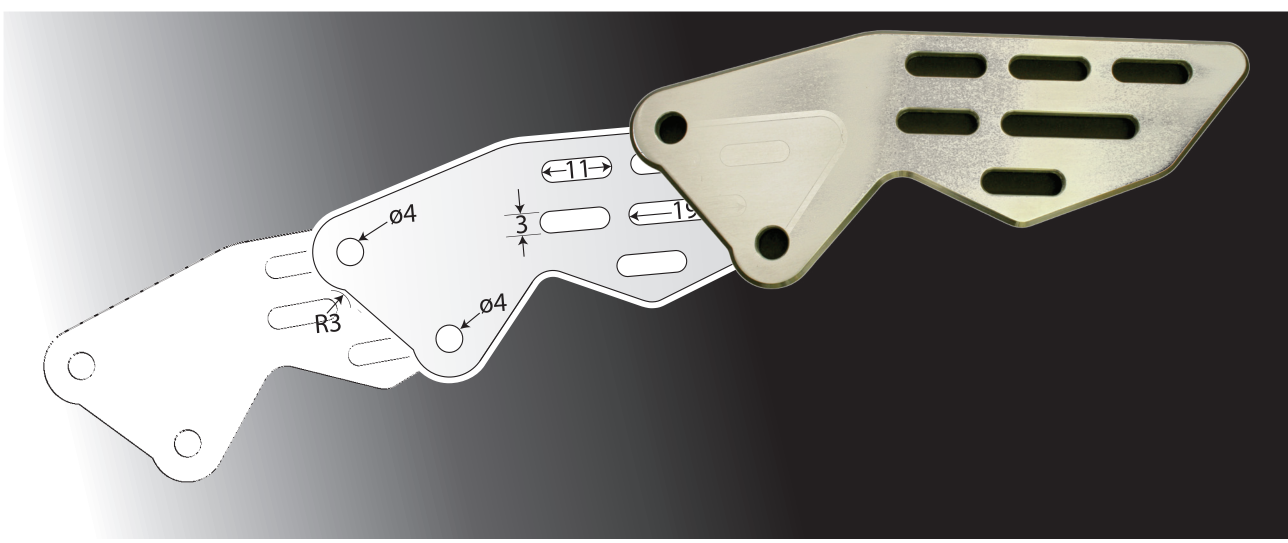
CNC in the Workshop



Part 8

Part 8

In this part of the series, we complete the Vice Stop

There is seldom just one way of taking a set of cuts, especially where the cutter must take several parallel passes to cut across the whole of a face, and you may prefer to take the first cut then move around the outside of the work, keeping the cutter clear of the sides and any other obstacles, before moving into position to take a second parallel pass. This avoids altering the Z height, with any attendant tiny changes in the height at each pass. There shouldn't be a difference, of course, but if you have not quite ironed out the Z backlash, this method will work around that. Take care, though, that the cutter will move in free air, clear of all obstacles. Take care, too, that you have enough Y axis travel on your machine. Fig 31 shows a possible path, in plan view, and a possible program segment might be:

```
G0 Z20
G0 X0 Y0 (position for first cut)
G0 Z-0.25
G1 X100
G0 Y100 (to clear the back of the vice)
G0 X0
G0 Y9 (position for next parallel cut)
```

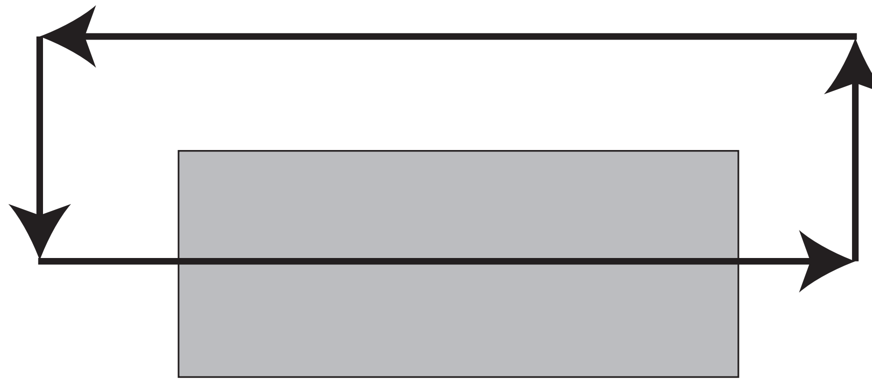


Fig 31: An alternative path for the cut..

