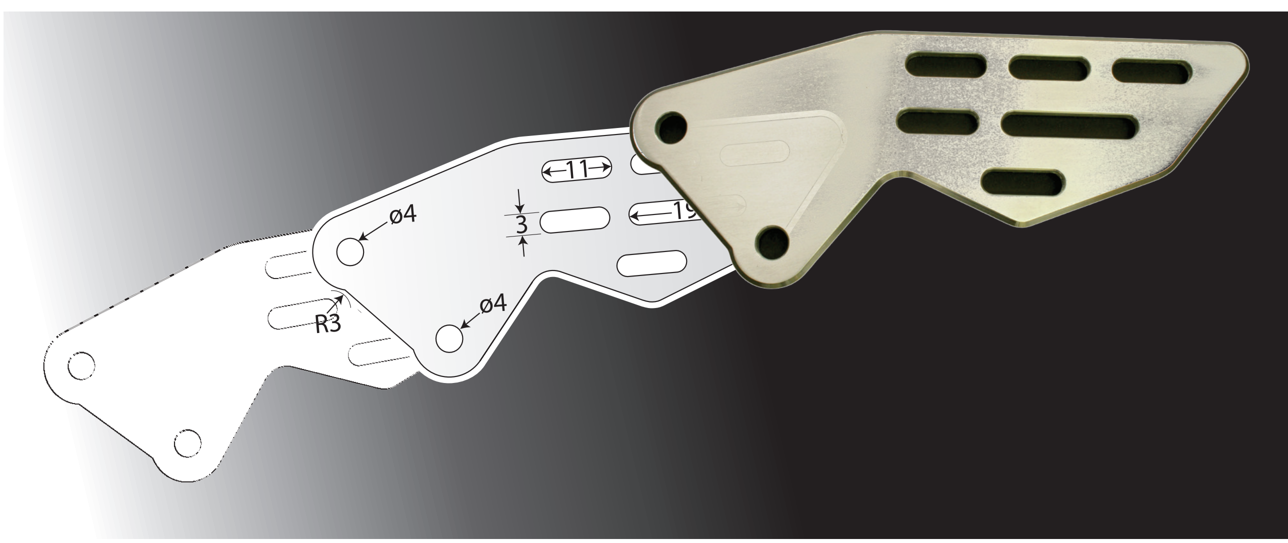
CNC in the Workshop



Part 6

Part 6

In this part of the series, we finish making the Vice Plate. Although this is a simple part, it helps illustrate many of the basics of using a CNC program.
The instalment progresses to making a CNC vice stop to provide a reference face for use when positioning work.

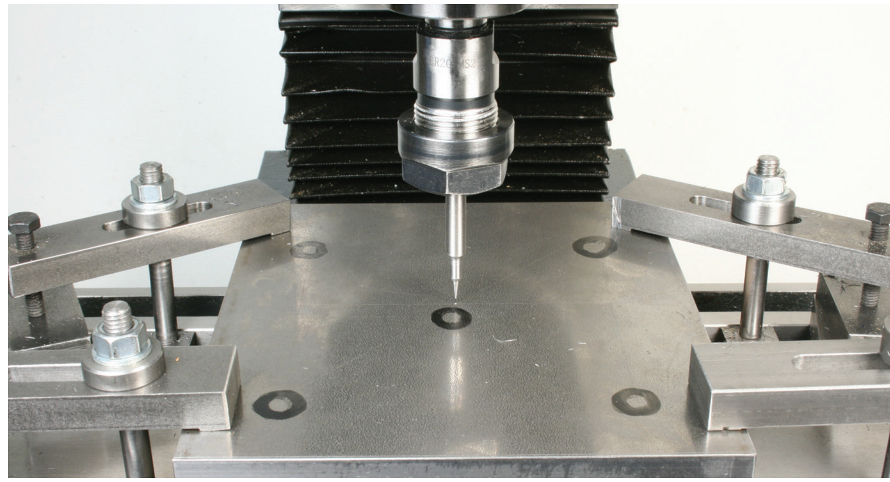


Photo 45: Using a pointer to indicate the centre of the plate.

Drilling the holes

Mark the centre of the plate. I used marking out blue, and scribed lines from opposite corners. Absolute accuracy is not important here; just get it reasonably close to the centre. Centre pop using a centre punch.
Although we will attempt to eliminate marking out from all our jobs, it is useful on this first job to mark the approximate positions of the holes to be drilled on the plate. I used a thick black marker pen. Set the plate on the table with one reference edge parallel to the Y axis and the other parallel to the X axis, and clamp it securely, avoiding the marked hole positions.

