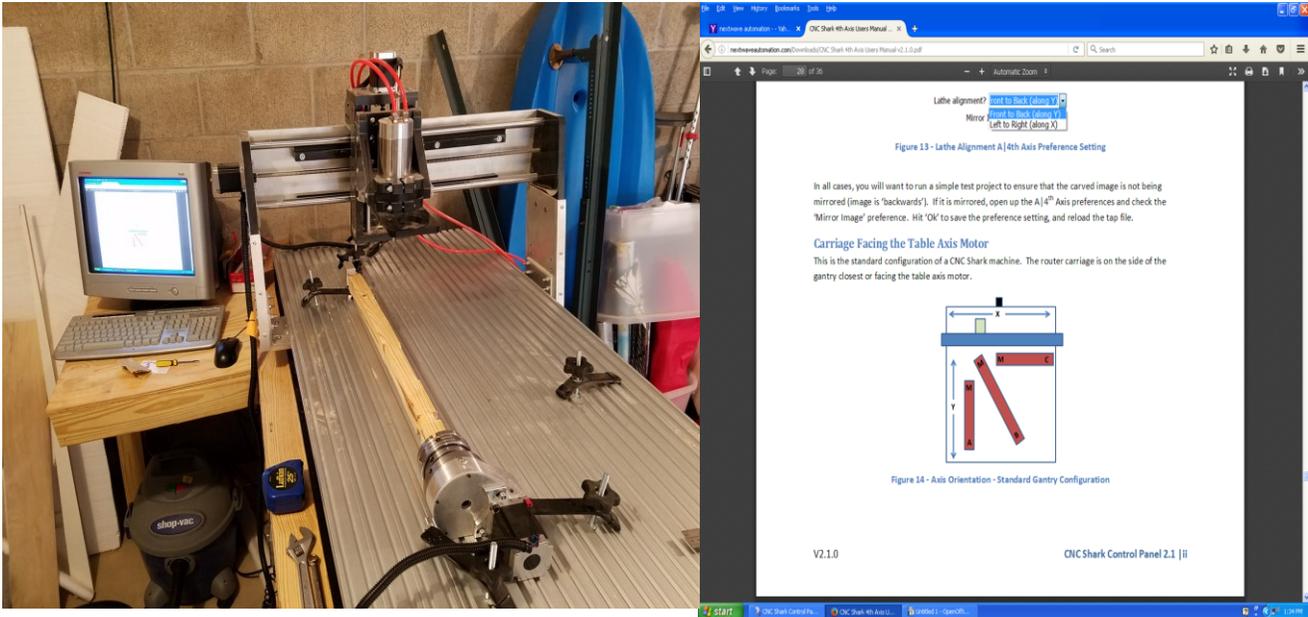


Hello,

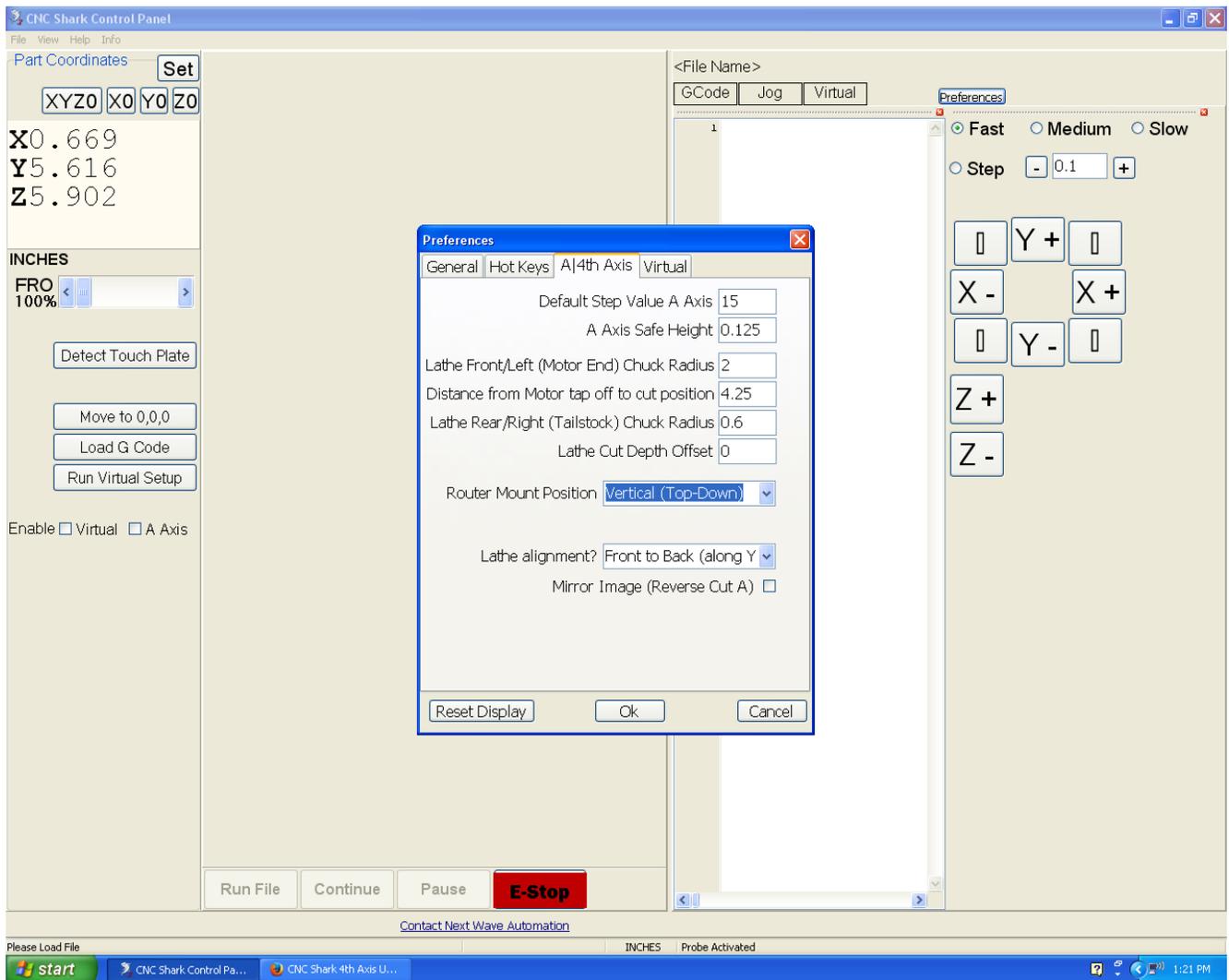
I am writing to request your assistance with an issue I am having with my 4th axis set-up on my CNC Shark. I seems that when I complete my set-up in Vcarve, I get a g-code that switches the length of material with its circumference. I've reviewed the operator's guide you provide and some on-line resources and I cannot get it to work properly. Please see the below process that I have completed for conducting a rounding tool path on my CNC Shark and let me know if I am doing something incorrect.

