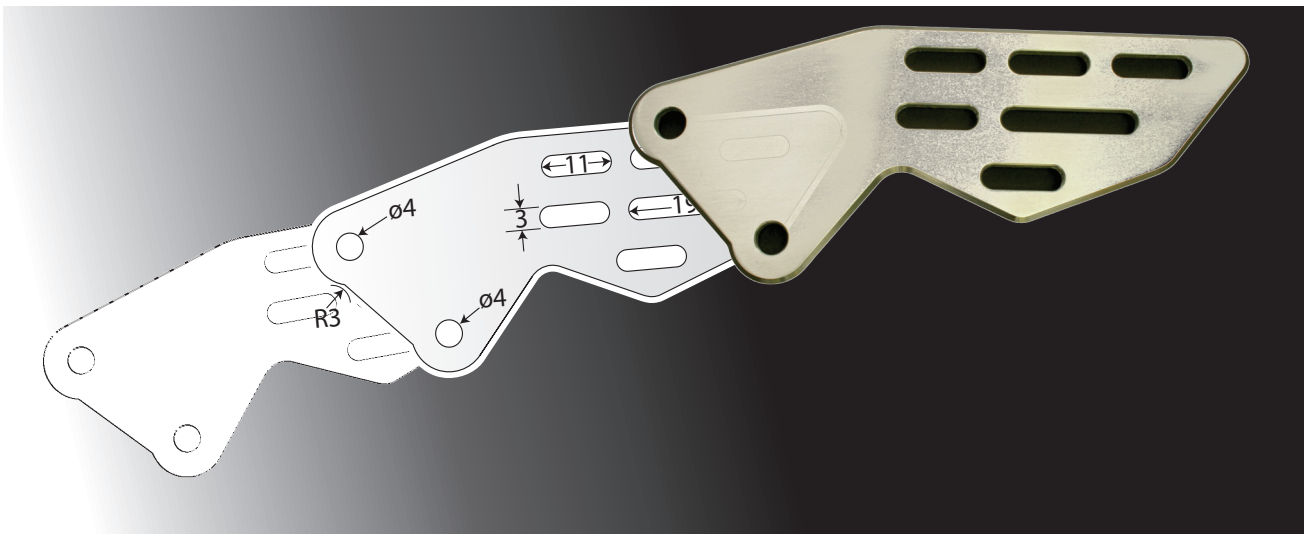
CNC in the Workshop



Part 3

Excludes the final section on "Backlash", which has been moved to Part 4.

Part 3

In this part of the series, we consider some of the important concepts an steps required to get your computer up and running Mach3, and the links between computer and mill.

THE COMPUTER

I said this would not be a computer-orientated series, and that's true; but we do need to discuss some related matters here. It would be a bit strange if you could run a CNC system without a computer, and there are some factors which fundamentally affect how we connect the computer to a CNC machine, so let's discuss those now.

There are some choices to be made concerning computer systems, governed by the requirements of Mach3, and the issues of the parallel port and the use of Windows XP or Windows 7, as mentioned in Part 1. The computing requirements for running Mach3 are low, and a powerful, fast and expensive computer is not required. In some respects, you will not want the latest all-singing all-dancing computer system because of issues with latency and jitter (see Part 1). The latest generation of computers feature all manner of ECO-friendly power-saving features which tend to wreak havoc with latency and jitter, so don't feel you are missing anything if you run Mach3 with an old computer. The minimum specification is at www.machsupport.com, a site with which you should become familiar.

First, the computer must have a minimum processor speed of 1GHz, and 512Gb memory, so that covers most computers produced over the last 10 years or so. Personally, I believe more memory is better, but it depends on whether you intend to use the computer to run software other than Mach3. To run Cut2D or VCarve Pro, as suggested later in this series, you will need at least 1Gb memory if the computer runs Windows XP or 2Gb if it runs Windows 7. Go to www.vectric.com then choose Support > System Requirements. The > symbol means "and then" usually as a sub-menu or as the next choice, and it's a convention we will use throughout this series. In this case, the Support menu produces a sub-menu so that System Requirements can be chosen from there.

Photo 20: The large, pink, 20 way parallel port (printer) connector.