Use an indicator of some kind to set the CP directly above the centre of the plate. I used a pointer made from a short section of silver steel (photo 45), but you could use a small drill or a small centre drill. Absolute accuracy is not essential, for this project. Zero the X and Y DROs in Mach3. The holes in the plate are symmetrical left-to-right, but sit a little bit closer to the front of the plate. In fact, the central hole is 8.5mm closer to the front edge than the centre of the plate. (The central hole is 75mm from the front edge, but the centre of the plate is 82.5mm from the front.) Move the CP by typing G0 Y-8.5
It should now sit above the position of the middle hole. Zero the Y axis DRO.
Two things to note:

- Each of the axes can be zeroed anywhere, and can be re-zeroed if necessary. There is a lot more to this, but we'll get to it. This is a useful facility because it lets us set our zero point to suit the work and the position of the work on the table.
- G0 and G1 commands don't always need an X and a Y and a Z value. They only require the new value of the axis for which a move is to take place. Any value not stated will be assumed to be the same as the current value for any axis.

The next set of operations uses twist drills, so mount a drill chuck in the spindle, or use ER-style collets.
We have a choice, now. The task is to move to each hole position in turn, and use the centre drill to create a shallow starter hole (say 6mm deep, so that there is at least a bit of a countersink at the entry). The simplest way to drill the hole is to use the spindle hand lever to drill the hole manually, but the mill can also do this by feeding the had downwards. To drill this hole using Z axis movements, the sequence is:

move to the hole
rapid move downwards until the drill bit almost touches the surface of the plate
move at cutting feed rate a short distance (say 3mm)
pause to break any continuous chips
move at cutting feed rate a further short distance (say another 3mm)
rapid move out above the top surface.

Set the Z height of the centre drill using a piece of cigarette paper or other thin paper. Place the paper on the surface of the plate. Jog down until the tip of the centre drill almost touches the paper. Alter the Jog Step to 0.01 and the Jog Mode to Step, then jog gently downwards, one step at a time, until the paper is just trapped by the drill. I usually slide the paper a little after each step, and when it stops sliding I stop jogging. Set the Z DRO to 0.
Table 2 shows the co-ordinates of the holes to be drilled, assuming the CP is above the centre hole in the pattern, at X0 Y0.

Hole reference Number	***Co-ordinates X, Y***
1	0, 0
2	-40, 0
3	40, 0
4	-50, -50
5	-50, 50
6	50, 50
7	-50, 50

Table 2: Hole co-ordinates

I used a BS3 centre drill, with a 3.2mm diameter pilot section.
Spindle speed: 3000rpm   Feed 150mm/min

(for a chip load of 0.025)
Note that this pushes the drill hard and fast, and you may wish to back off the feed rate quite a bit, to bring it into your comfort zone. But if you are brave, its worth joining me on the scary side.

Using G Code, the instructions for the first hole might be:

G0 Z1 (to take the drill above the work)
G0 X0 Y0 (it may already be there)
S3000 M3 (speed 3000rpm, start the spindle turning clockwise)
F150 (set the cutting feed rate)
G1 Z3
G1 Z6
G0 Z1
M5 (stop the spindle if you are drilling just one hole)

If you are typing these commands in MDI mode, you will find that your typing speed can't keep up with the drill descending, so that will effectively introduce a pause between Z3 and Z6. That's good enough for now.

Move to hole 2. Take care that the path you intend to move along is clear of obstructions. You may be able to use G0 X-40 if the way is clear. Now type the same instructions you used (above) for the first hole.

Move to hole 3 (G0 X40) and repeat.
Repeat the procedure for the remaining 4 holes, but take care that the drill will not hit any obstruction as it moves from one hole position to another. Adjust the clamps, if necessary.

All of that typing surely numbs the finger tips, so we will take a significant shortcut next. Instead of typing the same sequence of instructions at each hole position, we will enter the sequence just once, then tell Mach3 to repeat the sequence each time we need it.

Each of the holes can now be drilled to final size, one after the other, using this sequence for each hole:

move to a hole
rapid move downwards until the drill bit almost touches the surface of the plate
move at cutting feed rate a short distance (say 3mm)
rapid move out above the top surface, to clear the chips
rapid move downwards until the drill bit almost touches the surface of the plate
move at cutting feed rate a longer distance (say 10mm)
rapid move out above the top surface, to clear the chips
rapid move downwards until the drill bit almost touches the surface of the plate
move at cutting feed rate through the plate (say 18mm)
rapid move out above the top surface

The first hole is the centre hole, and that is 6mm diameter.
There are two holes to be drilled tapping size (6.8mm).
The other four holes are all the same diameter (8.2mm).
That means all seven holes can be drilled to 6mm diameter, then holes 2 to 7 can be drilled to 6.8mm diameter, then the four corner holes 4 to 7 can be drilled 8.2mm.

