

Setting Workpiece Offsets in VR CNC Milling

Article #2000

Note: Screens shown in this document have been pulled from VR Milling 2.13. If using an older version of software, the Control Panel may look different.

1. Open the VR CNC Milling software.
2. Connect to the CNC machine. To do this, click the *Machine* button. When connected, a Machine Mode window will appear.
3. Home the machine by clicking the *Home All* button. Make sure each axis goes home (moves). If not, click each individual axis button.

4. Click the Jog tab located in the Control Panel (Figure 1).

5. Move the cutting tool to touch the front left corner of the stock as shown in Figure 2. The cutter should be lightly touching the surface of the stock. Use the following keys to move the axes:

X axis = right and left arrow

Y axis = up and down arrow

Z axis = Page Up and Page Down

You may adjust the jog rate by moving the Jog Slider bar, Figure 1.


6. When the cutter is at the correct location, click the *Offsets* button.

7. Right click *Machine Offset*.

8. Click *Add Offset*.

9. An offset will be created named "New Offset". You may rename this offset by left clicking on the name three times. Type a name and press Enter.

10. To make the offset active, right click the offset name and select *Make Current*.

11. Click the datum button  shown after the X axis box.

12. Select a cutter alignment method. (If using an engraving cutter, accept the default setting (center alignment)). Click OK.

13. Click the datum button shown after the Y axis box.

14. Again select a cutter alignment method and click OK.

15. Click the datum button shown after the Z axis box.

16. Only one alignment is available. Click OK.

17. Click the *Offsets* button to close the window.

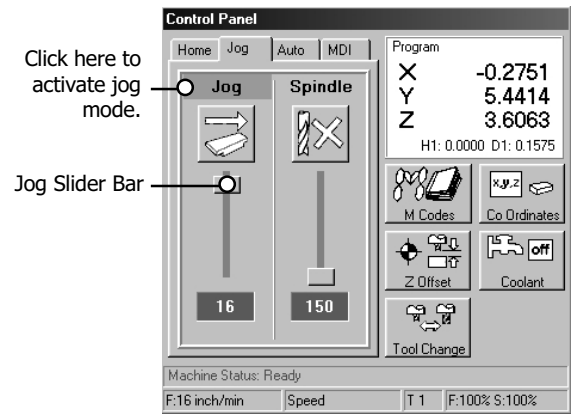


Figure 1

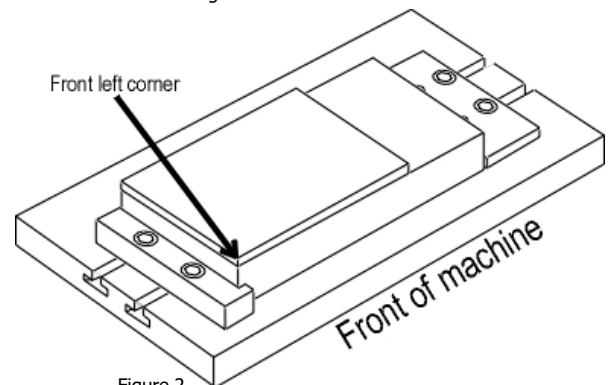


Figure 2



If the CNC machine you are using is fitted with a feedrate override knob, turn the knob a little past center. If the feed is too low, the axes will not move.