Second, running Mach3 requires a parallel "printer" port, or some way of coping with the fact that most modern computers do not have a parallel port, as it is being removed from computers very quickly because of the move to USB. Mach3 expects to send its control signals out of the parallel port, so we need a solution for modern computers. Several companies now make add-on parallel ports which usually take the form of additional "cards" of electronics which fit inside the computer and add a parallel port. We need to take care, here, as cheap solutions have proven to be frequently ineffective because they often make the port work for a printer, but have not fully implemented all the aspects of a parallel port and won't work fully with Mach3. There are at least two exceptions which work well, and a preferred option might be the PMDX-2PARPCI which will work on a computer which has a PCI connector on the main motherboard. There are lots of other opinions on this, on Mach-Support, in the thread "Recommendations on Best Breakout Board" by General Mach3 Users".

Alternatively we could abandon the parallel port altogether, just like the computer manufacturers, and send the Mach3 signals to a Motion Controller instead, using USB or Ethernet. The Motion Controller acts like a small computer and generates the signals required for the steppers. This is a good, workable, solution, which is increasingly being seen as the way forward. A preferred option might be the SmoothStepper, as mentioned in Part 1.

Taking all that into account, the basic choices are:

(1) The older, proven solution.

A desktop computer with a parallel "printer" port (photo 20), running Windows XP. Older computers are ideal, and you can add extra memory if required, increase the capacity of the hard drive if necessary, and add a

separate graphics card too (*if necessary* – see later), at relatively low cost. There are some notes on the support website.

Newer desktops, and desktops which were originally purchased by network users (such as universities, schools or government departments) may not have a parallel port, so check carefully. In a desktop with a larger case and suitable connectors on the motherboard it may be possible to add an extra card to provide a parallel port, but cheaper cards sometimes don't provide full functionality (i.e. not all the pins on the port are connected or respond as they should), so take care here. You may also find that a previous owner has set up the computer in a non-standard way, so there is some advice about viable options on the support website.

(2) *The Windows 7 32-bit solution.*

A computer which has a parallel "printer" port, running Windows 7 (32-bit). This might be a slightly older machine, capable of running Windows 7 (at least 1GHz processor speed, and at least 1Gb RAM (although I would advise more, if possible), 16Gb free space on the hard disk (so you will want 40Gb or more if you have only one hard disk, to allow for other software) and a DirectXe 9 (or higher numbered) graphics processor with WDDM 1.0 or higher (the Windows Display Driver). DirectX 9 was released in mid-2004, so really old computers are unlikely to be able to run Windows 7, but many from the last few years will be quite happy. I do not know of an easy way to tell if a graphics processor already runs, or is compatible with, WDDM 1.0.

Be aware, please, of the many comments on the website at www.machsupport.com regarding difficulties with Mach3 running under Windows 7. Results seem to vary by make and model of computer.

(3) Windows 7 and the SmoothStepper motion controller.

Increasingly, the way to go is with a more modern laptop or desktop running Windows XP or Windows 7 (32-bit), without a parallel printer port, but using an external motion controller (see Part 1) connected via USB or Ethernet. The motion controller of choice for this series is the SmoothStepper, simply because it works, and there are now lots of users. Amadeal, who advertise in MEW, sell a very neat ASUS computer and external SmoothStepper motion controller as a package, with an operating system and with Mach3 installed. I have no direct experience of that model of computer, but do have good experience of that make.

Other vendors may sell similar computers and/or motion controllers, although the tendency is to supply a computer with Mach3 as part of a package which includes a mill as well.

In essence, SmoothStepper acts like a separate computer. Mach3 sends commands to the driver for the main computer's printer port, and these are intercepted by the SmoothStepper driver and sent to the SmoothStepper via USB or Ethernet, depending on which version of SmoothStepper you have (fig 8). SmoothStepper then acts on those commands by creating its own set of signals for the stepper motor drivers. Those actions depend on the way the SmoothStepper has been programmed by its designers.

(4) Using Windows 7 (64-bit).

If you must run Windows 7 (64-bit version), the Mach3 support site states that you can only do this successfully if you also use an external motion controller. This seems to be because Mach3 has no parallel port driver software available for the 64-bit Windows7 OS. The core Mach3 program does run under Windows 7 (64-bit), its just that it can't get the output signals to the parallel port.

There are reports from happy users that if you use a SmoothStepper the SmoothStepper driver which replaces the Mach3 parallel port driver will work just fine under Windows 7 (64-bit). Parallel ports are not so common on Windows 7 computers anyway, so this is similar to the advice in (3) above. There are some comments from satisfied users, on the Mach3support website, as well as comments listing difficulties, and it does seem that the choice of computer has a lot to do with success rates. There is no list of makes or models of computer which work satisfactorily. Note that I have not personally run Mach3 under Windows 7 (64-bit).