Change the centre drill for a 6mm drill. I used a stub drill, but if you do that, you should first check that your clamps will be clear of the nut on the collet holder, because it will descend close to the table. A jobber length drill is fine, because we have already drilled a shallow pilot hole at each position. Set the tip of the drill at the top surface of the work, using the cigarette paper, and zero the Z DRO.

We will use a spindle speed of 1500rpm and a feed rate of 75mm/min.

Go to the MDI screen.

Click the OFFLINE button. This allows us to type commands but Mach3 will not send any signals to the steppers. In computer-speak the computer is "offline" so its not sending signals "down the line" to any other device.
Click the large Start Teach button. This allows us to teach Mach3 a sequence of instructions which it will store for later use.
Type these commands (but read the note in the box below so that you know what to do if you make a mistake):

Typing mistakes

Mistakes at the MDI screen:
When you are typing into the MDI command box, you can edit as you type, but once you press the Return key Mach3 memorises that command.
If you realise you have made a mistake, the easiest way to deal with it, for now, is to do this:

Click Stop Teach
Click Load/Edit
Go to the Program Run screen.
Click Stop
Click Close G-Code
Go back to the MDI screen and start the teaching process again.

Mistakes at the Program Run screen:
The Program Run screen's Program window shows you the sequence Mach3 has learned. If there are mistakes in that sequence, do this:

Click Stop
Click Close G-Code
Go back to the MDI screen and start the teaching process again.

There are better ways of dealing with mistakes, but for this initial exercise the best way is to follow these instructions to dump what you have done and start again.

F75
S1500 M3
G0 Z0
G1 Z-6
G0 Z1
G0 Z-5.9
G1 Z-12
G0 Z1
G0 Z-11.9
G1 Z-20
G0 Z1
M5

Click the Stop Teach button.

Now that Mach3 has a sequence of instructions, we need to send it to the Program Run screen.
Click the Load/Edit button.
Then go to the Program Run screen, where you should see your list of instructions in the Program window (upper left).
Check those instructions carefully. If you need to make any changes, at this stage, read the note in the box on the previous page.
At the moment, the computer is still OFFLINE, and the OFFLINE button will have a flashing yellow outline, so cancel that by clicking the OFFLINE button again, to cancel OFFLINE mode.
We will now use MDI mode to type commands to take the CP to a hole position; then we will make Mach3 use the sequence of instructions to drill the hole. That basic process can be used for all the holes we want to drill with this size of drill bit.

Go to the MDI screen.
Type G0 Z1
then
G0 X-40 Y0
to put the CP above hole 2.
Go to the Program Run screen. Click Cycle Start. When the hole has been drilled, click Stop, then click Rewind.
Go to the MDI screen, move to hole 3, return to Program Run, Cycle Start, Stop, Rewind.

Repeat for holes 4 to 7.

If, at any point, you find MDI mode will not carry out your G0 instructions, but simply beeps instead, go back to program Run and click Stop. Then go back to MDI and it should work.

Change to a 6.8mm drill, and set the Z height. We could use the same sequence of commands to drill the holes, but the feed rate and the spindle speed should be adjusted just a touch.

At Program Run, click Edit G-Code. That should put the sequence into a text editor (probably Notepad). The text editor is set using the Config > General Config menu, so if you are not seeing an editor you fancy using, go to that menu and change the Editor using the box near the top of the second column on the menu screen. Unless you have a compelling reason for choosing otherwise, leave it set to the default Notepad for now.

Change the feed rate to F70
Change the speed to S1470 M3
We can also eliminate the Stop and Rewind button clicks by adding M30 after the last line of the sequence. Press Return after typing that command. M30 signals the end of the sequence, and makes Mach3 rewind to the start of the sequence, automatically.
File > Save
File > Exit
That should put the amended version of the sequence into the Program screen.

Now, for holes 2 to 7, repeat the process of moving to a hole and making Mach3 carry out the sequence.

Change to an 8.2mm drill, set the Z height, edit the sequence, amending the feed to F60 and the spindle speed to S1200 M3 then drill the remaining "corner" holes (numbers 4 to 7).

Photo 46: Tapping the two holes with the plate secured to the mill table.

For the icing on the cake, use MDI commands to drill a shallow counterbore on holes 2 and 3, in preparation for tapping the thread. Use the 8.2mm drill and feed at F60 to Z-3 then retract.

Remove the plate and tap the two holes by hand. Some mills can tap under power and under the control of Mach3, but let's not get over-ambitious just yet. I bolted the plate to the end of the mill table and tapped the two holes by hand (photo 46). Clockwise a turn, back a quarter turn; repeat until done. Gently deburr the other holes.

