

# Coordinate Spaces - a Guide

Roger Caffin, with contributions from others

Understanding all the work spaces and offsets system in Mach3 can be bewildering at first. The Mach3 Mill manual tries to explain all this in Chapter 7, but sometimes it seems you need to have fully understood Ch 7 before reading it, so you can notice the bits in Ch 7 which really matter. It's a common problem, and the usual explanation is that the manual was written by the programmer rather than by a user. To make matters worse, the manual adds

'The final way of setting a work offset is by typing a new value into an axis DRO ...

You are advised not to use this final method until you are confident using work offsets that have been set up using the Offsets screen.'

Needless to say, the simplest method of resetting the coordinates is just that: typing into the DROs on the screen. I am sure many people start this way without understanding Offsets (I did). It works just fine for any simple machining task.

Unfortunately, if you want to do any macro programming, you find you cannot (easily) enter data into the DROs directly. Actually you can, using two different methods, including an undocumented feature which we will cover later, but not understanding how Offsets work can cause massive grief when your programming gets more sophisticated. So you have to RTFM. And you have to understand the difference between Machine Coordinates, the Controlled Point, the Workplace Offsets, 'other' Offsets and the Tool Offsets. So we will try to explain how it works.

I am approaching this from a CAD and mathematical perspective. The idea of 'spaces' is natural to me, but others will want a more mechanical or 'g-code' perspective. Sorry about that.

We will assume you have some familiarity with Mach3 already, so a few short cuts can be taken in presenting some commands (eg G0 and G1). After looking at the 'spaces' perspective we will reiterate a bit over the g-codes. We end with some unexpected details about hidden parameters in Mach3 as they do relate. This may be the most useful part of the whole document!

Please note: this has been written for a Mill. Mach3 for a Lathe is a bit different: for a start the job spins around an axis. You can access various offsets via the Tool Table, but there is no Settings screen.

## **Coordinate Spaces**

The whole idea of a coordinate space is fundamental to CNC use. Think of a sheet of graph paper laid out flat in front of you, with all those lines at right angles. The ones going sideways are conventionally called the X axis, and the ones going away from you are conventionally called the Y axis, and the ones going up and down are called the Z axis.

By convention, X values increase to the right, Y values increase going away from you, and Z values increase going upwards. You don't have to follow this convention, but if you don't then the conversation can get very confused. (I did start off back to front, but soon realised the error of my ways.)

Pick a point in the middle of that piece of graph paper, and call it the Origin. That means that  $X=0$ ,  $Y=0$  and  $Z=0$  at that point. (You will have to imagine the Z axis I am afraid: we are fresh

out of 3D graph paper.) Now you can specify any other point by its XYZ coordinates. A point X=4, Y=3 and Z=2 is often written as (4,3,2) for convenience. Such a point is 4 units to the right, 3 units away and 2 units upwards, all from the origin.

As to what units – that's up to you. Some use millimetres, some use inches, and I dare say a few might use something else. The difference is just a scale factor. A word of caution though: you really do NOT want to change the units you are using half-way through doing something. Utter confusion can result. And DO make sure you know what units you are using, or you might end up in the sort of trouble they had with the Hubble telescope. A single error with units there cost them a whole space shuttle mission to fix.

## **Controlled Point**

The Controlled Point is what any CNC program is ALL about. The Controlled Point is the middle of the tip of the cutter as it spins. You can think of it as being the tip of a tiny V-point cutter if this helps. A CNC program or machine moves this Controlled Point around. The fact that the cutter is really a lump of very sharp carbide some 20 mm in diameter and spinning at 3,000 RPM is incidental: hopefully any material in the way will be removed without damage.

This does mean that issuing a command such as `G0 X10` from X=0 will move the Controlled Point to X=10. That is the whole point of CNC after all. Normally the DROs on the screen show the values for this 'Controlled Point', so the command `G0 X10` will move the machine to where the X DRO shows 10 units. What really defines the value in the X DRO is more complicated. Note that while we will often talk about just the X axis, the comments really cover all the axes, even the rotary ones. I use 'X' as a short-hand.

In many places to follow I will use the term 'User Space' instead of 'Controlled Point'. User space or user coordinates or part space (the terms are largely interchangeable) is where you are working, and it may be offset from the Machine Coordinates. These offsets are the crux of the matter. Without these offsets a CNC machine would be worth only a fraction of its real value.

