MillRight

MILLRIGHT CNC QUICKSTART GUIDE FOR RUNNING ASSEMBLED MACHINES

Version 1.03

Important safety rules for operating your MillRight CNC machine:

Never place your hands near a spinning end mill or bit.

Unplug the router before changing cutting tools.

Always wear eye and hearing protection while operating your machine.

Always run dust collection or wear a mask while performing a milling operation.

Do not leave the machine unattended while running an operation.

Do not operate your machine while under the influence of alcohol or drugs.

Do not place your hands between the bed and frame while the machine is moving.

Visually inspect wires prior to power up to prevent short circuits.

Ensure work pieces are properly secured before running a milling operation.

Do not allow minors to operate the machine without adult supervision. Do not wear loose fitting clothes, jewelry, loose long hair while operation the machine.

NEVER use the laser attachment on plastics without extensively researching suitability first. Some plastics can catch fire or emit very poisonous gas when lased.

Getting Started with Moving the Machine:

This guide is intended to help you start running your assembled machine. CNC machining is a learning process that will take some time to master. The aim of this document is to get you basically familiar with the operation of the machine and to point you to other resources that will expand your knowledge. Machine technical support is available via our support email, support@millrightcnc.com, however machine application support is only provided via our forum at www.millrightcnc.proboards.com. MillRight CNC is here to help you. There are also fellow customers willing to help on the forum too.

You will connect your computer to the machine using the provided USB cable. On most computers the drivers for the control board will automatically load onto your computer in the background. Most likely nothing will pop up on your screen when you plug the machine in. If your computer doesn't connect to the machine, you might need to install the Arduino IDE linked here to get the drivers that you need.

You can now establish a connection using Universal G Code Sender (UGS), which you can download at our resources page at www.millrightcnc.com/resources. Use the drop down menu for the Port and select the port that you see available. Your COM # will likely be a different number from the example below. Your computer should select the correct COM port, but if nothing happens, refresh the port (two circled arrows) and select the highest numbered COM from the drop down window. You must have Java to run UGS. If you don't have Java, just do a Google search for "Java download."
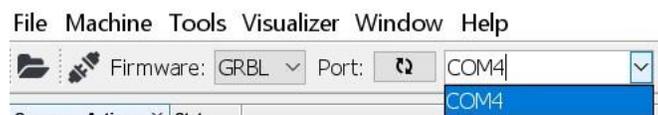
The board is preloaded with Grbl, which is the firmware that will control the machine. Do not attempt to "reflash" the board using X-Loader or the Arduino IDE. It is ready to go, just as it was shipped to you. In Universal G Code Sender, Grbl should be set as the firmware. Once you click to establish a connection (the plug icon in the picture above) the machine will tell you that it has been connected as follows:


Powering on the M3 or Carve King: Plug the 24 volt power adapter into the female barrel plug receptacle for the machine. If you have the laser attachment, you must be VERY careful not to

plug the machine's 24V power supply into the laser's female barrel plug receptacle. Doing so will may destroy the laser.

Powering on the Mega V or Power Route: Plug the black control box into a wall outlet. Turn the red E-STOP button on the front of the box clockwise to turn on the power. Push it in directly to cut the power. Note that this will cut the power to the motors but will not cut off your router. *Note that nuisance tripping may occur on some GFCI outlets.*

