Genmitsu CNC Router Trouble Shooting

UPDATED 1 YEAR AGO BY LYNN

Problem	Possible Solution
Get the serial port error When trying to connect, I get the “serial port error 1: no such file or directory”.	This is Windows saying it cannot connect to the router. Make sure that you have installed the driver and selected the correct COM port.
The machine lost connection I was able to assemble the machine and everything was working fine, but then in the middle of the sample carvings the machine lost connection. I have since tried it with two other computers and the offline controller which had worked fine when testing it earlier. Now no matter which usb port I try it will not connect. The machine doesn’t show up on the com ports at all in either computer I’ve tried. They both connected before as well. I’ve tried restarting, reinstalling the grbl software, and tried the reset button on the control board.	Please check whether the USB port of the motherboard is loose or not. Then use the offline control module to control the machine. If the offline control module failed to control the machine, we suggest that you **upgrade the firmware** for the motherboard.

Problem	Possible Solution
Control board doesn't work Have one can't get to work don't know if the USB cables connected correctly.	If the USB cable is connected correctly, the 5V indicator next to the RESET key will light up. The offline controller and the USB cable cannot be connected at the same time.
The machine catch and jump when working I am having an issue with the X-axis linear bearings, they are binding extremely bad causing the machine to catch and jump at certain times. When I test slide them with the shafts outside of the machine there is a ton of friction on them. The Y-axis bearings work perfectly for reference.	Check the alignment of the guide rods and the threaded screw moving the axis, check the coupler between the stepper motor and the X-axis threaded screw is correctly aligned and slacken the screws holding the guide rods. Move the spindle mount all the way to the left by turning the threaded screw by hand and tighten the left hand screws, move the spindle mount to the right in the same way and tighten the right hand screws. It should then move freely across the axis.
The spindle continues to rotate when the machine stops When the spindle speed is too high and when you are already recording the machine stops. The spindle continues to rotate but it stops moving the axes: x, y, and z. I already tried to slow down the spindle to S200 and the F100 values. It always stops. It seems like the spindle have some issues.	Does the spindle motor turn on or turn off properly when you click the button "spindle on/ off"? Can you use the slider to control the spindle speed? If the spindle speed can be controlled normally, we can confirm that the spindle motor doesn't have any problems. We suggest that you use our sample gcode to test it again.

Problem	Possible Solution
One of the axis doesn’t move The X-axis and Z-axis are Ok but the Y-axis does not move.	Check the connections; make sure it is plugged in correctly. Check that the axis is not binding by turning the screw by hand (disconnect the router first). Check that the grub screws on the coupler connecting the threaded rod to the stepper motor are tight and the coupler is not slipping. If this doesn’t work Please swap the X axis motor and Y axis motor cables. If the problem moves to the X axis, the X axis driver chip has the problem. The control board should be replaced.
The offline control module cannot move the axis. The 3018 PRO works fine, but I am having issues with the offline controller. The controller powers up and I am able to scroll through all of the menus, but I am unable to get it to move the cnc axis's. The controller says it loads a file, but then nothing. I have read all the instructions and material I can find on the controller.	The offline controller cannot be connected at the same time as the USB cable. Please make sure that you have unplugged the USB cable when using the offline control module and vice versa.

Problem	Possible Solution
The laser doesn't function at all The laser does not function at all, after following all available instructions. Both cooling fans in the kit operate fine. No signal from daughter board for laser. Woodpecker 3.2 control board does send on/off signal via 3-pin laser plug to daughter board.	Please make sure that the wiring is correct at first. You cannot connect the spindle motor and the laser module to the control board at the same time. Also check the switch position on the Laser Control Module for a 5.5W Laser
The laser intensity is not strong enough to burn anything I have the 5.5W laser . I am trying to control it using lasergrbl. I can see the laser flash on and off during printing, however, it is not strong enough to burn anything, even cardboard. If I push the "on" button on the control board, I get full power from the laser, but it is too strong to be useful.	We suggest that you adjust the focus manually by turning the knob at the bottom of the Laser Head until the Laser forms a tight dot. Then adjust the output power through the software.
The spindle just pulses a lot of the time (on-off on-off on-off)	This issue is caused by a faulty power supply.(The Power supply needs to be replaced) The current used by the spindle starting is high and it triggers the overload protection of the power supply. We have already changed the power supply we supply.

Problem	Possible Solution
The Laser is engraving extra lines When I use the Laser it leaves extra straight lines across my engraving	When using a Laser the Grbl motherboard needs to be set into Laser mode to prevent this. Send a $32=1 command from the console pane. When you replace the spindle motor send a $32=0 command.
When I run some Gcode it causes the bit to immediately plunge into the work	This is often caused by the router being told to go to a ‘safe’ position, unfortunately this is often an ‘unsafe’ position due to the limitations of these small routers. This is set by a G28 command so preferably tell your CAM software not to use G28 commands or if your Gcode has G28 commands at the top then delete the
My spindle motor does not have a red dot or my spindle motor is rotating anti clockwise	When connecting the spindle motor i cannot find the red dot to show which way round the wires go. If your spindle motor is turning in the wrong direction swap over the 2 wires on the top of the motor. Newer motors will often have a faint + mark on the top of one of the white ‘dots’ on the motor, connect the red wire to the connector closest to the red dot or + mark.

Problem	Possible Solution
When I run some Gcode the size of the engraving is wrong	Check the units you are using on the router and in the design and manufacture software are the same, a mismatch between Inches and mm can cause this. Another possibility is that your router settings of $100 to $102 are incorrect or have been modified. These should all be the same value and, depending on the router motherboard you have, should all be 800 or 1600.
When I connect to the router it shows an ALARM state	This is perfectly normal for a router with limit switches fitted and the Homing cycle enabled. Just run a homing cycle and the machine will position the spindle to the home position and clear the Alarm state.
I want to add a Z-Probe to my 3018-PRO where can I connect it.	The Z-Probe connects to the pin marked A5, one side to the + and the other to the -

How did we do?



Related Articles

Cubiko CNC Router Resource Page
Genmitsu PROVerXL 4030 Resources
3030-PROVer MAX Resource Page