Also note that you can switch the DRO display to show 'Machine Coordinates' rather than 'User Space'. You will eventually need to do this. The equation across the top of the Diagnostics screen may help you understand a lot of this too.

## **Machine Coordinates & Home**

While the Controlled Point relates to the space you are working in or the part you are machining, Mach3 really has no understanding of that. It's a program and it knows little about the physical machine, but it does know how to move the Controlled Point around – by outputting signals which move the axes. The movement of the Controlled Point (ie the axes) takes place in the Machine Coordinate space, and this is what Mach3 uses internally for all its movements. The coordinate values seen in the DROs on the screen are in the User Space (program space, part space, etc) and are there for the user. Keep this firmly in your mind. We will go deeply into how they relate.

But any coordinate system has to have an origin: a place where X=0 etc. This place is called Home. Home can be set in two main ways: by moving an axis until a Home switch on the machine is triggered, or by telling Mach3 that 'right here' is Home. If you have defined Home pins in the Mach3 Ports&Pins screen, then the first method applies. If you want to load pallets onto the milling table with a robot for instance, you will have to have Home switches so the robot and the machine can relate to each other. A common place for the Machine Coordinate Home is

at the front left bottom corner of the available working volume. That puts the Home switches all at the ends of travel.

### Home Switches

If you do not have Home switches – that is if they are not defined in Ports&Pins, the second method applies. This is an arbitrary definition that ‘right here’ is ‘Home’. You get it by clicking on the ‘Ref All Home’ button on the Program Run screen. It could be argued that not having Home switches can be more flexible, but it all depends. It can be fine for a hobbyist, but it creates problems in a 24/7 production environment. Having Home switches which are not super-reproducible can actually be a hazard as the Home point may wander around a little.

### Going to Home – G28, G30, G53

You can go to the Home position in three different ways. G30 will take you there in a straight line from where ever you are now. If there is something in the way – tough, big crash follows. Be cautious with G30.

G28 will also take you there, but via an intermediate position given in the command. This *may* let you move around an obstacle – or it may not. Read the fine manual and consider carefully. Personally, I favour very explicit move commands in such a case.

You can also go to the Home position with the command `G0 G53 X0 Y0 Z0`. Including the G53 in the command tells Mach3 to work in the Machine Coordinate space rather than the Controlled Point space - for straight line moves (G0, G1). Done blindly, this could be the same as G30 – crash. Fortunately this method is far more flexible as you can put any XYZ values there, not just zero. In particular, the command `G0 G53 X0 Y0` (without any Z movement) can be very useful as we explain below.

## **Absolute vs Incremental Distance Modes: G90 & G91**

### **XYZ**

In absolute distance mode, the axis values (X~ Y~ Z~ A~ B~ C~) in a command represent positions in terms of the currently active User Space, *relative to the origin for that user coordinate space*. (The ~ sign means ‘some value’.) This is the normal mode of operation. You get into this mode via G90.

So a command `G0 X10` will send the Controlled Point to X=10 from wherever it was. It does not matter whether the Controlled Point was at X=-50 or X=+80: it will go to where X=10 in the current User Space. (The X value will be quite different in the Machine Coordinate space.)

In incremental distance mode (reached via G91), the axis values (X~ Y~ Z~ A~ B~ C~) in a movement command usually represent increments from the current position, but there are numerous exceptions. For example, the command sequence `G91 / G0 Z10 / G90` (where the slashes represent the New Line character) will move the Controlled Point up by 10 units *from where ever it was*. That is, in incremental mode you are not going to a point, on an axis, you are moving on that axis by a given distance. This can be very useful for small moves, like retracting the cutter upwards from wherever it was last. That said, the incremental space can be a dangerous space to work in for any length of time.

A reminder: ‘incremental’ applies only to *moves*, not to other things.

## **IJK**

Some commands allow you to use I, J & K variables in addition to or in place of X, Y & Z. The manual for G91 says that the I & J values are always incremental, regardless of the Distance Mode. The K value is also incremental except for G87 (a special potted boring command which is not defined in the Mach manual). Commands to machine circles and helices (G02 and G03) can use these IJK variables to some benefit: they specify the centre of the circle.

The alternative to using IJK values in G02 and G03 commands is to use the R format, where R is the radius. Some prefer the R format as it has slightly more flexibility.