The pin and boss

The pin is simply a length of 6mm silver steel, or something similar.
The boss is a simple turning job. Turn it to fit the hold in the underside of the vice, then drill a 6mm hole right through. You could drill 5.9mm and ream the hole, for the deluxe version.
Loctite the pin in the hole. My advice is to assemble it dry, to check the fit, before applying the Loctite, because you must not hesitate in pushing the rod fully home before the Loctite grabs.
When the Loctite has bonded, slip the projecting 6mm rod into the 6mm hole in the plate. I didn't bond it into the hole, but you can please yourself. Grease or oil the projecting boss.
Before mounting the vice on the plate, put

bolts of a suitable length into holes 4 and 5 because they sit under the overhanging vice body and are difficult to fit afterwards. Check the bolts are of a suitable length. Mine are 33mm long. Don't forget the washers.

Setting the vice on the plate

Using a square, set the long reference edge of the plate on the left side, square across the table. The other reference edge should be nearest the vertical column and parallel to the X axis. Bolt the plate firmly to the table, in that position. To be efficient, cut some bolts the right length to secure the plate.

Cut some bolts long enough to go through the lugs of the vice and the plate, but not so long that they will protrude through the plate. Leave then a couple of millimetres short.

Remove the rotating base from the vice, then locate the main body on the boss projecting from the plate and bolt it down using the lug holes. Leave the bolts tight enough to just stop the vice rotating, but no tighter. Now clock the rear jaw of the vice parallel to the X axis. Nip the lug bolts up tight, then check the clocking again.

The plate makes it quick and easy to remove or replace the vice on the mill table.

Do another one while you are at it, for an identical second vice. Very useful for long work. The second one can always be used on the table of the drilling machine where the plate provides a handy way of clamping the vice and more opportunities for adjusting the position of the vice on the table than using the lugs on the rotating base.

VICE STOP BLOCK FOR REPETITION WORK

One of the joys of a CNC machine tool is that once you have a sequence of instructions to produce a finished piece of work this same sequence can be used to make a second identical item, and a third, … and so on. This can radically transform what we do in the workshop, because it means repetitive work becomes relatively easy. Yes; conventional manual machine tools benefit from setups designed to produce repetitive work, with end stops, accurate measuring scales, pre-set tools and the like, but they are not in the same league; not by a long way. You can see the effect in modern manufacturing techniques which have brought us mass produced items, manufacture on demand, and useful outcomes like low cost machine tools, cheap workshop accessories, and kit locomotives. There are two requirements to do repetitive work on a CNC machine conveniently. The first is a suitable sequence of instructions and a way of easily repeating some or all of those. The second is a way of repeatedly mounting a series of workpieces in the same place in the machine. Let's look at both of those, and let's do it by making something useful for repetition work, a vice stop block.

Designing the vice stop

Vice stops can take several forms, but this is a straightforward block of material attached to the left hand side of the vice (photos 47 and 48, and fig 24). There's a reason for mounting it on the left, but you could, if you wished, attach it to the right hand side, or you could be ambitious and make arrangements for it to be mounted on either side.

In fact, starting with the height of 37mm (fig 24) and subtracting the 10mm from the top

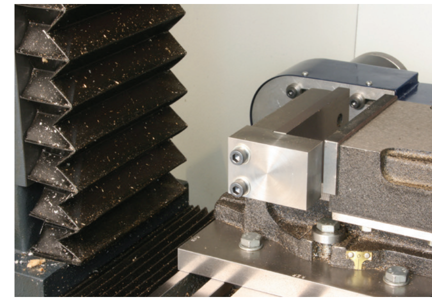


Photo 47: Vice stop block mounted on the vice.

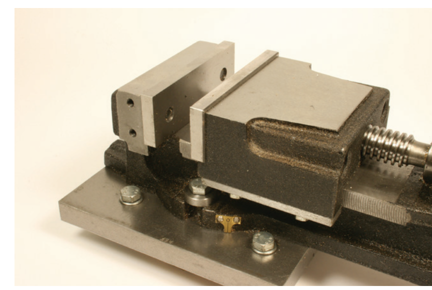


Photo 48: Two additional attachment holes required in the side of the vice body, behind the rear jaw.

