

VR CNC Turning Step-by-Step Tutorial

Article #3003

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

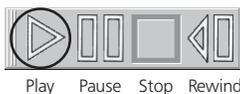
1. Open the VR CNC Turning software.
2. The part that will be machined in this tutorial is shown below.



3. The following stock is needed for this tutorial:
 - aluminum, 3/4" dia. x 1.5" long
4. Load a piece of stock into the chuck on the lathe making sure that at least 1 inch of exposed material is available for machining.
5. Load the CNC file:
 - a.) Click File > Open
 - b.) Select the file lathe.fnl located in the `\program files\denford\vrturning\CNC Files` subdirectory.
 - c.) Click the Open button.
6. Simulate the CNC program:
 - a.) Click the 2D Simulation button.
 - 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.



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



Simulation may show the cutter cutting the end of the piece off. Do not worry about this; cutting will be accurate.

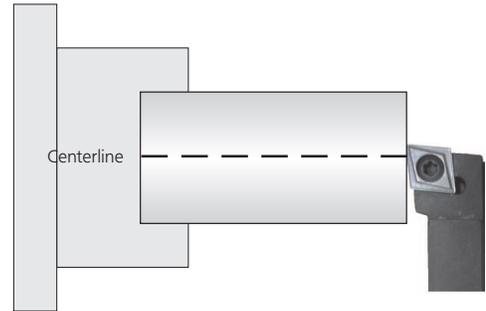
- f.) Click the 2D Simulation and 3D Simulation buttons to close the windows.
7. Set the workpiece offsets:

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

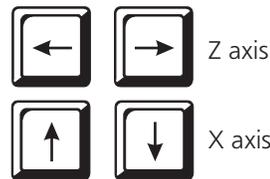
 - a.) Connect to the CNC machine. To do this, click the Machine button. When connected, the Control Panel will appear.
 - b.) Home the machine by clicking the Both XZ button. Make sure each axis goes home (moves). If not, click each axis button.
 - c.) Click the Jog tab in the Control Panel.



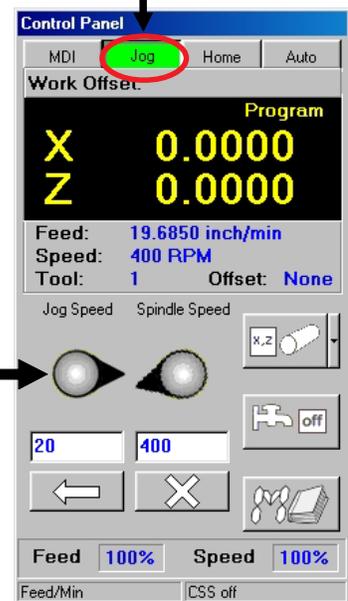
- d.) Jog the cutter so that it lightly touches the end of the part as shown below. This will locate the Z zero position. *The tool DOES NOT need to touch the centerline. Just make sure that the cutter is touching the end of the stock.*



Use the following keys to move the axes:



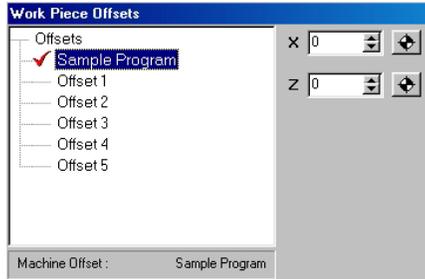
The word "Jog" must be shown with a green background. If not, click this area to activate jog mode.



The jog speed can be increased by moving the jog dial or by turning the override knob on the front of the machine (if fitted).

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.

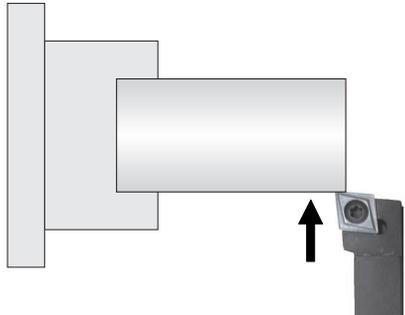
- e.) The Workpiece Offsets window will appear. Right-click Offset 1 then left click Add.
- f.) A new offset will be added. Left-click the new offset name three times. This will do two things; first it will add a checkmark to the offset. This activates the offset. Secondly, it will allow you to type a new name. Type a unique name and press [Enter].



- g.) Click the datum button next to the Z box.



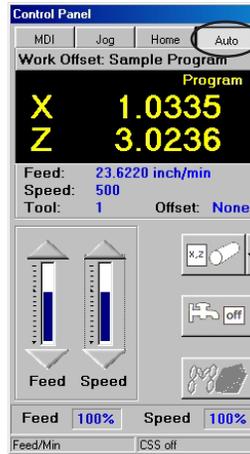
- h.) The Z Offset window will appear. Type a 0 (zero) for both the Distance and Clearance. Click OK when done. The Z part datum is
- i.) Move the cutter to the position shown below. The cutter should lightly touch the diameter of the stock. This will locate the X zero position.



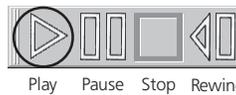
- j.) Click the datum button next to X in the Offset window.
- k.) The X Offset window will appear. First select Include Diameter. Change the Distance to origin to 0 (zero). Change the Diameter to read the diameter of the stock to be machined. Click OK when done.
- l.) The offsets are now complete. Click the Offsets button to close the window and home the machine.

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 Play button to begin machining. You may be asked a few questions before machining. If this is the case, proceed through the prompts.



If an emergency stop is required, click the Stop button in the software or the emergency stop button on the machine.