However, the Mach manual for G90.1 and G91.1 gives a different mode of operation intended for use in G02 and G03 commands. (One can only imagine the intense in-fighting between different vendors supporting different definitions during the committee sessions which NIST used to create the Standard for g-code!) These two commands have been called 'Set IJK Mode' rather than 'Set Distance Mode'. G90.1 countermands the above, so that IJK values are absolute positions rather than incremental distances. Why one would want to use IJK in this manner is a bit strange, but some people like to machine circles around a specified centre. Perhaps it allowed some vendors to claim compatibility. G91.1 accords with the above, such that IJK usually represent increments from the current position. As far as I can see, G91.1 is the same as G91 - but I may be wrong as I have never used it. The best explanation I have seen (and about the only one) is that it 'returns I, J & K to their normal behaviour'.

Note: seeing large, strange and very unexpected circles in place of rounded corners in the Mach tool-path display usually means you may have the wrong IJK mode in the set-up. This happens. Some CAM post-processors are prone to this.

## **Machine Coordinates vs User Space: Workspace Offsets**

So how do these two spaces relate to each other? Basically as follows. We will use the term 'Controlled Point' here rather than User Coordinate because that's what the Mach manual uses.

Machine Coord = Controlled Point + Workspace Offset + Other Offset

or

Controlled Point = Machine Coord - Workspace Offset - Other Offset

What is the point here? What you are doing is moving the origin of the User Space. You might not want your job to have its origin at the Home position: you might want it offset a bit, for any of several very good reasons. Any time you position the cutter where *you* want the origin to be and zero the displays on the 'screen', you are defining an offset.

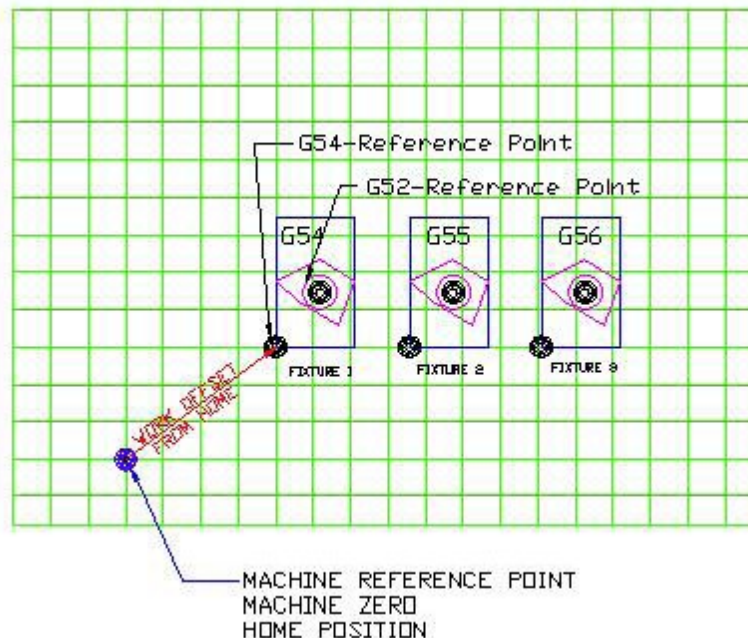


Figure 1: Workspace Offsets (drawing courtesy RICH)

So Mach3 (following the NIST Standard) has the idea of applying a Workspace Offset to the basic Machine Coord origin. Actually, Mach3 allows up to 254 workspace offsets (or 256 - the manual says different things in different places), with the first 6 being directly accessible by simple g-code commands. To go to Workspace #1, program G54; for Workspace #2 program G55, etc. Workspace #6 is reached by G59; to reach Workspace #7 you program G59 P7, and so on. The offsets are all stored in a Workspace Table, which can be saved. (Why not just G54 Pn? More committees?)

Work Offsets							
Work Offsets							
G-Code Pos	X	Y	Z	A	B	C	Name
G54	154.4985	137.0476	421.7326	1110.0000	0.0000	0.0000	G54
G55	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G56	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G57	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G58	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59P7	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59P8	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59P9	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	

Figure 2: the Work Offsets table

To set the XYZ offsets for Workspace #n you program G10 L2 Pn X~ Y~ Z~ A~ B~ C~; you can omit the axes you do not want to affect and they will not be changed. So the Workspaces are fully programmable.

The first form of the above equations is what Mach3 uses internally. When your program says go to X=5 (in the user or part space), mach3 has to convert this into Machine Coordinates.

