V2 Troubleshooting Guide

Table of Contents

Revision History

First Release: October 2017 (JD)

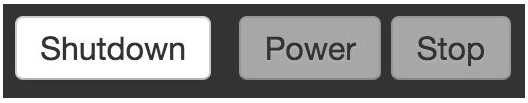
Introduction

Our goal at Pocket NC is to produce an excellent product at an affordable price so that people can turn their ideas into reality. We hope that using our product is a straightforward process and the the machine functions perfectly. But, we know that things don't always go perfectly. Hopefully this troubleshooting guide can help you resolve some of the problems that come up.

If your problem is not covered here, or if you need some more information, please contact us.

For instructions on how to run the machine, set-up parts for machining, measure tool offsets and other operation instruction see our tutorials.

Section 1: Common Problems

Problem	Possible Solutions
I can't connect to my Pocket NC or the user interface is not loading correctly on my computer.	1. Check that the Pocket NC is is plugged in and is powered on. The red LED on the E-stop button will light up when the machine is on. The user interface is ready to boot when the red LED starts blinking. It takes 2 and half minutes to fully boot the Pocket NC. 2. The computer must be connected to the mini-USB port on the mill not the standard USB port. 3. The mini USB may not be fully plugged into the machine. The pocket is deep so the plastic around the plug may have to be trimmed to fit. 4. The machine may not be fully booted. Wait 1 minute after turning on the machine before trying to connect. 5. The user interface may not load properly in every browser. Try using Chrome, Firefox, or Safari 6. Try rebooting the machine: power off the machine, unplug all the cords, wait about a minute for the capacitors to drain then plug it back in and restart. 7. You may need a different driver. Try downloading a new driver from the Beagleboard website. 8. Contact the Pocket NC help desk or post of the Pocket NC forum for help.
The Pocket NC is unresponsive, nothing happens when I tell the Pocket NC to move.	1. Check that the Pocket NC is plugged in and that the power switch is on. The Pocket NC's electronics can run off computer power, but the motors can't. 2. Make sure that the Pocket NC is not in E-stop mode. When the machine is in E-stop the red button on the machine will blink and the buttons in the upper right corner of the user interface will be gray/white. 

	Push the E-stop button to enable the motors.  3. Check that the computer is still connected to the Pocket NC with the USB cord. 4. Try rebooting the machine: power off the machine, unplug all the cords, wait about a minute for the capacitors to drain then plug it back in and restart. 5. Contact the Pocket NC help desk or post of the Pocket NC forum for help.
One of the axes on my Pocket NC is not working	1. Try rebooting the machine: power off the machine, unplug all the cords, wait about a minute for the capacitors to drain then plug it back in and restart. 2. Check the electrical connections for that axis. Look for damage and debris. 3. Contact the Pocket NC help desk or post of the Pocket NC forum for help.
The Pocket NC motors are noisy when the machine is running	Particularly at low feed rates, the stepper motors can produce quite a bit of noise. This is due to the electrical interference in the driver circuit. Increasing the feed rate usually decreases the sound.
The Pocket NC is not cutting in the location that I expected.	1. Make sure that the origin for your toolpaths is the B table offset/machine origin point of the Pocket NC. This is a point in space about 0.839 inches above the center of the B table. 2. Check if the coordinate system of your machining setup in CAM is set up so that the positive Z axis points at the spindle of the Pocket NC, the positive X axis points at the A table, and the positive Y axis points straight up from the B table. 3. Check if the toolpath origin is also the B table offset/machine origin point and that the toolpath coordinate system is appropriate. Remember that the Z axis is always the Pocket NC spindle. 4. Make sure that your stock is located in the same place

	on your machine as it was relative to the origin point in your CAM program.
I can't find/load my G-code file	1. Check if the selection set for the file type includes your file extension (file type) for example you can't find machining files with a .nc extension if the selection set is looking for only .ngc files 2. The file must be a text file that the Pocket NC is capable of reading. Generally these will have a extension like .ngc, .nc, or .cnc but any machinable file should work.