If you have homing switches you could type $H in the command line of the Console window then press enter, or click the "Home Machine" button under the Common Actions tab. This will initiate a homing cycle.
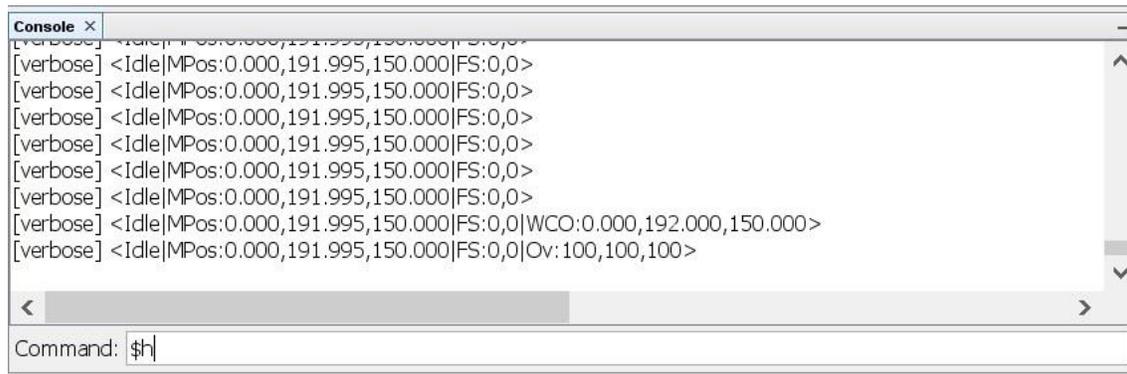
On a homing cycle, the Z axis will lift first. It will raise Z, hit the switch, back off, move very slowly into the switch again, and then pull off the switch a few millimeters. Afterwards, the X and Y motors will begin simultaneously moving the machine towards the switches (*switches are in the positive direction, except for the X axis on the Mega V)* until those switches are engaged. It will do the same pull off, switch re-engage, then slightly pull off on each one of those axes. After a homing cycle, the machine is ready for additional motion commands. The advantage of homing is that the machine will obtain a known position which will allow you to utilize work position systems, which will be discussed later. Homing is not entirely necessary, but it is a good practice.

Now you can jog the machine. Jogging means the machine is just being moved around to get it into position, such as to get it into place for a cutting operation. Use the Jog Controller to click

the axis control buttons to move the machine around. The X- button will move the router to the left; X+ will move the router to the right. Y+ will bring the table towards the front of the machine with the M3 or send the gantry to the back with all other machines; Y- will move the table towards the back with the M3 or send the gantry to the front for all other machines. Z+ will move the router up; Z- will move the router down. Be mindful of the step size. The step size figure is in millimeters. It will move when you press an axis button. It's possible to move the machine too far and crash it, so be careful. If you just homed your M3, Carve King, or Power Route, you'll need to first move X-, Y-, and Z- because each axis homed towards the positive side of travel. This is the conventional home direction for CNC routers. The Mega V, however, homes to the negative side in X so you will have to move the machine X+, Y-,Z- after homing.
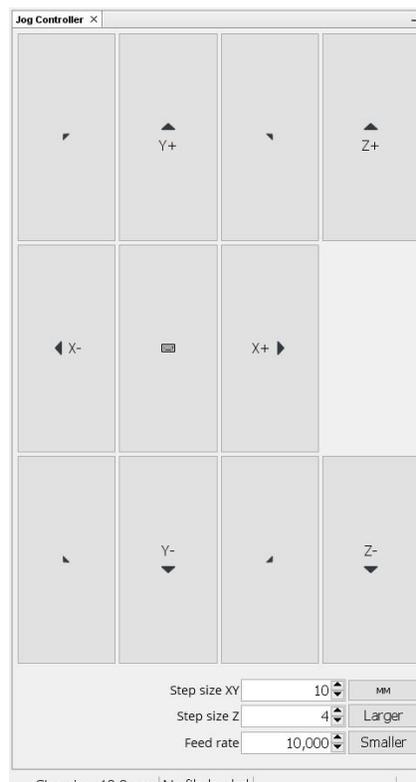
If the machine does not move when you give job commands, make sure the step size and feed rate are large enough. For instance, if the feed rate were set to 10mm a minute you likely would not be able to see the machine move.

Basic G Code Commands:

In addition to using the axis control buttons, you can type in G code commands into the command line. Let's cover some basic G codes now.

G0   Rapid movement at the default feed rate (feed rate is how fast it moves)

G0 movements are just for getting around. You don't cut anything with a G0.

G1   Linear movement at the specific cutting feed rate.

G2   Clockwise arc movement at the specific cutting feed rate.

G3   Counter-clockwise arc movement at the specific cutting feed rate.
      G2 and G3 movements are accompanied by X, Y, or Z, and I, J, or K "words" to tell the machine
      how exactly to arc. *You do not need to understand any of that right now because the CAM
      software (the program that makes your g code for cutting stuff out) will do all of that for you.*

G20   Sets units to inches

G21   Sets units to millimeters
      Since these commands are "modal", it remembers that you set it (while the controller is on!)
      and won't be changed until you issue the opposite command. If your CAM software generated G
      code in millimeters, you need to set a G21, and vice versa.

