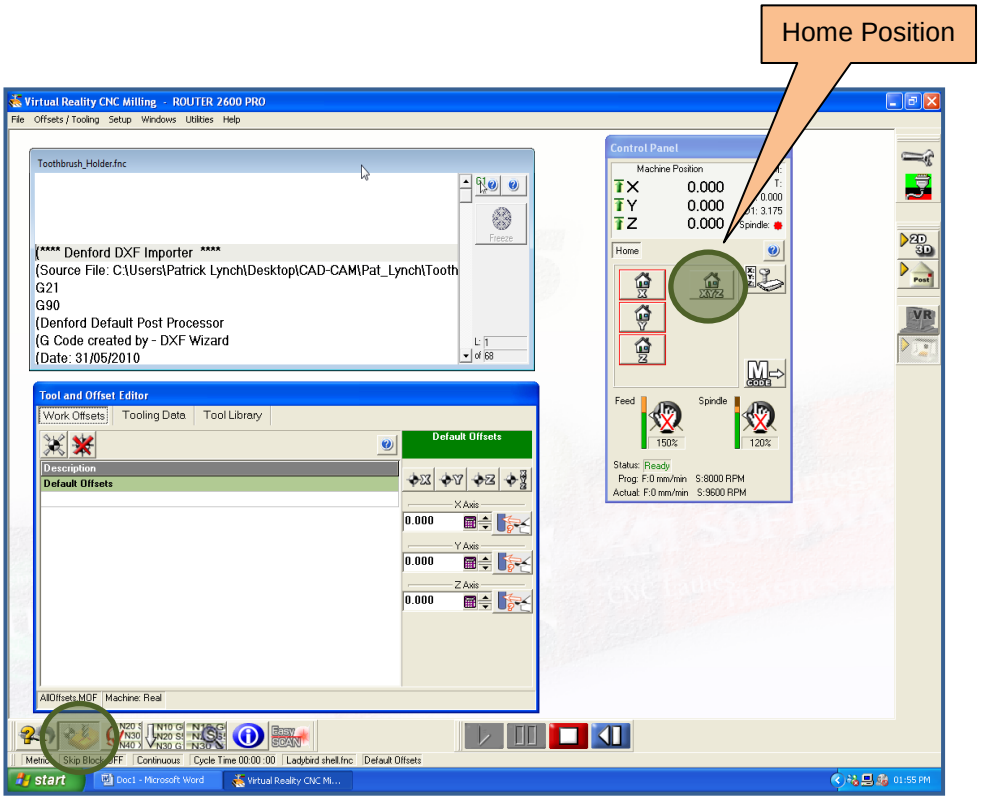


Figure 19

The offset position is where the cutter is positioned before the machining begins. This is normally the front left hand corner on the top of the work-piece.

To Edit the Offset position, click on the tool and offset editor icon (Figure 19 –bottom left).

Put the machine in jog mode by clicking on the jog tab in the Control Panel (Figure 20).

This will allow the operator to manually move the cutting tool to the required offset position by using the left, right, up and down arrows on the keyboard for movement in the horizontal planes (X & Y coordinates) and the Page Up and Page Down keys for movement in the vertical plane (Z coordinates).

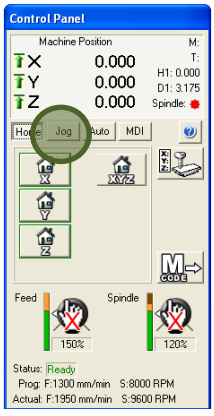


Figure 20

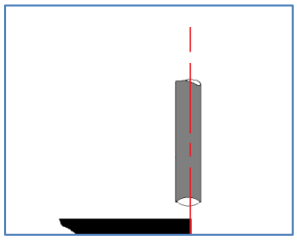


Figure 21

CNC Router operations begin at the top front left corner of the work piece and this should be set as the new offset position. Make sure that the central axis of the machine tool is in line with the edge of the work piece (Figure 21).

