



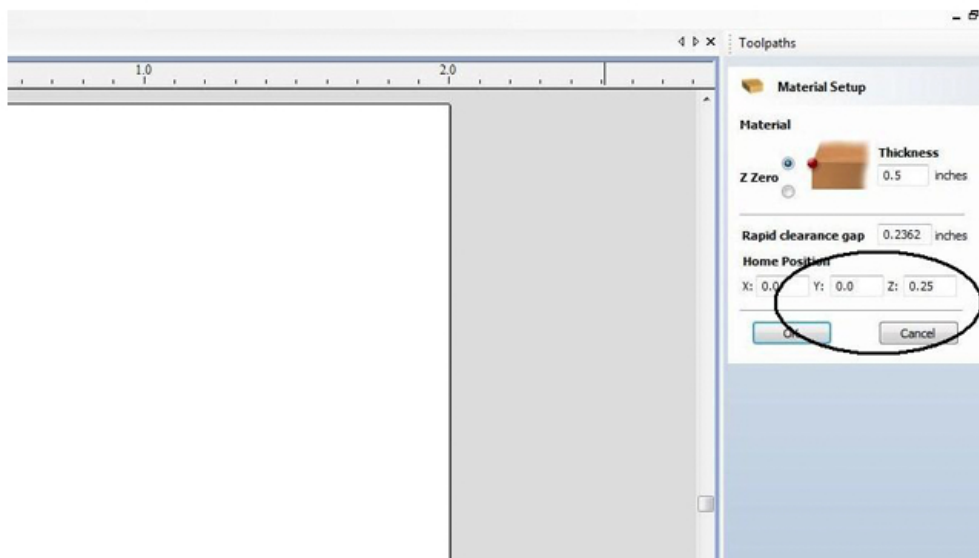
TECHNOLOGY FROM A
NAME YOU CAN TRUST.

TROUBLESHOOTING TIPS

Z AXIS CUTTING TOO DEEP

The following outlines the procedure to solve the problem of the Z axis cutting to deep.

1. Open V-Carve Pro.
2. Open an existing file previously created.
3. Switch to the “Toolpaths” tab.
4. Open the first icon in the “Toolpath Operations”.
5. Change the Home Position coordinates to X-0, Y-0, Z 0.25



6. When you select “OK” the software will warn you to re-calculate all toolpaths to reflect these changes. Be sure that ALL programs designed will need to have the Z home position changed and recalculated or these changes will not take effect.

If this does not correct the problem then log onto the General CNC website and on the technical support page fill in a service request ticket and detail the problem. We will contact you very soon to resolve the issue.



www.generalcnc.ca