Bench Set Up



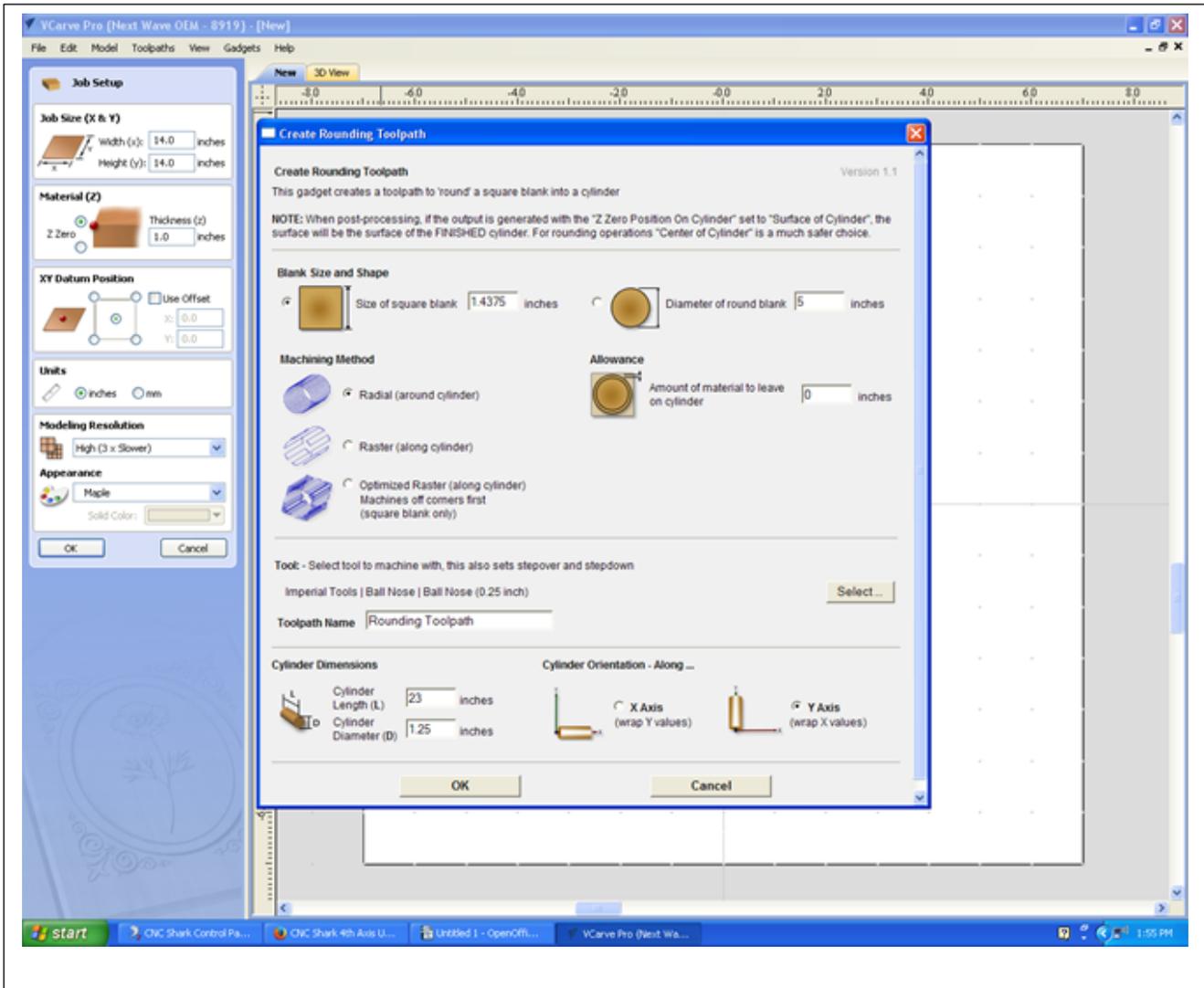
I have the bench set-up in a Y-wrap / Y-orient configuration.

Control Panel Preferences

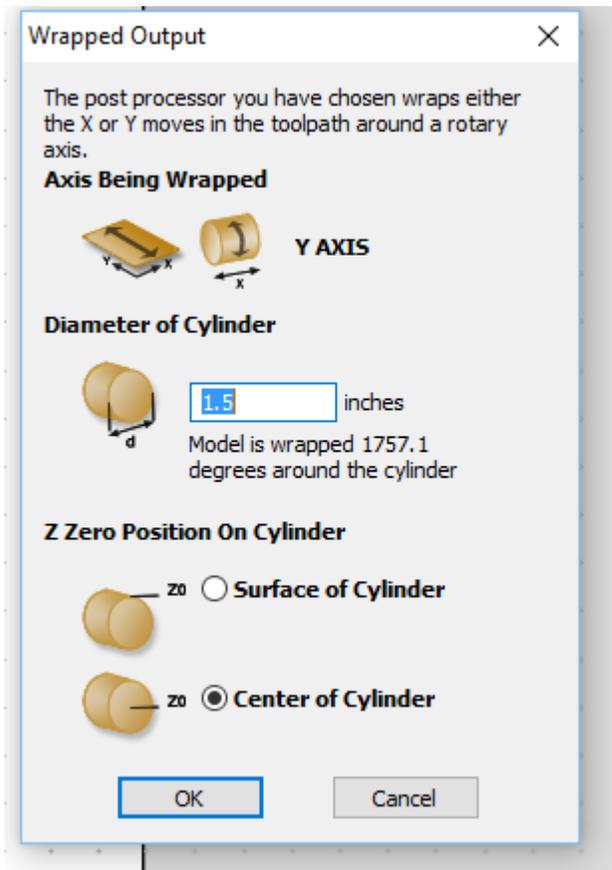


Preferences are identical to bench set-up.

V-Carve Set-Up



Toolpath set-up



Output

I have attached the Vcarve file and g-code for your reference.

Thank you for your help with this.