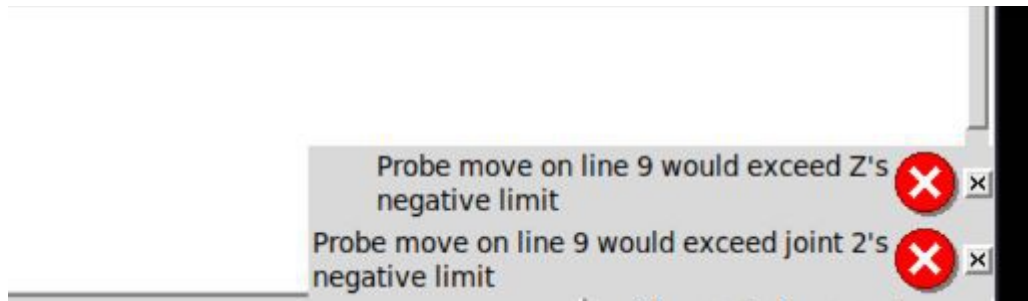
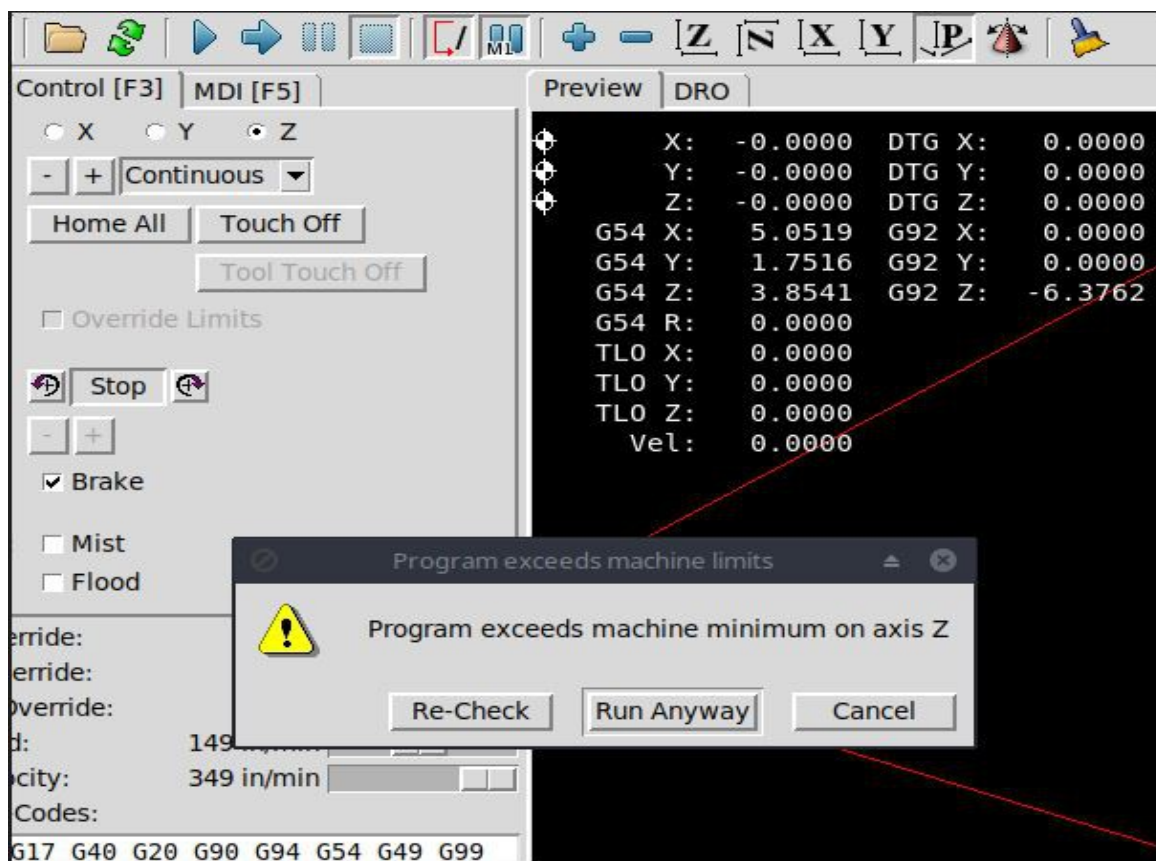


I have a routine I have used with other tables and software that has a small problem.

This is a G-code I use when starting a router operation to establish the bit exact height from the surface of the material. It moves the Z with the probe command down where the bit touches the surface moves back up and moves the Z down again slower to touch the surface. It then backs up and sets the Z height off the surface at 0.750", this allows me to know how high the bit is at the start of an operation. Now I get an error show in this image, if I don't touch off the Z axis before running the G-code. Not a real problem but I don't think I should get it.




If I do the touch off for the Z axis it gives me this message which I click on the "Run Anyway" and it will run the G-code exactly as I want. I think if I change a setting in the Z setup I should not get it, but not sure what I should change.



Here is the setting in the Z setup that I think may be where I need to make a change but not sure if that is correct.

Point and click configuration - Router.pncconf

Help Cancel  Z Axis Back Forward

Positive Travel Distance (Machine zero Origin to end of + travel):	0.0000
Negative Travel Distance (Machine zero Origin to end of - travel):	-4.7500
Final Home Position location (Offset from machine zero Origin):	-0.5000
Home Switch location (offset from machine zero Origin):	0.0000
Home Search Velocity:	inch / min 70
Home Search Direction:	Towards Positive Limit
Home latch Velocity:	inch / min 50
Home Latch Direction:	Same
Home Final Velocity:	inch / min 35
Use Encoder Index For Home:	No
Home Search Sequence:	1
<input type="checkbox"/> Use Compensation Filename:	zcompensation
<input type="checkbox"/> Use Backlash Compensation:	0.0000 - +

Type 1

This is the end result where the Z axis is positioning the bit at 0.750" above the surface. This procedure may be different then others use but has worked for me very well.