(5) Roll your own.

Another viable choice is to assemble your own computer system. This is not at all difficult, and it will allow you to construct a desk-

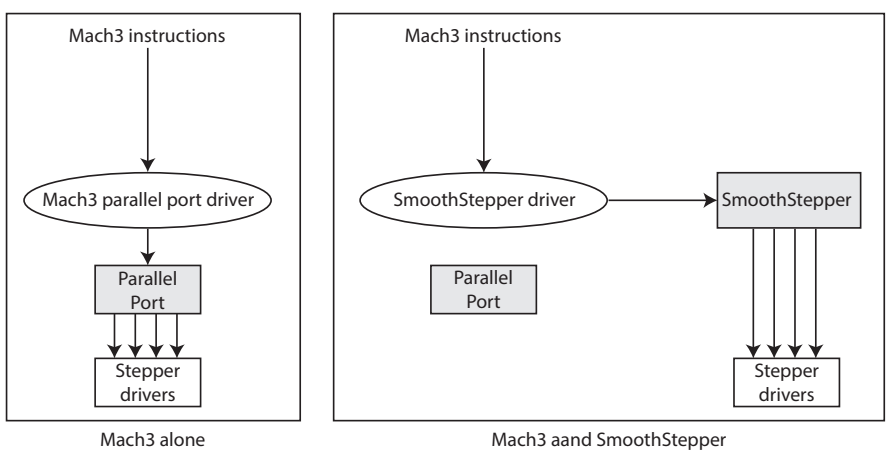


Fig 8: Signal path diverted by SmoothStepper driver.

top or rack-mounted computer which has a parallel port on the motherboard, running Windows 7 32-bit, and set it up specifically for Mach3. A few motherboards still have a parallel port built-in as standard (photo 21), and although those particular models tend to be slow by the standards of more up-market computers, and have limited facilities for expansion, they are low cost and readily available, at the time of writing, costing from £47 complete with processor, to around £80 by the time you have added 2Gb memory. A complete computer with that kind of specification, running Windows 7, should cost around £400.

Photo21: A modern single-board computer with a parallel port..

Windows 7 32-bit Home Premium is the appropriate operating system, because you will no longer be able to obtain a usable installation disk for XP, and I would caution you against any you may see for sale on the internet. When Windows is installed on a computer, it locks itself to the identity of the processor in that computer and cannot be used on any other computer. Despite assurances, it is very likely that any you now see for sale will not pass the Microsoft Genuine Advantage test, and although they can be installed initially, you will end up with a perpetual warning from Microsoft displayed on your screen. See the support website for further information.
A specification for a self-built computer for CNC appears on the support website.

Windows 8 is best avoided at present. There are lots of comments on MachSupport about problems preventing Mach3 from running under Windows 8.

And finally, the Mach3 website specifically says computers running Mach3 should use a separate graphics card. This is so that Mach3 can be relieved of interruptions caused by graphics processing taking place on the motherboard. That is a potential problem, but I have run Mach3 quite successfully on several desktop computers which do not have a separate graphics card. I suspect it is more of an issue with laptops and with specific makes or models of desktop. Run the demo version of Mach3 and find out.