G90   Specifies absolute positioning

G91   Specifies incremental positioning
      G90 and G91 are modal commands as well. This is best understood by way of example. Let's say
      you were at X50 and then issued the command G0 X60. Where would it move? It depends. If you
      had a G90 set for absolute positioning, it would move 10 to the right. If you had a G91 set for
      incremental positioning, it would move 60 to the right. See the difference? Now let's say you
      were at X50 and issued the command G0 X0. Where would it move? If you had G90 set, it would
      move 50 to the left to take the machine to the position of X0. If you had a G91 set, it wouldn't
      move it all. G91 is incremental, so 0 from where you are at is… well, where you are at. Be
      advised that **using the axis control buttons under the Machine Control tab in UGS will set a
      G91**. It does this every time you click one of the axis control buttons.

G92   Used to create a temporary work coordinate system.
      This is most often used to zero out each axis so you can declare the origin of your work. For
      instance, you might jog the machine to where you want to start the work, setting the cutting
      tool to the point you want to start the cutting file. For instance, G92 X0 Y0 Z0 will set the "work
      coordinate system" to zero for each axis. Many cutting files use X0 Y0 Z0 as the origin, so this is
      the position you will most often want to set before starting a file.

F   Specifies the feed rate for G01, G02, and G03 moves.
      The F command must be used with a number. F 1000 or F200.0 for example.

S   Specifies the spindle speed or laser intensity.

The S command must be used with a number. S6000 or S 120000.0 for example. On machines shipped assembled with a laser installed, S12000 is the maximum value, with S3500 being about the minimum value that will come on after an M3 command.

M3 Turns on the spindle or laser. (Only if a spindle is configured for control by enable pin) An S command alone won't turn on the laser or spindle. An M3 will allow it to actually turn on.

M5   Turns off the spindle or laser. (Only if a spindle is configured for control by enable pin)

Running Your First Job:

You might be interested in running a cutting file now. You can run a very basic engraving file using this g code file that you can download and save a copy here. This will require a piece of wood about 8 inches by 8 inches (200mm by 200mm). A larger piece is okay. Put the cutting tool in the router. This file was designed with a ¼" core box router bit in mind, but an end mill, ball nose, or V bit in a size ¼" or under will work as well. This program is not set up for speed or efficiency but to check that the machine is working correctly. Your personal programs can be ran at more optimal speeds and more efficient tool paths for a more enjoyable experience.

Place the board on the table and clamp it down. Keep the clamps towards the edges so the cutting tool won't collide with them. Set a G92 X0 Y0 Z0 with your cutting tool towards the lower left side of the board, with the tool just at the top surface of the board. An easy way to set this is to jog the machine to the desired X and Y position, then start lowering the cutting tool towards the board. Be mindful of your step size here. Once the tool gets close to the board, change the step size to 0.1mm and keep dropping it. Place a sheet of notebook paper underneath the cutting tool and on top of the board. Slide the paper back and forth to see if it catches on the cutting tool. Keep lowering the Z axis until it just barely drags on the paper. At this point, issue the G92 X0 Y0 Z0 command. Now, because it's best to turn the router on when it's not engaged with the stock, lift the Z axis by issuing a G90 G0 Z5 command. Assuming you had a G21 (millimeters) set, this will raise the Z axis 5 mm. The program will move the machine about 6.5 inches (165mm) in the Y+ directions and 7.5 inches (190mm) in the X+ direction. You won't collide with your clamps, will you?

Now load the G Code file you just saved by clicking on File at the top of Universal G Code Sender. Go to open and select the file where you saved it onto your computer. Turn on the router, and click Send, which is the Play symbol at the top in Universal G Code Sender. Stay clear and do not reach into the cutting area under any circumstances. You will end up with something like this, depending on the type of cutting tool you chose:

Designing your own parts:

You'll need to use some other software to design your own parts and make g code for them. We do not officially support software, but we do have basic tutorials for Autodesk Fusion 360 and Inkscape on the MillRight CNC Resources Page. These software packages are free. You'll need to get a hobby or free small business license (making less than 100K annually) from Autodesk for Fusion 360, as appropriate. Do a google search to download these programs or use this link for Fusion 360.