Machining the other faces

If the side we have just machined is "A" (fig 24) turning the workpiece over will allow the opposite side (D) to be machined by simply running the same program again. If you took the time to position the left face of work in line with the left end of the jaws, before the first cut, do that again now. That way, your X0 position will still be in the same place relative to the work, so you will not need to work out the length of the cut; it will be the same as before. Because the vice and the work are in the same position relative to the Y axis, the Y moves will also be the same as before. The only difference will be in the Z height, because the workpiece is less tall, as a result of the previous cuts. There are two solutions to this. The first is to reset Z to zero using the same method as before. The second is an act of faith. Take the cutter clear of the work, move to the last Z cut position you used (say Z-0.6), click in the Z DRO and enter 0. If there are minor discrepancies, these will not matter on this job. In practice, though, this second method should work just fine. It is the basis of some useful techniques with a probe, where the position of one physical surface is used as the basis for working out the position of another.

Measure the thickness of the work, top to bottom. Knowing the intended final thickness (35mm) you can subtract the actual from the intended thickness to find how much needs to be machined from the upper surface. Edit your program so that you carry out enough passes to machine that amount off. Ideally, the last pass should be a small finishing cut. Save your program, perhaps under a new name, then load and run your program. You will check that Mach3 is using the updated program, before you actually run it, won't you?

To machine the end face B, stand the work vertically in the vice with B uppermost. Use a set square against the bottom surface inside the vice and either face A or D to set the work vertical (photo 60). If that's not convenient, you can reference off the top of a vice jaw (photo 61). Both of these assume the vice base or the top of the vice jaw is parallel to the bed of the milling machine. If in doubt; check. Then modify your program to face the top surface (B). It hardly seems worthwhile changing the position of X0 or Y0, because we still have another large face to surface.

Turn the work so that E is uppermost, and

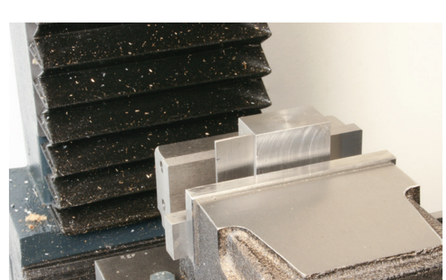


Photo 60:Using a square to set an edge vertical by referencing off the inside bottom surface of the vice.

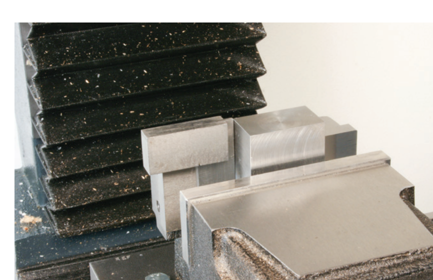


Photo 61:Using a square to set an edge vertical by referencing off the top of a vice jaw.

repeat. Photo 60 shows the work gripped more or less centrally in the vice jaws, for security. To position face E in roughly the same place as B, you can place a ruler along the top of a jaw as shown in photo 62, where the ruler is secured by two small magnets.
Once you have cleaned up face E, measure the height of the workpiece and calculate how much more needs to come off to bring the work to a length (or height, in this orientation) of 53mm as shown in fig 24, or whatever length suits your own vice. Then edit your program so that it makes the passes required to bring the work to size. Save it using a different name, then go back into Mach3, load that new version and run it to make the cuts.

Grip the workpiece in the chuck with face F uppermost, to allow you to surface that face. You will need to set Z to 0 using a roller or cigarette paper. Edit your program to ensure the appropriate number of parallel passes at each depth of cut. Machine just enough to clean up the surface, finishing with a fine cut.