It is worth noting that when you move the cutter so you can zero the DROs where you want the user space origin to be, you are in fact changing the parameters in the Workspace offset.

## 'Other Offset' – G52

There is also the 'Other' Offset. This has two components: the second one is dealt with later. For the first component, imagine you have a group of 6 parts to machine, all in a line at equal spacings. Instead of changing the Workspace values or Workspace number (which is indeed quite OK as a method), you can apply a temporary Offset to the base Workspace. This is done using `G52 X~ Y~ Z~`, where the XYZ values are the offset, the distance between the parts. What G52 does is effectively to move the User Space origin (again).

What happens if you program `G52 X~` with the same value a second time? Nothing. You have defined the offset, and that is it. Going into incremental mode does not change this because incremental mode only applies to movements. So if you program `G0 X0 / G52 X10 / G0 X0`, the controlled point (ie the cutter) will actually move along +10 units at the second G0 command (not earlier). You don't have to start at X0 of course: this is just a simple example. If I have a row of parts to machine, all 20 mm apart in a line, then `G52 X20`, `G52 X40` etc, at the start of a single subroutine, are the way.

There is another way of doing this: use `G92 X~`. This is a highly deprecated command, although it has its uses. What it does is to jam the given X value into the current X Coordinate Space DRO by altering the 'Other Offset'. In effect, G92 simulates typing new values into the DROs. But it can really foul up things if you try to mix G52 and G92 in a program, as they use the same special Offset variables! Total confusion is probable; some damage is likely.

A caution here: whereas the full sequence for G52 given above will move the X axis +10 units, the same sequence with G92 (`G0 X0 / G92 X10 / G0 X0`) will move the X axis -10 units (at the second G0 command). The X DRO was reading 0, the G92 command put +10 into it, so G0 X0 had to move 'left' to bring the DRO back to 0.

It can be most frustrating trying to set an axis value to something if you don't understand these offsets. Trust me, I know.

## Tool Offsets

While Workplace Offsets relate to the coordinate system inside the CNC, Tool Offsets relate to the cutter itself. They were temporarily included in the above under 'Other Offsets' for convenience. However, Tool Offsets are for the length of the cutter, so they only affect the Z axis. (There is an X parameter in the command, but it has little to do with the X axis.) Again, this can be used in at least two different ways.

If you are using tooling mounted in holders such as BT30, then the Tool Offsets can be effectively the distance from the BT30 reference plane to the tool tips. As you change tooling you change which Tool Offset you use. In more practical terms, the Tool Offset is the difference in length between some nominal Tool #0 and other tools. You don't have to go back to the BT30 reference plane. When using Tool Offsets, the rest of the Workspace stays unchanged. In particular, note that the Tool Offset value affects the Controlled Point; it does NOT affect the Machine Coordinates. (The latter relate after all to some real or virtual Home switches on each axis.)

The Tool Offset value is set by the command `G10 L1 Pn Z(offset)`, where 'n' is the tool number. The Offset is an entry in another table, also savable. You select which physical Tool you want to use with the command M6 ('change the tool') or with Tn M6 if you are using an ATC, but note that all this does is to change the physical tool: it does not change the Tool Offset. It is crucial to understand the difference. To change a Tool Offset you need to program `G43 Hn`, where n is the Tool number. (There is also a G44 command for use when the Tool offsets have the wrong sign - more committee in-fighting? I have not studied this G44 command.)

Can you load Tool 7 and Tool Offset 3? Yes. However, the results might not be exactly what you wanted (or wish to afford).

What this means is that the 'Other Offset' in the above section will also include the Tool Offset value, at least on the Z axis. Yes, that means there are no less than 3 different offsets to be applied to the Machine Coordinate to get the Controlled Point. Actually, the Tool Offsets Table also allows entries for tool wear in both Z and diameter, but Mach3 does not use this data afaik.

You can turn the Tool Offset system off by programming G49. Effectively that selects Tool 0, which has unalterable zero offset values. In many cases this will be perfectly OK, and may make life easier and safer. The Tool Offset table can also hold the Tool Diameter (X~), but that is not touched here.

And that leads to the other method of handling the concept of tool length compensation. You can turn the Tool Offset table off and do all offsetting via the Workspace. If you are sticking with just one or two manually-loaded cutters for the entire job, a direct calibration of a Workspace for each one is quite reasonable, especially for a fully manual operation. Position the tip of the cutter some fixed distance above the top of the part (feeler gauge, ground HSS rod, whatever) and type the thickness of the gauge into the Z DRO. Now Z=0 puts the tool tip just in contact. You can also do this using the Mach Mill Offsets screen.