Fusion 360 is a great program that we use to run our entire industrial machine shop. However, some users find it to be more technical and with more features than they need. If you want different software, please check out programs such as Vectric Cut2D or Vectric Vcarve. Vetric programs are very simple to use and boast a smoother, more intuitive and creative interface. We are an authorized dealer for the entire line of Vectric software products, so please get in touch with us if you'd like to purchase one of those programs or go to https://millrightcnc.com/buy-now/ and then click "Software".

Those with lasers may wish to review the laser setup instructions here. That document describes hardware set up as well as the particulars of software settings for laser jobs.

Troubleshooting:

Problem: Universal G Code Sender gives an error when trying to open a connection, or the machine has stopped responding.

Solution: Close UGS, remove USB cable from USB port, wait five seconds, plug back in, open UGS, and attempt to open a connection.

Problem: The motors make a noise that sounds like radio static.
Solution: This is normal when the machine is first booted up and no motion commands have been issued.

Problem: The machine isn't moving to the commanded position.
Solution: Most often this happens when incremental positioning is set (G91) but the user thinks the machine is in absolute positioning (G90). Set the appropriate mode.

Problem: The machine appears to be missing steps or not moving the full distance commanded on one axis in particular.
Solution: The set screws should be checked on the motor pulley and tightened if loose. If this is not an issue, the belt may need to be tightened

Problem: The motor stalls or the machine jumps off the planned path when cutting.
Solution: Slow down the feed rate and/or shallow up the cut.

Problem: The machine stops during homing before it hits the homing switch.
Solution: Separate motor wires from switch wires to reduce electromagnetic noise creating a false trigger.

Work Coordinate Systems:

> *Warning, this part is a bit advanced. Most people won't understand this until they have some experience. You can keep running things using the G92 method described above until you get ready for this. Here is a work Coordinate video for further reference.*

Programmed work coordinate systems can **only be used if you have homing switches**. This allows you to move the machined to predetermined starting positions even after powering off the machine. This is especially useful if you have a clamping fixture and wish to run the same operation over and over, or if you wish to machine a part with two or more different tools.

Let's cover the basics here. Work coordinate systems are designated with G10 commands and accessed using the G54 through G59 commands. Let's cover the basics of the commands:

G10 L20 P1 X0 Y0 Z0   This command would designated the current position of the machine as the origin (0,0,0) point of the G54 coordinate system.

G10 L20 P2 X0 Y0 Z0   This command would designated the current position of the machine as the origin (0,0,0) point of the G55 coordinate system.

G10 L20 P3 X0 Y0 Z0   This command would designated the current position of the machine as the origin (0,0,0) point of the G56 coordinate system.

See the pattern? P4 would be for G57, P5 for G58, and P6 for G59. There is another way to set work positions, but this is perhaps the most easily understood.

G54   Modal command that tells the machine that it should interpret all further absolute positioning commands (those commands issued when a G90 is set) as places to go in the G54 system.

G55   Modal command that tells the machine that it should interpret all further absolute positioning commands (those commands issued when a G90 is set) as places to go in the G55 system.

G56, G57, G58, and G59 are set the same way. You can tell where the origins of these work coordinate systems are relative to the origins of the machine coordinate system by typing $# into the command line.

G53   Specifies the machine coordinate system for the motion command that follows on the same line. This is a "non-modal" command, so any further commands issued without a G53 on the same line will be interpreted within the context of the work coordinate system.

Let's give a real world example. Imagine that you wish to do a 3D carving that will first be roughed (material removed in bulk close to the finished contour) with an end mill and then finished with a ball nose mill. After you lay your board on the machine table and clamp it, you might have a particular point on the board that you wish to begin your work (possibly at the bottom left corner). You could jog the machine to that location then establish that point as the G56 work coordinate origin by typing G10 L20 P3 X0 Y0 Z0. You would type G56 and then run the roughing operation and change the cutting tool to a ball nose end mill. You would then home the machine again and type in G56 X0 Y0. Notice that you didn't issue a Z0. The ball nose tool you installed will almost certainly be sticking out more or less than the end mill you used, so you will **not** want to command the machine to Z0 before resetting it. Jog the Z down to the stock top. You can either issue a G92 Z0 or reset the Z origin for this coordinate system using G10 L20 P3 Z0. Keep in mind, we are using "P3" because in this example we are setting the G56 coordination system.