INSTALLING THE SOFTWARE

Installing Mach3

If you have bought a CNC package, with machine and computer, Mach3 should have been set up for you, so you can skip this installation section. You may wish to drop in again when we deal with Setting Up Mach3 as that may give you an understanding of how Mach3 behaves on your machine. But before you go… let's just take a moment to discuss the issue of versions of Mach3.
Mach3 continues to evolve in response to user feedback. That's one of its strengths, but it also creates ongoing problems, as with any major software package. There have been many released versions of Mach3, and they come on two flavours. The first is the Lockdown Versions. These are versions which are considered stable, and these are really the only versions you should consider using. But as with almost all software, including packages from major software vendors, not all Lockdown versions are bug-free, and "stable" simply means "works in a known way". With complex software what tend to happen is that some bugs are fixed, but the fixes cause other knock-on effects and other bugs (or quirks in the system). This means some versions are more bug-free than others. In addition, the story of the development of Mach3 includes a change in personnel and methods of managing the development process. Some suppliers have decided to issue a particular Lockdown version which they are happy to support, but this is unlikely to be the current version of the software. If you are working with a mill and software bought as a package, it is most important that you do not change the version of the software you are using, otherwise the supplier may be unable to offer support. There is nothing unreasonable in that. Supporting any software is a complex and time consuming process, so it is perfectly reasonable to expect that you will lose support if you change the software. It's a bit like buying a Ford car and expecting after-sales support from the dealer. If you decide to change the engine for one out of a Vauxhall, it's not reasonable to expect the Ford dealer to be able to offer continued support. You will have invalidated the warranty by changing a major part. So, if you have bought a mill with software, don't upgrade the software. There may be minor variations in what you see on the screen, or in how the software behaves, but that should not cause major difficulty for you in following this series. One popular Lockdown version used by some suppliers is version 3.041 which is known to be reliable in its behaviour. Another flavour is V3.041 with a modified screen which makes some operations easier by changing the way the controls are laid out on the screen (more on modified screens much, much, later). If these are what you have installed, stick with them, please. If you change to the latest version, don't expect the supplier to be able to offer the same support. If you are itching to upgrade, check with your supplier first, and be guided by their advice.

If you do not have a version of Mach3 installed, the next set of instructions tells you how you go about getting it onto your computer.

Download the demo version of Mach3 from

the website at www.machsupport.com > Downloads>Mach3. The version you want is the current "Lockdown" version, and you can ignore the "Mach3 Changelog" which is simply a list of the changes from version to version. At the time of writing, the current Lockdown is Mach3 R3.043.066 but you should take whatever Lockdown version is listed.

Then scroll down the "Downloads" page to the section on Documentation and download

- Win XP Optimisation

Then click on the link to the Documentation page (it's embedded in the first line of that Documentation section) or Go to Home>Support>Documentation, and download the following:

- Mach3Mill Install and Config Guide
- Using Mach3Mill

You can ignore everything else, for the moment.

You will now have 3 documents, and a file to install Mach3.

There are good instructions for the installation process in the Install and Config Guide version 3. My firm advice is to allow yourself plenty of time, in a quiet place where you will not be disturbed. Pour yourself a cup of tea, and get as close to a calm Zen-like state as you can, before attempting installation. There is nothing difficult here, so give yourself a chance to enjoy the process and think about what you are doing. The second paragraph of section 1.1 is important, but some tens of thousands of users have succeeded so far, so you should feel confident that all will be well.

Note section 2.1.2.1 and disconnect the computer from the mill before you go any further.

The reason for this is that during installation and initial commissioning of the software, it is possible that signals may be sent from computer to mill, or at least the mill may think it sees signals from the computer, and may attempt to move table or head or power itself on. Experience suggests this is likely to be a small brief movement, but that's dangerous, and can take you by surprise, so disconnect the connecting cable at the mill. If you are using an external motion controller that's the cable from the MC to the mill, and not the cable between computer and MC.

Installing the SmoothStepper driver

If, and only if, you have a SmoothStepper motion controller, you should install the software for that after you have installed Mach3. The SmoothStepper comes in two variants, depending on the connectors on that device; USB or Ethernet. The Ethernet device does not require the installation of additional driver software, but the USB-equipped SmoothStepper does. For the moment, I will not deal with the Ethernet version of SmoothStepper, and all references to SmoothStepper mean the USB version, unless otherwise stated.

Refer to the instructions in the SmoothStepper manual to install the additional software for the SmoothStepper. Go cautiously here, and read the whole manual carefully. The manual is a developing work in progress, so there are screenshots and instructions for installing on an XP system, but not the same level of detail about Windows 7, just yet (I have version 1.0). Make sure you visit the downloads page, as suggested in section 5 of the SmoothStepper manual. Note the comments under "FTDI USB Driver for the USB SmoothStepper" referring to installation on a Windows 7 system.

Installing the Mach3 licence file

If you have bought a licence for Mach3, install it now. You should have received instructions with the file. Just copy it into the Mach3 installation folder. Licences can be bought and installed at any time, without losing any of your existing data.

Testing your computer's performance

First; it's worthwhile taking the time to run the Driver Test Program (Mach3 documentation: section 2.2.1). That will give you some idea of the performance of your computer, although it is not an absolute test, and I wouldn't become too despondent if your results are not perfect. The Driver Test Program gives you results which will indicate the degree of jitter in your system, amongst other things. Your system will not be jitter-free; I guarantee it. If you are running Windows XP, and the Driver Test Program gives a poor result, read the XP Optimisation file now. You might want to press on regardless and come back to any optimisation later, though.