## **Rotations – G15/G16, G68/G69**

Just when you thought you have it all worked out, someone goes and rotates the world on you. You can do that in g-code too. Caution: here be dragons.

Imagine you want to draw a ring of little lines like graduations on a dial. The first line is easy: you go from X=10, Y=0 to X=11, Y=0. But for the rest you have to do some trigonometry. Well, no: you can get Mach3 to do the trig for you. And Mach3 offers two different ways to do it!

### **Polar Coordinate Moves – G15/G16**

This first method tells Mach3 to interpret your commands differently. The command G16 tells Mach3 to go into polar coordinates; the command G15 cancels the polar coordinate mode. Thus the G16 part of the commands `G16/G0 X10 Y45/G15` tells Mach3 to go into polar mode, treat the X value as a radius and the Y value as an angle, and then drop out of the polar mode. The cutter will go to a point 10 units *from wherever it was beforehand* (the radius) at an angle of 45 degrees (to the X axis). That is, the starting position is treated as a temporary origin.

Caution: the Mach manual says you must not make X or Y moves other than with G0 or G1 when G16 is active. But of course X will mean radius and Y will mean angle in any such moves.

With the command sequence as follows you could drill a ring of holes (assuming Z is above the surface which is at Z=0):

```

G16                % polar mode
G0 X10 Y10         % go to radius=10 at 10 degrees
G81 Z-3.0          % drill hole
G1 Y20             % rotate to radius=10, angle=20
G81 Z-3.0          % drill hole
G1 Y30             % rotate to radius=10, angle=30
G81 Z-3.0          % drill hole
...
G15                % exit polar mode

```

Note that we do not need to change the X parameter as the radius stays constant in this example.

Having got somewhere and dropped out of polar mode, subsequent moves will be made along the conventional Cartesian axes. This might be more easily understood after the next section.

### **Coordinate space rotation – G68/G69**

Whereas G15/G16 tell Mach to interpret the XY values differently, the G68/G69 commands actually rotate the whole coordinate space around the specified position. The command `G68 A10 B20 R45 I1` causes the whole user space to be rotated around the specified A,B point by R degrees. In this case the user space will be rotated 45 degrees around the point (10,20). It would seem that they use A & B here (instead of X & Y) to avoid confusion. If the I parameter is present, the angle of rotation is incremental, otherwise it is absolute. G69 cancels all rotation.

The big difference is in what subsequent commands do. If G68 is in effect the X & Y axes in the Machine Coordinate space are rotated, but you can pretend that you are still working in an unrotated XY space. You can for instance use this to engrave numbers around a dial (using the Mach wizard for instance) such that they are always oriented to the dial markings. That's fine and very suitable for a *rotating* dial, but you would NOT use this to engrave the hours of the day around a watch dial. You don't want the hours rotated!

This is a very powerful command, but if you issue subsequent G68 commands about a different centre point I can almost guarantee that you will get lost. Good programming practice is to issue the G68, do what has to be done there, then immediately issue G69. Only after the G69 should you issue another G68 command. I have no idea what would happen if you tried to mix G15 and G68!

(Footnote: the Mach3 manual does have some typographical errors, including in this section.)

### **Test it for Yourself**

This is all very well, but you may need to see it for yourself to really understand it all. This is very easy to do. Fire up Mach Mill and go to the Program Run screen. Move the table and spindle to somewhere near the middle. If you have Home switches, you will have to edit the following a bit. I will assume you do not. I will refer to the on-screen DROs as 'User Coords' for convenience.

Click on the Ref All Home button. The DROs will change. Click on the Machine Coords button and you should have all zeroes in the DROs. Click again back to get the 'User Coords'.

Now zero each of the X, Y & Z axes via the individual buttons on the screen. Check that the Machine Coords are still zero.



Go to the Offsets screen. You should have Current Work Offset = 1 (from the G54 in the Init string). Active Work Offset should read G54. The Part Offsets should all be zero. The user coords DROs above the Machine Coords button (on the right) should all be zero. Click on the Machine Coords button and you should still have all zeros.

Now go back to the Program Run screen and move the X & Y axes a bit. Alternately, go to the MDI screen and enter G0 X10 Y-15 (assuming these values are reasonable). The screen DROs should update. Leave the Z axis alone for the moment: you will see why shortly.