After 'jogging' the machine to the new offset position, the new offset values is set by clicking on the XYZ offset position icon (Figure 22).

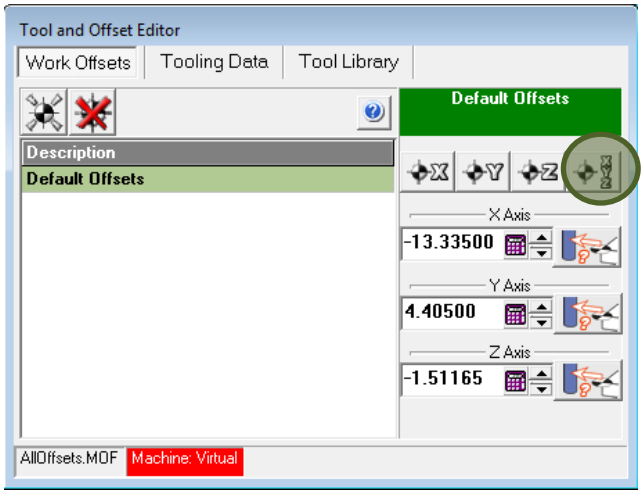


Figure 22

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 to be used click on the Tooling Data tab in the Tool and Offset Editor (Figure 23).

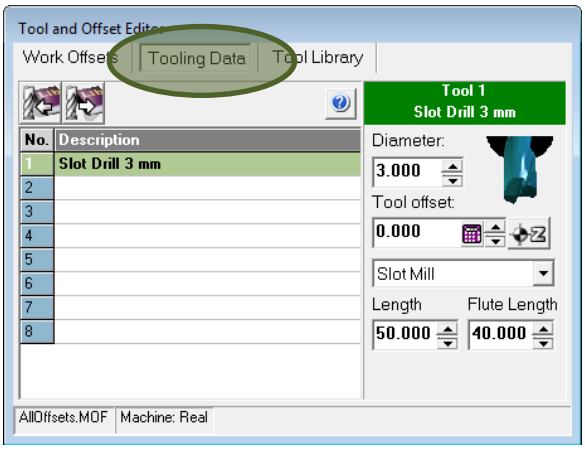


Figure 23

If the tool in slot No 1 one is not the one you require select the required tool from the tool library.

Running the file.

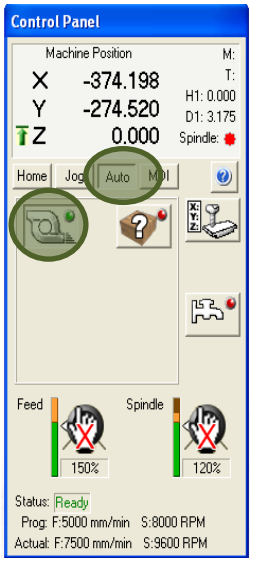


Figure 24

Click on the Auto tab and then on the Turbo icon (Figure 24).

Begining the Machining sequence (Figure 25).

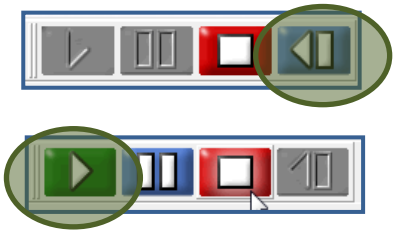


Figure 25

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

A warning box will appear to indicate that this is not a simulation but a real file execution (Figure 26).

Click "Yes" to begin the machining.

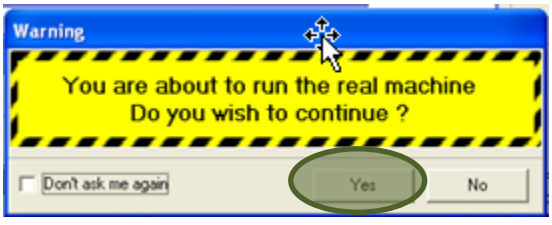


Figure 26

Machined Blank (Figure 27)

Figure 27