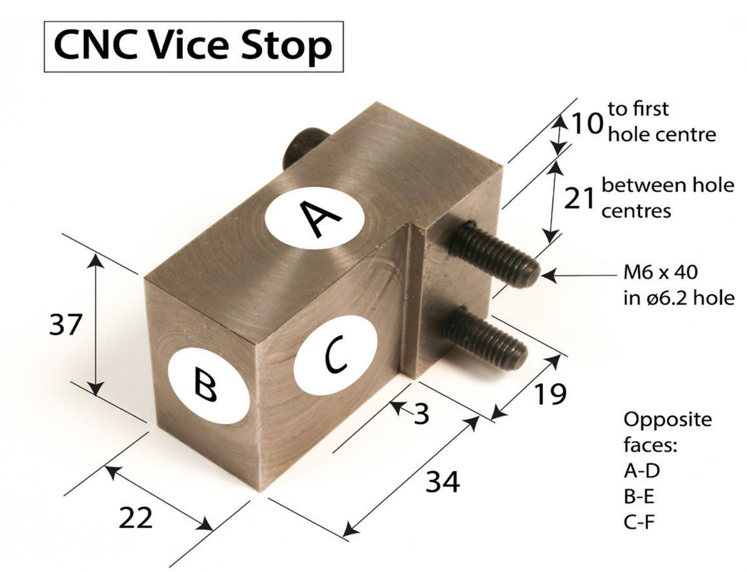


Fig 24: Dimensions of the stop block..

to the first bolt hole, then subtracting the 21mm between bolt centres reveals that the distance from the bottom bolt hole centre to the bottom of the stop is 6mm. That's the distance from the top of the body of the vice to the first attachment hole on the side of the vice. When the stop is attached to the left side of the vice, it is level with the top of the jaw. When it is attached to the right side, the bottom becomes the top (to keep the step facing inwards), and it is level with the top of the vice body because the tops of the jaws are 4mm higher than the body. Despite feeling this is a cunning aspect of the design, I have not made much use of it.

The inner flat face "C" sits at right angles to the rear jaw, but there is a little gap between the left end of the rear jaw and the inner face of the stop, as shown in photo 49.

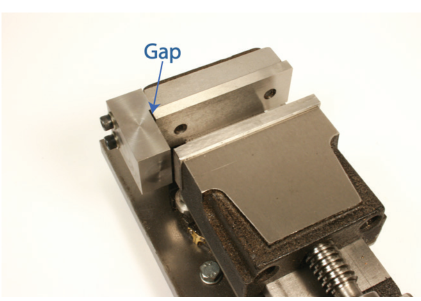


Photo 49: Gap between the end of the jaw and the inner face of the stop.

That little gap allows for any burr on the workpiece, so that the main end face of that work can butt accurately against the stop despite burrs which may have been created by a previous machining operation. Yes; burrs should be cleaned off after each machining operation, but although that should still happen, the job won't suffer just because of a minor omission or discrepancy. If you look at a good quality engineer's square, you will see the same feature right in at the corner where the blade meets the stock (photo 50), and we would do well to emulate professional practice. That gap also allows plenty of clearance for the moving

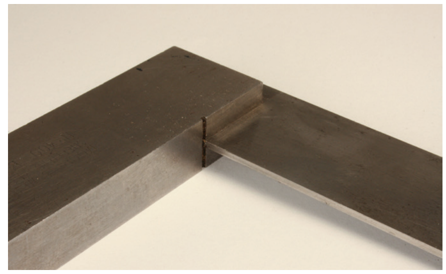


Photo 50: Clearance groove between the blade and stock on an engineer's square.

jaw to move in and out without fouling the stop or dragging against its inner face.

The vice stop is screwed to the side of the vice so that it remains firmly attached. The body of the vice is cast, so we need a good compromise between the diameter of the securing screws and the coarseness of the thread, so M6 x 1 is about as fine as we can go. The material for the stop can be chosen according to what is in the scrap box. Steel is ideal, aluminium will be fine, although it is less resistant to abrasion, and brass would be best polished and mounted on a mahogany plinth. All sizes are approximate and can vary significantly, but although we will not be banging workpieces forcefully against the stop, there's no point in making the thing flimsy. We need reliable repeatability. When in place, the top of the stop does not protrude above the top of the vice jaws. The depth of the stop (top to bottom) is enough to ensure that a thin piece of work sitting right down at the bottom of the jaws can still be located against the stop.

Strictly speaking, we only need to machine the inner face of the stop (face C), and the rear part of that face, which locates against the vice. To ensure ultimate accuracy, we should also machine the side of the vice, where the stop will attach, but we will treat that as a separate job. However; because we take pride in our work, we will machine all the faces of the stop. It won't take much longer; and it's a good excuse to practice some useful techniques.

Photo 51: Facing cutter with3 inserted carbide teeth.