Click on the Machine Coords button and the DROs should not change. You have moved the Controlled Point a bit after all, and both User Coords and Machine Coords should reflect this.

Go to the Offsets screen and look at the DROs. Current Work Offsets will still be zero, since you have not changed anything there. The User Coords on the right will have changed. So far, as expected.

Now zero the User Coords on the right. They will go to zero, but the Current Work offsets will now show whatever the User Coords were. Why? Because you have not moved the machine (or the Controlled Point) when you 'zeroed the user DROs', and you have not moved the Ref Home position. If you want the user DROs to read zero, then Mach3 will change the values in the Work Offset to accommodate you.

Type a zero into one of the Current Work Offsets DROs. The value which was there will move back to the user DROs. It has to: you still have not moved the machine. You are still at the same distance from the Machine Home.

Tool Offsets are a shade more complex to understand at first. The Tool Offsets area is at the bottom right hand corner. The Tool DRO should read 0 and the Z offset should read 0.000. You cannot change the offset value for Tool 0. Also, if necessary, click on the Zero Z button on the user DROs. This all says that the Controlled Point for Tool 0 is at Z=0. We will call that the surface of the part.

Now type 1 into the Tool DRO, type 5 into the DRO under Gage Block Height and click on Set Tool Offset. You should get -5 in the Z offset DRO and +5 in the Z DRO. This says that if you have a Tool 1 which is 5 units shorter than Tool 0 when mounted in the spindle but you have NOT moved the Z axis, then the tip of Tool 1 will be +5 units above whatever was the surface of the part. A 5 unit high gage block will just fit under Tool 1. While you are at it, check the current Machine Coords: Z will still read 0 because you still have not moved the machine.

To bring the tip of Tool 1 down to the surface of the part you will need to apply a Z Offset of -5 units. This is now obvious from looking at the picture of the gage block. Change Tool number to 0 and the user Z DRO will go back to 0.

You can also test G52 here. Let's assume Work Offset 1 has a Y value of 0 and user Y DRO has a value of 0 (so Machine Coord Y=0). Go to the MDI screen and type G52 Y5. The user Y DROs will now read -5. You have told Mach3 to move the user origin for the Y axis 5 units north. The Machine Coords will still show Y=0, because you have not moved the machine. But if the USER origin is now 5 units to the north, while the machine has not moved, then the current user Y coord must be -5 units.

You cannot see the G52 value on the Offsets screen, but you can see it on the Diagnostics screen under the G92 label. Curiously, while you can change other DROs on the Diagnostics screen, it seems you cannot change the G52/G92 DROs there. This may be a quirk of Mach3.

## **Parameters**

Mach3 allows about 10,000 user parameters (like #10, #123, #1000 etc). However, there are also some special cases outside the 10,000 range which seem to be poorly documented. Obviously, these are Mach3 special bypasses, and may not be in the NIST spec. If you know of any more, please, let me know!

Parameters #500 - #600: these seem to be saved in the XML file and restored the next time you fire up Mach3. They are 'persistent'.

Parameters #5211 - #5216: these give you direct access to the offsets for the G52/G92 commands. Touching these without really, really knowing what you are doing could be exciting and expensive. Exactly how these parameters work is actually not entirely clear, so writing to them may or may not do what you want. Better by far to use G52 explicitly – although being able to read the current offsets may have some value.

Parameters #15001 - #15255: these are linked (to a certain extent) with User DROs 1001 – 1255. You would normally only need to use these if you are doing some fancy screen creation.

Parameters #99nnn: these are also special, at least for some values of nnn. Noting that the addresses for the X, Y, Z DROs etc for macro commands are 800, 801, 802 etc (ie for the get/setOEMDRO commands), it turns out that you can also access the on-screen DROs from g-code by using the parameters #9980n. I have not explored other values for nnn. So #99802=0 is the direct equivalent of G92 Z0.

## **Practical Defaults**

Not knowing which Workspace offset you are in can make life difficult, but do not despair. The default initialisation string for Mach3 includes a G54 command. This will select Workspace #1. Many people will never leave that workspace – and will never *need* to leave that Workspace. That's fine. Not knowing what Tool offset you have set up can be equally exciting, so the default initialisation string for Mach3 also includes a G49 command to turn the whole Tool Offset system off. You can in fact do everything you want to do in the G49/G54 space alone.