Turn the work over to place face C uppermost, and make sure there is a maximum of 20mm down between the jaws, to ensure that you have enough above the jaws to allow for bringing the work to size, and for the step to be cut down some 3mm further. Make sure, too, that the left end of the work is aligned with the left end of the vice jaws. Ignoring the step for the moment, measure the thickness and work out how much needs to come off to bring the work to the fullest size (25mm as shown in fig 24 i.e. 22mm + the 3mm step). Machine it to that size.

Creating the step in the inner face

The inner face has a step 34 x 3mm (fig 24), These dimensions should be varied to suit your own vice, the intention being that the step is long enough to deal with most workpieces, and deep enough to clear the end of the vice jaws when the stop is secured in

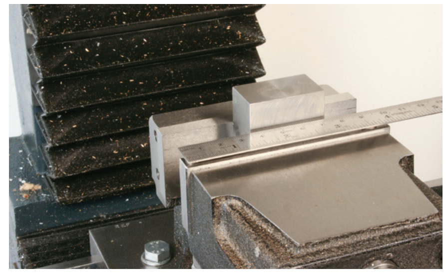


Photo 62:Using a ruler to position the work.

position, leaving a little clearance for burrs.
To create the step accurately to size we need to be able to locate X0 and Z0 accurately in relation to the edges of the workpiece. It is not essential for this job, but is wise, to establish Y0 accurately too.
This is a basic requirement for most workpieces and most jobs, and although it was not necessary to be terribly accurate when machining the faces, we did still set X and Z to 0 at suitable positions in relation to the work. Bearing in mind the orientation of the axes, it is most convenient to set X0 Y0 and Z0, which we will call the Work Origin, at the front left corner of the top face of the work, as shown in photo 63.
Let's just mention Home too, at this stage.
Home is a known location in relation to the physical machine. It is not related to the position of the workpiece. Home is, if you like, the origin (0, 0, 0) of the machine's co-ordinate system. It may be defined by Home switches, or not; but when Mach3 starts up it makes one position X0, Y0, Z0. This is called absolute machine zero and it is not necessarily shown on the DRO display on the Program Run screen. To see the machine coordinates, click the Machine Coords button. Click again to go back to the work co-ordinates. Clicking back and forth between the two, it is likely that they will be different. The difference is shown on the Offsets screen. For most practical purposes, machine co-ordinates are of little interest because they simply tell us where the Controlled Point is in relation to Home. It's the work co-ordinates we want, because those tell us where we are in relation to our chosen reference point

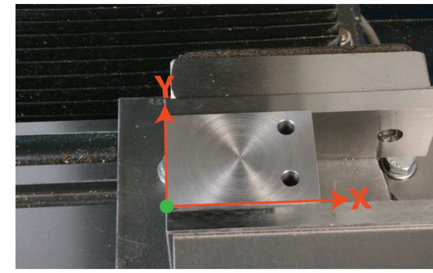


Photo 63:The green dot shows the work origin at the front left corner of the top face of the workpiece.

on the workpiece. Home does have its uses, of course, as do the machine co-ordinates, especially for automated tool changing and for automated Z height setting, but we will leave those for now because we need to master some other techniques first. On the Program Run screen, the Ref All button next to the DROs is designed to make Mach3 move to the Home switch on each axis and set absolute machine zero in that position. If you don't have Home switches, or have not set up Mach3 to perform a Home sequence, it will not do this. If you have no Home switches or Home sequence, Mach3 will pick a position for Home. We won't mind.

To set the Work Origin, position the Controlled Point where you want the Work Origin to be, and set the DROs to zero by clicking the Zero X, Zero Y and Zero Z buttons.
The question then is: how do we set X0, Y0 and Z0 accurately? Z0 can be set sufficiently accurately for the moment by using the paper or roller methods already outlined.
X and Y axis positioning is rather different, just because the axes move horizontally instead of vertically.
techniques include:
touching the side of the work with the side of the cutter;
using a touch point sensor;
using an optical alignment aid.