Facing cutters

Each face can be machined using a facing cut, in which a cutter passes over a surface, cutting as it goes. That could be done by using the end cutting edges of an end mill, but it is easier and quicker to use a facing cutter. Photo 51 shows a facing cutter which has 3 inserted carbide teeth. This is an Arrand tool which I bought many years ago, and it has been very useful. Other makes are available, of course. The rear of the cutter is tapped 3/8 Whitworth or 10mm (3/8W in this case) to take a drawbar. A drawbar is essential with a cutter like this, because the large diameter means the teeth each exert a sizeable off-centre force trying to wobble the cutter loose, and in fact it is wise to use a drawbar with all cutter holders. 3/8W survives as the most common standard size for small cutters

Photo 52: Metric and imperial drawbars.

and holders, but the world is gradually moving to the alternative 10mm thread. When Bridgeports have finally gone, we can probably move over to metric threads for drawbars, but we are a long way from that, so we must live with the dual standards for now. My mill was supplied with a metric drawbar, but one of the first accessories I made was a 3/8W drawbar (photo 52), and although

I have tried to ensure metric threads on all accessories, that has not been possible. It's a minor irritation.

Photo 53: A flycutter..

Photo 53 shows a flycutter. This is a single-tooth cutter in which the tooth sweeps a sizeable diameter. Although it has just a single tooth, it does the same job as the 3-tooth facing cutter. One important difference, though, is that the shape and geometry of inserted carbide teeth is fixed and although there is sometimes a choice of tip radius, that's almost the only choice we can make. The flycutter tooth, on the other hand, is made of High Speed Steel (HSS) and can be ground to any geometry, so the tip radius, nose shape, side and back rake can all be controlled quite easily, especially if you have a good tool and cutter grinder. The downside is that you will have to make the tooth. In practice, I use the 3-tooth cutter for roughing, although it does give a decent finish, and the flycutter, with its tooth ground to a more generous tip radius, for finishing. On aluminium, that flycutter can give a mirror finish so shiny you won't want to touch it with your greasy fingers. The inserted teeth have a tiny radius, and the width of the chip they remove per tooth is small compared to the broad sweeping curve on the flycutter's tooth. That's part of the reason why the flycutter can produce such a superb finish. The single tooth presents a considerable out-of-balance load, though, and so cannot run as fast as the 3-tooth cutter, even when sweeping the same radius. In any case, with only one tooth, the calculated feed rate will be 1/3 of the feed rate for the 3 tooth cutter, for the same tooth chip load.

A large diameter cutter will be able to cut the surface in one pass, but a smaller diameter cutter may need two or three parallel overlapping passes at each height setting, like ploughing parallel furrows in a field (but leaving a much smoother finish, we hope) as shown in fig 25.
On a conventional mill, having to take lots of light cuts across a surface by manually applying each cut is a pain. On a CNC mill, of course, we can simply stand by and watch as the cutter repeatedly sweeps across the surface, descends, and repeats the cut as often as is required. Tea and biscuits in the spectators' gallery, please.

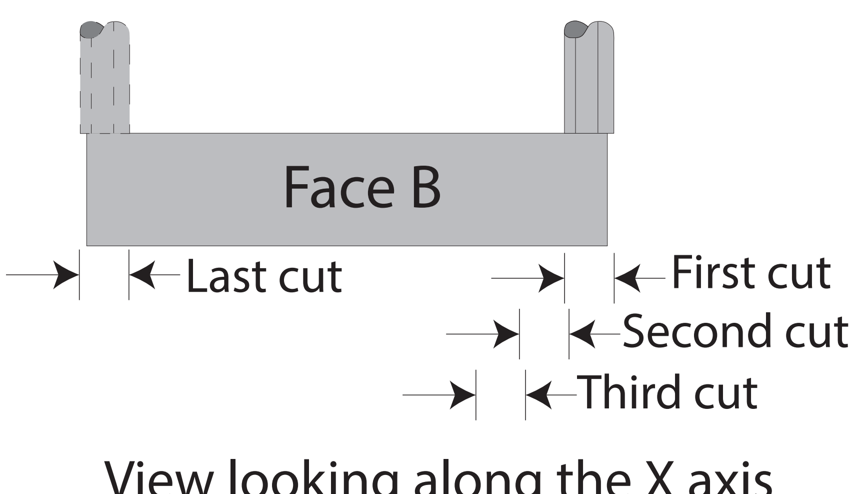


Fig 25: Surfacing, using overlapping parallel passes of an end mill..

The usual calculation for spindle speed and feed rate applies (explained in an earlier instalment), but increase the spindle speed by 30% for carbide teeth as opposed to high speed steel. I will assume the material is mild steel (080A15 being a commonly used grade).
The diameter of sweep of the 3-tooth cutter is 40mm, so with the carbide teeth the spindle speed should be 333rpm and with 3 teeth and a feed per tooth of, say, 0.1mm, the corresponding feed rate is 99mm/min.
The diameter of sweep of the flycutter is 30mm but it only has one HSS tooth so the spindle speed should be 333rpm and the feed rate should be 33mm/min.
On a 4-tooth 10mm end mill, the spindle speed should be 1000rpm. Experience suggests the chip load per tooth can't be as high on an end mill, because it projects much further from the spindle nose, and that around 0.025mm per chip is a good place to start. That gives us a feed rate of 100mm/min.