Section 2: Machining/Performance Issues

Problem	Explanation	Possible Solution/Things to Try
My tool chatters during the cut.	Vibration causes the cutter to not cut smoothly. This vibration is due to the natural frequency of the tool/spindle.	1. The feed and speed, and/or step-down and step-over need to be adjusted. See the feeds and speeds tutorial for more information. 2. Turn the spindle speed up. 3. Adjust the feedrate, usually moving slower will be smoother, but this is not always the case 4. Increase the step-down, taking a deeper cut into the material 5. Increase or decrease the step-over, the width of the cut. Typically the ideal step-over is about 60% of tool diameter.
Spindle sounds like it is slowing down during the cut.	The spindle power is insufficient for the amount of material being removed.	1. Decrease the feed rate in 5% steps until the spindle stops bogging down 2. Decrease the step-down and step over in your toolpaths. 3. Switch to a tool with more cutting flutes.
The Pocket NC is not cutting in the location that I expected.		1. Make sure that the origin for your toolpaths is the B table offset/machine origin point of the Pocket NC. This is a point in space about 0.839 inches above the center of B table.

	The stock may not be located in the same position relative to the origin point that it was in the CAM software used to create the toolpaths	2. Check if the coordinate system of your machining setup in CAM is set up so that the positive Z axis points at the spindle of the Pocket NC, the positive X axis points at the A table, and the positive Y axis points straight up from the B table. 3. Check if the toolpath origin is also the B table offset/machine origin point and that the toolpath coordinate system is appropriate. Remember that the Z axis is always the Pocket NC spindle. 4. Make sure that your stock is located in the same place on your machine as it was relative to the origin point in your CAM program.
The drive/lead screws squeak.	The end of the lead screw is touching the frame of the machine when it turns.	1. Make sure that the screw and the hole are free of debris. 2. If the noise it minor is is best to let it wear into position. The noise should decrease with time - Use a very small amount of general purpose synthetic grease to lubricate the tip of the screw. - Do not grease the whole screw. - Do not put grease where it will contact the anti-backlash nuts. - Grease will collect debris and may cause increased wear on your machine.

Section 3: Error Codes

Error Code/Message	Explanation	Possible Solutions
Near Line XX Linear Move Exceeds Joint X's Positive/Negative Limit	This error means that the toolpath/G code is telling the machine to go past one its travel limits Joint 0 = X axis Joint 1 = Y axis Joint 2 = Z axis Joint 3 = A axis Joint 4 = B axis This error is often caused by a tool length offset being set incorrectly or by a part that is outside the Pocket NC's work envelope. The move that is causing the error is located on or near the line of the program indicated	1. If the error is on joint 2 (Z axis), check if your tool length offset is set correctly. See the "Tool Length Offset" tutorial for more information. 2. If the error is for exceeding the negative limit of joint 2 and the tool length offset is correct, increase the tool stickout or switch to a longer tool holder. 3. If the error is for exceeding the positive limit of joint 2 and the tool length offset is correct, decrease the tool stickout or switch to a shorter tool holder. 4. Check if your toolpath is asking the machine to exceed its limits of travel. Open your code in a text editing program and use the search bar (CTRL + F) to search for any values that are greater than the limits of travel for the Pocket NC mill. a. The X limits are -2.00 to 2.55 inches b. The Y limits are -2.4 to 2.6 inches c. The Z limits are

		-3.45 to 0.10 inches* d. The A limits are -25 to 135 degrees e. The B limits are -9999 to 9999 degrees *you have to add the tool length offset to the Z axis value 5. Adjust the location of the part/stock so that it is within the Pocket NC's work envelope 6. Decrease the retract height on the toolpaths if the part is close to the limits of the Pocket NC's travel. 7. Oversized parts may be able to be fixtured and cut one side at a time instead of all in one program.
14 Unexpected Real Time Delay on RT Thread 1	This error means that there has been a delay in the Beaglebones processor.	1. This error can be caused by a temporarily dropped connection between the computer and the Pocket NC mill or by a program that is so large and complicated that the Beaglebone's processor cannot keep up. 2. Check that the connection between the computer and the machine is good. 3. Try slowing down the feed rate or reducing the file size of the G code file

		by breaking it into more than 1 operation or by loosening the tolerances in the toolpath and reposting it.

Section 4: Resetting/Updating the Machine

This section will have instructions on how to update the machine when that capability is available.

This update process will require an internet connection through the ethernet port on the Pocket NC.