Touching with the cutter

Attach a piece of cigarette paper to the left end of the workpiece and start the spindle turning slowly, then gently jog in X until the cutter touches the paper. Set the X DRO to – half the diameter of the cutter. Photo 64 and fig 32 show that just as the cutter touches the paper the CP is half the cutter diameter away from the work, but it is to the left of where we want X to be 0 so it has a negative value i.e. for a 10mm diameter cutter, enter -5. It is possible to do this operation without

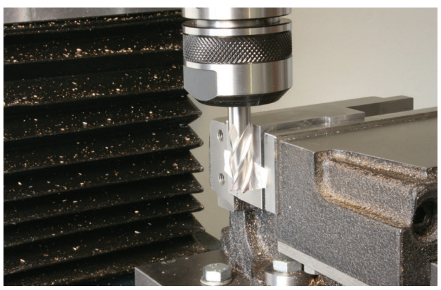


Photo 64: A cutter is half its diameter away from the work when it touches the paper.

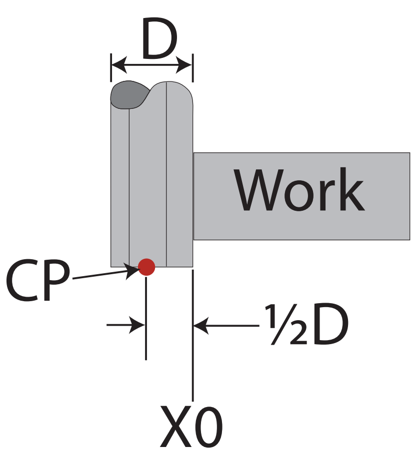


Fig 32: Position of the CP along the X axis as the cutter touches the paper.

starting the spindle. Instead, just gently turn the cutter backwards and forwards by hand, gripping the chuck rather than the sharp edges of the cutter. It's easier on the nerves, and sometimes easier on the workpiece too. Attach a piece of cigarette paper to the front of the workpiece, where it projects above the vice jaw and start the spindle turning slowly. Take the cutter round to the front of the workpiece then gently jog in Y until the cutter touches the paper. Set the Y DRO to – half the diameter of the cutter. Fig 33 shows that just as the cutter touches the paper the CP is half the cutter diameter away from the work, but it is to the front of where we want Y to be 0 so it has a negative value.

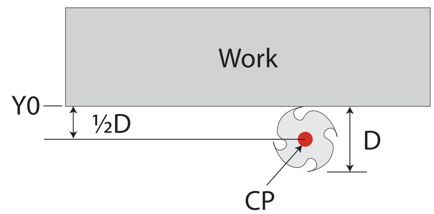


Fig 33: Position of the CP along the Y axis as the cutter touches the paper.

Using a touch point sensor

An illuminating touch sensor is a form of probe and it can be used in much the same way as a piece of paper, but with the spindle stopped, jogging along X or Y until the light in the sensor comes on. At that point, the sensor is half its end diameter away from the work (photo 65 and fig 34). Probes are

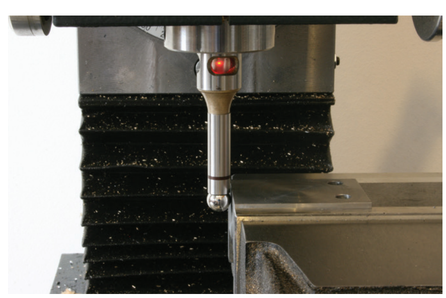


Photo 65: A probe lights when it touches the work.