Cutting the material

The dimensions of the block are not at all critical, and the material can be any metal. I like working with plastic, but its not appropriate for this accessory because this stop needs to be rigid and have hard-wearing faces. I will refer to the sizes as shown in fig 24, but it's the method that's more important than the actual finished sizes. The dimensions of my block were suggested by the piece of material I had to hand. Yours will be too.
I cut the workpiece from a slightly larger length of steel, so it had two hand-cut ends. You may have to hack it from a larger piece, if that's what you have.
Ignoring the step in side C, for the moment, the basic method is to surface opposite sides so that we end up with a fully surfaced block, to bring the block accurately to size. Then create the step; and finally drill the attachment holes.

Machining the top of the stop

The side which will be the top surface of the stop is a good place to begin. I will assume that faces B and E are the cut ends of the workpiece. If you have cut the material in a different way, just do the best you can to follow my sequence. Grip the block in the chuck with the two largest faces (C and F) against the jaws and face "A" upwards. Photo 54 shows the position, illustrated by gripping the finished block rather than the raw material, to make the orientation more obvious.

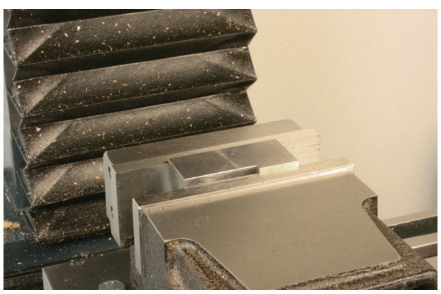


Photo 54: Grip the raw material by the largest faces, with side A uppermost.

Ideally, the block should be sitting with the top face "A" more or less horizontal, and protruding perhaps 3mm above the top of the jaws. This might mean sitting the block on a small parallel piece of material. I sometimes use a piece of unground high speed steel toolbit, but alternatives include small parallels, and a drill blank (a length of precision ground round high speed steel like a twist drill but without the flutes). It will also

Photo 55: Butt the end of the material against a square spanning the ends of the jaws..

be useful, for later, if the end of the block is roughly aligned with the left end of the vice jaws. This is not essential, and we can only be approximate anyway, if the end has been cut with a hacksaw, but place an engineer's square, or a flat piece of material across the ends of the jaws and just butt the material up against it (photo 55).
Mount the cutter in the spindle, then jog until the cutter is a few millimetres clear of the left end of the work. Zero the X axis DRO in this position.

Jogging is a useful action, but the behaviour of the jogging keys differs depending on which screen Mach3 is displaying.
At the Program Run screen, the jogging keys should function as expected. The jogging control display can be toggled out (visible) or in (hidden) using the TAB key.
If the computer enters screen saver mode the first jog key press to re-activate the display can be lost. This is not a problem, but it can give you a bit of a surprise when you press a jog key and nothing happens. Just press it again.
Continuous jogging takes place at the default G0 speed set in the Preferences, but modified by the Slow Jog Rate percentage shown between the – and + buttons. You may want to set the G0 move speed to a high value but have a little more control at times by reducing this to, say, half that speed for jogging. In that case, set the Slow Jog Rate to 50%. I use 1000mm/min most of the time, but you do need to keep your wits about you at that speed.
Step jogging moves take place at the current feed rate, as set by the last F command. To ensure a predictable step jogging feed rate, use the MDI screen to enter F100 or your choice of rate.

At the MDI screen, jogging is sometimes disabled because when you are typing a command it is legitimate to press the arrow keys to go up and down the list of previously typed commands, and there would be some confusion over whether the arrow keys should cause scrolling or a jogging move. So, when the MDI input window is active, jogging is automatically disabled.
Above the MDI command input window the Jog ON/OFF indicator is lit when jogging is available, and unlit when jogging is disabled. Pressing ESC greys out the MDI input and allows jogging.
Pressing RETURN puts the cursor in the MDI input window, making that the focus of typing, and disables jogging. Or you can just click in that MDI window to make it the focus.
Mach3 may also display the message "Jogging disabled for MDI input" in the message window in the lower part of the MDI screen. This looks like an error, but is simply factual information to serve as a reminder.
When jogging is allowed, TAB makes the jogging controls visible or invisible.
Flipping backwards and forwards between Program Run and MDI can be done using ALT-1 and ALT-2, but the sequence is more usable if TAB and RETURN are also used, like this:
From Program Run: ALT-2, RETURN switches to MDI and puts the cursor in the input window ready to type commands.
From MDI: ESC, ALT-1 switches to the Program Run screen.
The Jog ON/OFF button is also present on the Program Run screen, in the Tool Information section. If pressing the jog keys does not cause jogging, check the status of this ON/OFF button. Because it can be toggled using the CTRL-ALT-J key combination, it is just possible you have inadvertently toggled it off.
The keys used for jogging can be set via the Config > System Hotkeys menu. However, I would encourage you to use the defaults unless you have a need to change those. The default key codes are not shown in the initial setup of the System Hotkeys Setup table, which displays 999 in each entry box (999 meaning "the appropriate default code for the default key" in this case). Making,

or changing, a definition will put the ASCII code for the appropriate key into the chosen hotkey box (see page 5-19 of the Mach3 manual). Once you have made changes to any box, you may find it surprisingly difficult to return that box to a value of 999.