And if you are running XP and Driver Test completely fails to run (just doesn't start up at all) you are in good company. One of my computers has run Mach3 perfectly well for years but refuses to run Driver Test. Just ignore it and assume all is well.

Setting up movement

Mach3 will send signals to the motor controllers to move slides, but we need to specify the correspondence between a pulse and a real-life slide movement. This is dealt with in section 4.4.2 of the Mach3 installation notes. Set that up now.

STARTING TO USE MACH3

The main Mach3 screen

When Mach3 starts up, you should see the main screen (fig 9). Technically, this is the Program Run screen, and the Screen Selection buttons along the top of the screen allow you to switch to any of the 6 available screens. Be brave and try that now. Then come back to the Program Run screen.

Before you go any further, you will need to deal with the Reset button. If you are not connected to a mill with an E-stop button,

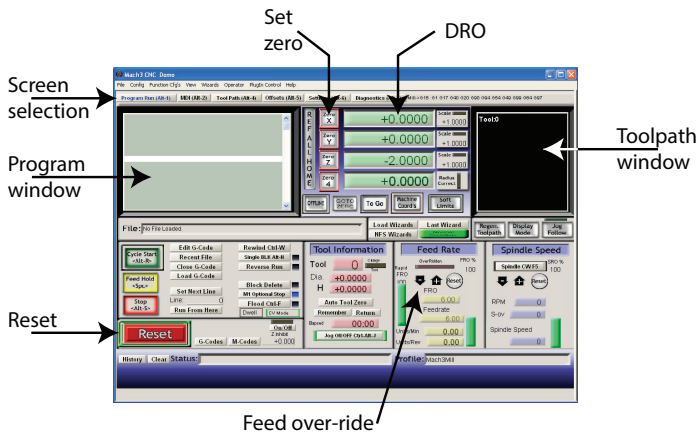


Fig 9: Mach3 main Program Run screen.

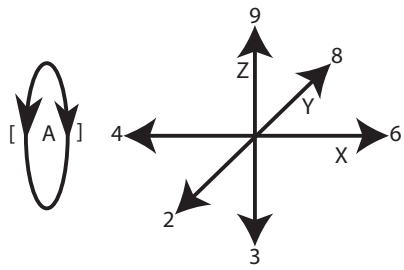


Fig 10: Keys for jogging.

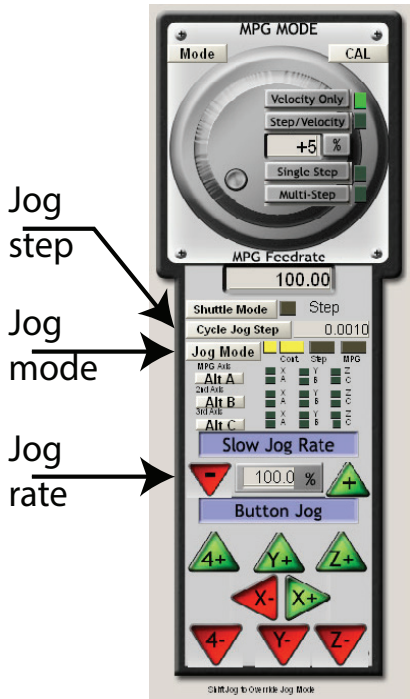


Fig 11: Mach3 jog controller.

or are running the software on its own, disconnected from a mill, you will see a message on the Program Run screen, next to the reset button, complaining about E-stop. E-stop is there for safety reasons, but can be worked around temporarily, so that you can use the software on its own. You MUST undo this and return E-stop to its proper function before connecting to a working mill.

Go to the Config menu > Ports and Pins > Input signals. You should see a table you have met before when setting Mach3 up. Scroll down to the line entitled EStop and put a tick in the Enabled column, and a tick in the Emulated column. You may have to scroll across to see the Emulated column. Note that if you try the logical strategy of putting a cross in the Enabled column, Mach3 will change that to a tick as soon as you leave this screen, to protect you. The manual says clearly that you should not Emulate an EStop, for safety reasons, but, as long as you are not connected to a mill, and you stop the emulation before using your mill under power, this will allow you to check the software works.