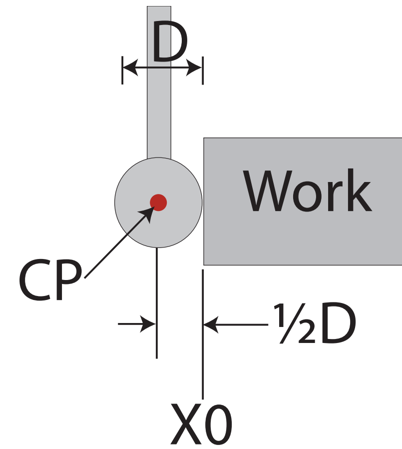


Fig 34: A touch point sesnor indicates when the CP is half the diameter away from the workpiece.

a common way of setting a Work Origin, so Mach3 makes provision for this method, on its Offsets screen. A little care is required to use this facility, but it is a handy way of quickly setting the Work Origin. At the bottom left of the Offsets screen there is a section in which you can enter the diameter of the edge finder, probe or sensor (fig 35). Then, when you move the sensor in, say, the X direction, click the lower left SELECT button next to the left edge of the work; and when you move the sensor in the Y direction, towards the front edge of the work, click the SELECT button at the lower right. Take care here, and note the orientation of the axes as shown on the diagram. It assumes the Work Origin is at the front left corner, which it will be for most work, but the SELECT buttons

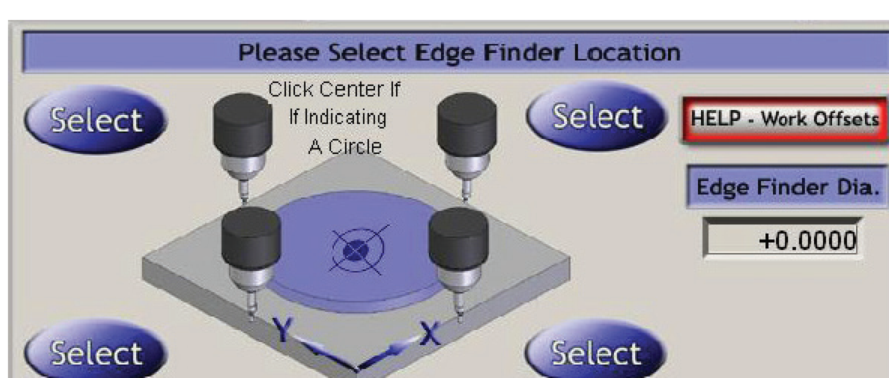


Fig 35: Probe settings on the Offsets screen.

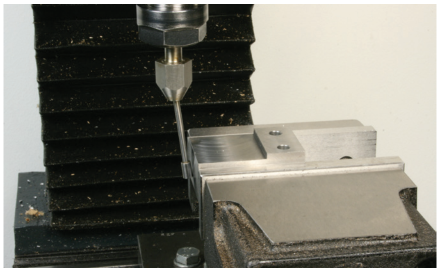


Photo 66: A conventional wiggler detects the position of the face of the workpiece.

refer to touching the sides as shown, so the lower left SELECT button applies to the left edge and sets the X axis, while the lower right SELECT button refers to the front edge and sets the Y axis.

This method can be used with a conventional wiggler (photo 66) instead of an illuminating sensor. With the spindle turning slowly, jog the wiggler until it touches the workpiece, then continue jogging in small steps until the wiggler's arm runs true. Continue jogging very slowly and in small steps. When the wiggler arm flicks suddenly, and tries to run along the face of the work, that's the indication the CP is half the diameter away from the edge. You may scoff at the old fashioned low-tech wiggler, but that fancy illuminating touch point sensor only works when the workpiece conducts electricity (and when there is still some charge in the probe's batteries). Try setting the work origin on a block of plastic if you want to appreciate the conventional wiggler.

Photo 67: A spotting scope allows you to see the edges of the workpiece.

Sighting the Origin

Optical aids are a great help when setting the Work Origin, and the machine scope (photo 67) acts like a microscope, allowing you to see the edges. Mount the scope in the spindle, either directly or in a chuck. Sight through the eyepiece and watch for the crosshairs to align with an edge. Set 0 at that point. Easy; and accurate too. My favourite method by a long way. The benefit of this method is that you can use the scope to sight any feature on the workpiece, like the edge of a projection somewhere within the boundary of the plate, or a hole, or the point where two scribed lines cross.