On the Jog Controller, repeatedly clicking the Cycle Jog Step button will cycle through a set of values for the step size. This is useful when Jog Mode is set to Step because you can use a coarse step size for rapid positioning, then increasingly finer steps as you approach a final position. In Config… > General Config… the "Jog Increments in Cycle Mode" section allows you to control the jog step sizes. Any changes you make in this table of entries will be reflected in the Step sizes shown in the jogging panel. Personally, I have not found the need to alter the default settings as they seem to make logical sense. In any case, back at the Jog Controller you can click on the Cycle Jog Step number and type your own value there, so you do have an immediate way of altering step size to some other value to suit your purpose. There is no harm in permanently altering the table of step sizes, as they are easily restored to their initial values. Be aware, though, that entering a value of 999 anywhere in the list forces the Jog Mode to change to Cont(inuous) when that step size is selected. That can be useful, although it is somewhat limited by the fact that the Cycle Jog Step button cannot subsequently force the mode back to Step.

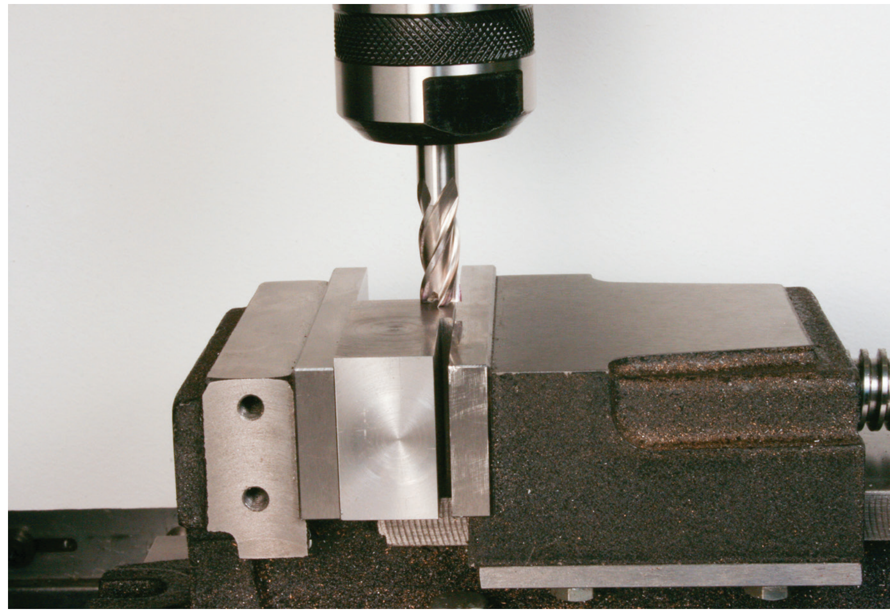


Photo 56: Initial Y position of a small diameter cutter for the first of a series of overlapping cuts.

With a large diameter cutter, jog along the Y axis to position the cutter so that one tooth just sweeps right over one edge as the whole cutter spans the work, then zero the Y axis DRO. This position may mean the cutter is not centrally located over the work, and that's the intention. Centring the cutter over the work will get the job done, but its good practice to avoid a perfectly symmetrical coverage to minimise the possibility of chatter. It's perhaps counter-intuitive, but having the teeth cutting symmetrically across the surface may result in a greater tendency for the cutter to vibrate and chatter. I can't imagine we will be pushing the cutter quite that hard, on this job, but this slightly off-centre setup is good practice. Incidentally, this is a reason for having an odd number of teeth on a cutter of this sort. It helps avoid the kind of balanced cutting action that would result from diagonally opposite teeth. Reamers have an odd number of cutting edges for exactly the same reason.

With a small diameter cutter, prepare for the first of a set of parallel cuts by positioning the cutter nearest the front of the machine with almost the whole diameter over the surface of the work (photo 56 and fig 25) then zero the Y axis DRO. As with the larger cutter, avoid the situation where the cutting forces are completely balanced. If you have the luxury of choice, choose an end mill with 3 teeth (an odd number) rather than 2 or 4 teeth (because even numbers places pairs of teeth opposite one another).