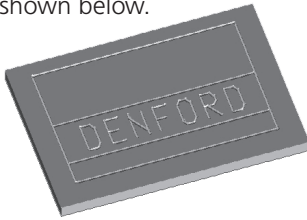


VR CNC Milling Step-by-Step Tutorial

Article #2003

This article takes you through each step to load a program, simulate a file, and manufacture the piece.

1. Open the VR CNC Milling software.
2. The part that will be machined in this tutorial is shown below.



3. The following items are needed for this tutorial:
 - 4" long x 2.75" wide x .25" high piece of acrylic
 - Engraving cutter
4. Load a piece of plexiglass on the machine and check that the engraving cutter is installed in the spindle.
5. Load the CNC file:
 - a.) Click File > Open
 - b.) Select the file Denford.fnc located in the **\program files\denford\vr milling\CNC Files** subdirectory.
 - c.) Click the Open button.
6. Set Tooling:
 - a.) Click the Tooling button.
 - b.) Right-click the number 1.
 - c.) Scroll to Insert Tool.
 - d.) Click 1/16" (0.0625") End Mill
 - e.) A red arrow should be pointing at the number 1. If not, right click the number 1 and select Change To This Tool.

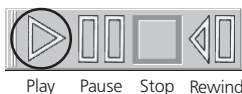


7. Simulate the CNC program:
 - a.) Click the 2D Simulation button.
 - b.) From the menu bar, click 2D Simulation. A checkmark SHOULD NOT be present before "Use X,Y Offsets". If needed, click the option to remove the checkmark.
 - c.) Click the 3D Simulation button.
 - d.) If the rewind button is active, click it. This will rewind the program. This is important to do before simulating or machining.



active inactive

- e.) Click the Play button to begin simulation.



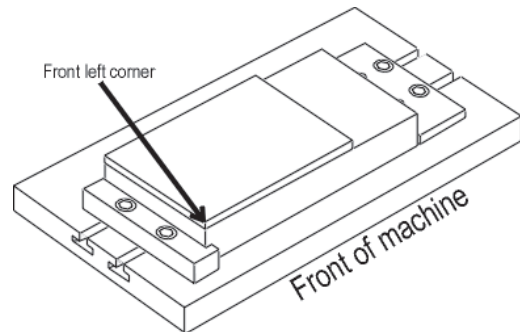
Play Pause Stop Rewind

- f.) Click the 2D Simulation and 3D Simulation buttons to close the windows.

8. Set the workpiece offsets:

Workpiece offsets must be set or loaded before part manufacture. The offsets tell the CNC mill where to begin machining on the stock.

 - a.) Connect to the CNC machine. To do this, click the Machine button. When connected, the Control Panel will appear.
 - b.) Home the machine by clicking the Home All button. Make sure each axis goes home (moves). If not, click each axis button.
 - c.) Click the Jog tab in the Control Panel.
 - d.) Move the cutting tool to touch the front left corner of the stock. The cutter should lightly touch the surface of the stock.



Use the following keys to move the axes:



X axis (left & right arrow keys)

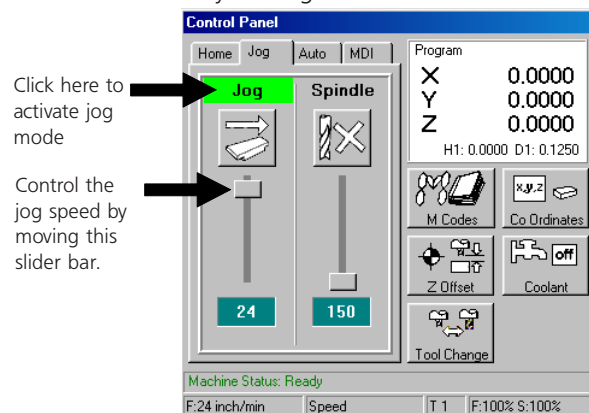


Y axis (up & down arrow keys)



Z axis (page up & down keys)

The word Jog must be shown with a colored background. If not, click the area to activate jog mode. In addition, the jog rate can be controlled by moving the slider bar.

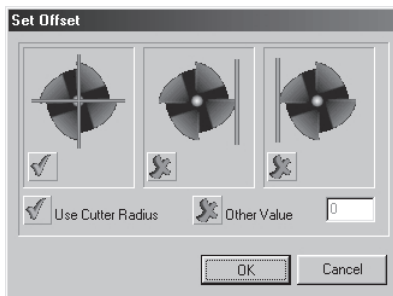


It is recommended to turn on the spindle while setting offsets (just in case the cutter happens to go too deep into the stock). If the machine is equipped with a programmable spindle, click the M codes button and select M03 Spindle Forward. This will turn on the spindle.

- e.) When the cutter is at the correct location, click the Offsets button.
- f.) Right-click Machine Offsets.
- g.) Left-click Add Offset.
- h.) An offset will be created named "New Offset". Left-click on "New Offset". Type a name and press Enter.
- i.) To make the offset active, right-click the offset name and select Make Current.
- j.) Click the datum button shown after the X axis box in the Work Piece Offsets window.



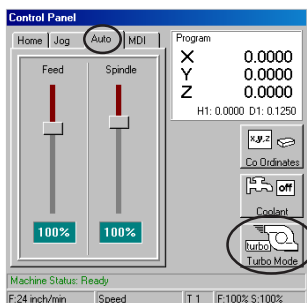
- k.) A Set Offset window will appear. Click OK.



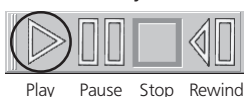
- l.) Click the datum button after Y in the Work Piece Offset window.
- m.) A Set Offset window will appear. Click OK.
- n.) Click the datum button after Z.
- o.) Click OK.
- p.) Click the Offsets button to close the window. The offset file defined may be used over and over again as long as the stock size does not change and the stock's position on the machine does not move.

9. Manufacture the part:

- a.) Make sure the guard is closed.
- b.) Rewind the CNC program (if needed).
- c.) If the machine being used is fitted with manual spindle control, turn on the spindle and set the speed to about 75%. If the machine has a computer-controlled spindle, proceed to step d.
- d.) Click the Auto tab.



- e.) Click the Turbo button.
- f.) Click the Play button.



- g.) You may be asked to change the tool, click OK.
- h.) Machining will begin. If an emergency stop is required, click the Stop button in the software or the emergency stop button on the machine.