Setting HOME

Mach3 needs to know where the CP is in relation to the slides, at least at the start of a session, and the best way to do this is to set a reference position referred to as HOME, where the X, Y and Z axes are all defined as 0. Home can be set in any physical position. If your mill is fitted with Home switches, Mach3 can be set up to move the slides until the Home switches trip, then set itself so that point is X0 Y0 Z0. It does need to be set up to do that, though. If you are not connected to a mill, or if your mill does not have Home switches, you can manually set the DROs to 0.

For the moment, I will assume you do not have Home switches, or are not connected to a mill.

Click the Set Zero buttons on screen, one at a time, to zero the DRO readings. There is a lot going on behind the scenes, at that point, but we will leave the explanation until later.

Jogging

If we talk about "running" a program, maybe "jogging" makes sense as a slower movement. Jogging refers to moving the Controlled Point (CP) a little at a time, under keyboard control. Mach3 uses the numeric and symbol keys to move the X, Y, Z and A axes, as shown in fig 10, but it also incorporates some jogging controls on screen which are not replicated in the keyboard. Jogging is a frequently used facility, but the jogging screen is hidden, and you may want to write a sticky note and paste it to your monitor to tell you that:

Pressing the Tab key reveals the jog controller for each of the four axes (fig 11).

Press Tab now.

If you have ever sat in a railway carriage next to another carriage on the adjacent line, you may have experienced that strange sensation where it seems as though you are moving, when you are actually stationary and the other carriage is moving. You are about to replicate that, so watch closely. Although jogging moves the slides on the mill, its purpose is to move the Controlled Point in relation to a workpiece which is secured to the table. So focus on what the CP is doing, in relation to the axis being jogged. Imagine the CP was moving, and the slides were

stationary. The slides are stationary, as far as the workpiece is concerned, and the CP is moving in relation to that workpiece. It's just your eyes that are deceiving you; but don't get in the way of a slide, otherwise you will become even more confused when it hits you and you realise that although you are standing still, you are moving in relation to the slide and the workpiece.

Basic command moves

The next stage is to make the CP move in response to a typed command. A typed command which is to be carried out immediately is known as a Manual Data command, and the process is known as Manual Data Input or MDI, for short. You will use this kind of command frequently, and Mach3 has a specific facility for handling these commands. Amongst the Screen Switching Controls at the top of the Mach3 window (section 6.2 of the Mach3 manual, or see fig 9) is the MDI button. Click it, or hold down the Alt key on the keyboard and dab (press and release) the 2 key (referred to from now on as Alt2). That should take you to the MDI screen. Note that if you are using the software from Arc Euro Trade, the MDI input bar is on the main Mach3 screen, so you do not need to press ALT2 to see it.

Click to put the cursor into the MDI input box (fig 12). Strictly speaking, in computer terms, this is known as making the MDI input box the "focus", and that is a generic computer term for clicking in any window or input box to make it active.

The computer will respond to commands from what has come to be known as the G Code language. This is a bit of a misnomer, but it's the term everyone uses, so it will be good enough for us. G Code commands consist of a letter (usually G, but may also be M, S, F or O, depending on what we trying to instruct the machine to do) followed by a number, and may then be followed by other terms.

The G0 command is used to move the CP at rapid rate, and it takes the form:

G0 X~ Y~ Z~ A~ B~ C~

The ~ character is the "tilde" character, and it is used to signify that when the command is entered, there should be a number in this position, so you would never type the command exactly as it appears above.

The letters X, Y, Z, A, B and C denote the axes, up to 6 of them, but you would only include those you required for a particular command. Most mills will have X, Y and Z axes under computer control. Many, but not all, will have a fourth axis (usually a rotary axis) A. Very few will have B or C axes.

So typical G0 commands will look like:

G0 X100 Y20 Z40

G0 X-20 Y10 Z-0.02 A120

The numbers will vary depending on the position to which the CP is to be moved. Axes which are not listed after the G0 command will be ignored and the CP will not move in relation to those commands. For example:

G0 X70

will move the CP along the X axis only, to position 70. The position of the CP in relation to all the other axes will remain unchanged.

We are about to make some moves. Some of you will be working on small benchtop machines, like the little Sherline, while others may be working on a Bridgeport with a much larger table and a greater range of movement of the head. I don't know which machine you have, or where you have set HOME, so it is your responsibility to check whether a given move will be too large from your machine, or whether a cutter will hit an obstruction, BEFORE you use a command. This is a general rule, whether you are following the commands in these articles or using your own commands. Check before you carry out a move. It's a bit like the old adage "measure twice; cut once"; it's good practice and, given the cost of cutters which may snap, or the potential cost of rectifying damaged machine parts, it's a wise precaution. Experience suggests you may need to accept that mistakes happen from time to time, so practice saying "Ah well…" in a calm and measured manner; and keep that soothing mug of tea handy.

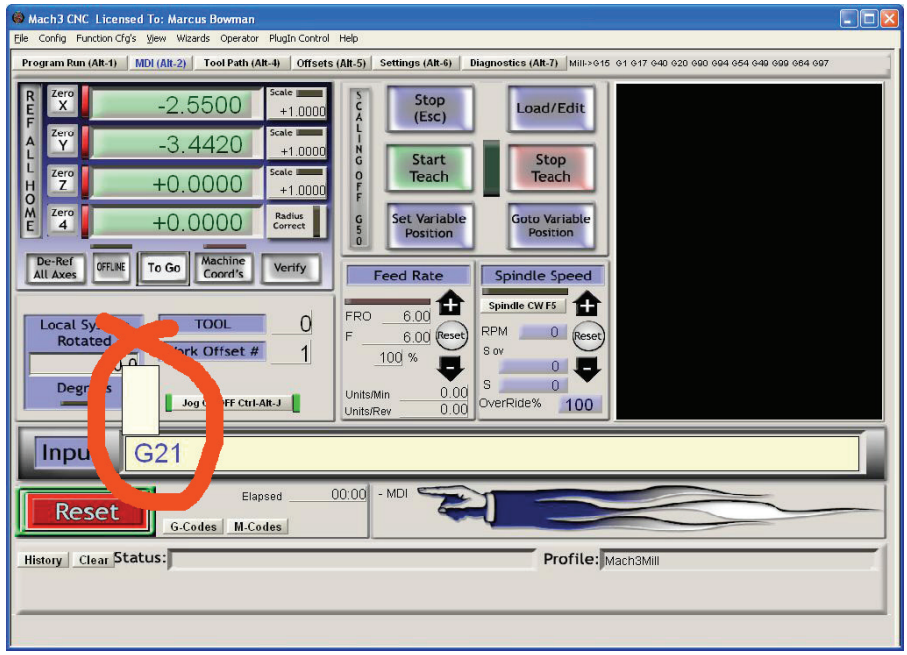


Fig 12: Entering G21 into the MDI input window.

From now on, all units are metric. If you are using inches, just divide the numbers by 25. I can no longer think in inches; I let it go when education went fully metric, in 1965. Even if you are determined to think imperially, I suggest you work in millimetres temporarily, by making sure the cursor is in the MDI input window then typing G21 and pressing the RETURN key. The G21 command makes Mach3 treat all distances as millimetres. When you want to use inches, type G20 and press RETURN.

My STRONG advice is to work in millimetres if you are following this series. Then we will not get confused. You could use the numbers as inches, but that would mean some of the movements would be much larger than your machine could cope with. Internally, Mach3 doesn't care what units are used. It simply moves a number of steps per unit. But for us humans, it helps if we call the units millimetres or inches, so that we can relate movement to distances that mean something in the real world.

To make a rapid move to X50, enter G0 X50 into the MDI input box then press the RETURN key. (I will not repeat the instruction to press the RETURN key to actually carry out a typed command; just take it as read from now on.) . That should result in the CP moving to the position where the X co-ordinate is 50. This is an absolute position in relation to the HOME position where X=0. Let's just test this is what has happened.

Enter G0 X0

then G0 X50

then G0 X50 again.

The CP should have moved back 50 units to where the X DRO on-screen reads 0.

Then it should have moved to where it read 50.

Finally it should have remained at X50 without moving.

If it moved to X100 and the X DRO reading is now 100, the CP is moving relative to the last position, rather than moving to an absolute co-ordinate position. Fix this by going to Config > General Config then looking in the second column, about half way down, for the Distance Mode box. Click in the Absolute radio button. While you are there, make sure the adjacent box "IJ Mode" is set to INC. Click OK. Then jog to a suitable position, zero the DROs, and try the movement test again (from the beginning of this paragraph). Incremental movement is useful, in some circumstances, but is not useful at this stage. Let's walk before we run.

The feed rate for the G0 rapid moves is set by Mach3 when it starts up, and was stored in a file created when you set up your stepper motors by following the instructions in the Mach3 manual (section 5.52). At this stage, it doesn't much matter what that speed is, but if it seems dreadfully slow, go back to the setup procedure and increase the speed. I set mine to maximum. You will also notice that the G0 moves begin with a period of acceleration and end with a period of deceleration. If those take too long, make the initial and final slopes on the display shown in 5.5.2 steeper. Realistically, the slides cannot accelerate from stationary to full speed in no time at all, because of inertia, and the acceleration rate will depend on the power of your stepper motors as well as the gearing between stepper and leadscrew. The important thing is that the movement of the slide keeps pace with the speed of the stepper as it accelerates and decelerates. Suck it and see.

G0 X0 Y0 Z0 takes you back to the Home position. Y and Z were probably still at their 0 positions, but stating where we want each axis to be as part of the command is a simple way to make sure we end up where we want to be, as opposed to where we assume we will be. There is a general principle here, and it is always wise to specify where you want each axis to be, even if we are fairly sure that some of the axes are there already. Certainty is better than assumption.

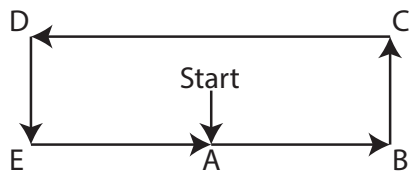


Fig 13: Controlled Point following a rectangular path..

Now move in a rectangular path (fig 13):

G0 Y-30

G0 X100

G0 Y30

G0 X-100

G0 Y-30

G0 X0

G0 Y0

Those commands changed only the values which changed between positions, and where X or Y did not change, they were not mentioned in the command. Mach3 only changed the values indicated by the X or Y.

So far; so good.

Go back to X0 Y0.

Of course the movements can be a bit more complex. Here's a path a bit like a sharp figure 8 lying on its side (fig 14)

G0 X100 Y-30

G0 X100 Y30

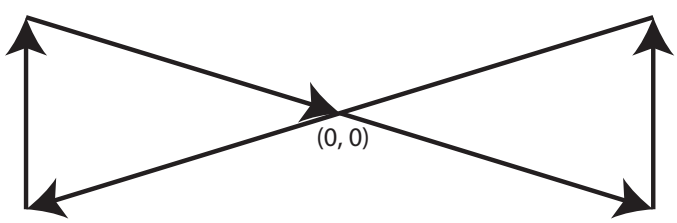
G0 X-100 Y-30

G0 X-100 Y30

G0 X0 Y0

Those commands specified the X and Y values of each point, even when one of those values did not change. It is easier to read and understand those commands, although it

Fig 13: Controlled Point following a rectangular path..



takes more typing.

And more complex still:
G0 X0 Y-30 Z50
G0 X100 Y30 Z50
G0 X-100 Y-30 Z10
G0 X-100 Y30 Z10
G0 X0 Y0 Z0

Here, the points on the path were fully specified in 3D space. This makes it very clear where the CP is to be, in 3 dimensions. This is always my preferred method, even if it involves a bit more typing. It is much easier to look at any line in the program and understand where the CP is supposed to be. That's another good principle: make your programs readable, so that when you return to a program later, you can understand what the program does. Memory is a wonderful thing, but I find that, as time passes, it is not always 100% reliable.

Cutting feed rate

G0 is a rapid move and is intended for moves which take place without cutting, as positioning moves, usually in preparation for a subsequent cutting operation.
G1 moves are at cutting feed rate (mm/minute or inches/minute); otherwise, it behaves just like the G0 command. The speed of movement for a cutting command is set by the F command (F being short for Feed) which takes the form F~ where ~ will be a number. The units are whatever units Mach3 has been told to work in; so if you are working in millimetres, F 100 would set the speed to 100mm/min. If you were working in inches, F100 would set the speed to 100 in/min.
The best place to put an F command is on a line before a cutting move, as that is easiest to see and understand if you are looking back at a set of instructions,
for example:

F100
G1 X75 Y20

but you can put the commands on the same line, in any order, and they will have the same effect. I find that a bit confusing, and not at all easy to read, but you need to know this for later, when we come to read programmes generated automatically by other software. So, the same effect can be achieved by typing

F100 G1 X75 Y20
or by
G1 X75 Y20 F100

Useful feed rates depend on the material being cut, the cutter being used, and the rigidity of the machine, but I have often found that a conservative feed rate in the range 25 to 100 mm/min is a useful place to start, for metals. More on feed rates later.