A variation on this method is the Laser Centre and Edge Finder (photo 68). There are

Photo 68: A laser centre and edge finder.

arguments over the accuracy of this tool because of the physical size of the spot, but the current models use polarising filters to produce a tiny spot, and the device is very easy to use. Line up an edge by splitting half the light from the spot down the side of the edge (photo 69), and that's the zero point. Like the scope, the spot can be aligned with a feature within the boundaries of the workpiece such as a centre dot or scribed lines.

Work out the position of the edge

Once the Work Origin is set, that indicates the position of the CP, which is normally aligned with the centreline of the cutter. The co-ordinates of the edge of the step will be 33mm to the right, less half the diameter of the cutter (fig 36). For a 40mm cutter, we need to move the CP to X13 to align the right hand edge of the cutter with the edge of the step.

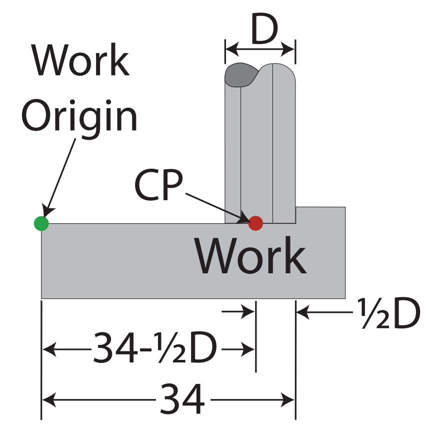


Fig 36: Calculating the position of the CP when creating the step.

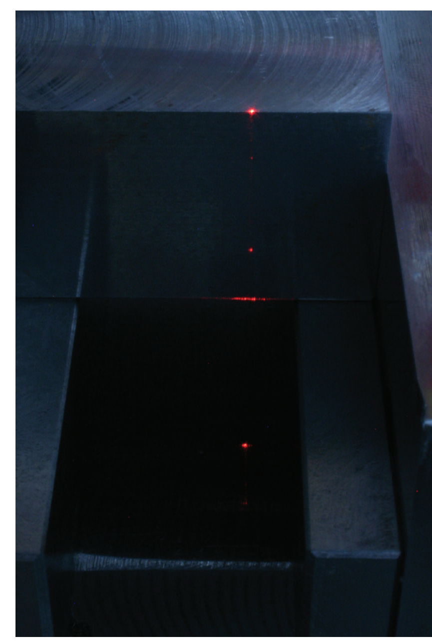


Photo 69: Splitting the dot on the edge sheds light down the edge and onto the vice base.

Cut the step

Create a program (or modify an existing program) to:

move the cutter above the work, and clear of the rear of the work, to, say, X13 Y21,

lower the cutter to Z0, then
take several cuts to remove material to a depth of 3mm to create the step,
retract the cutter clear of the work.

You may need to take more than one pass at each height, to sweep the whole of the step, and your movements will be mainly in Y this time, so that the cutter creates the straight face of the edge of the step.
Note that cutting is best done with the cutter passing from the rear of the work to the front, as it passes the edge of the step. This would be the appropriate direction for conventional milling.

Drill the holes

The Work Origin at the front left allows the co-ordinates of the holes to be calculated as X43.5, Y6 and X43.5, Y 27. This assumes the holes are positioned half way across the width of the protruding part of face C.
Use a centre drill, then a pilot of perhaps 3mm and a finishing size of 6.2mm to produce the holes. Finish with a light countersink. MDI mode is good enough for this, and your choice is to move repeatedly from one hole to the other, for each size of drill.

Attach the stop to the vice

Ideally, you should clean up the side of the vice. This may simply require the removal of the paint with a scraper or a file. If you wish to take a cut across the side, to ensure a flat face, one way is to remove the rear jaw then support the vice, with its plate, lying on its side against an angle plate (photo 70). The vice side needs to be cleaned up at right angles to the vice jaw seatings, so use a square to set the rear jaw seat vertical. This is a really awkward setup, so note the additional jacks under the sides of the vice; one under the casting for the moving rear jaw and another further along under the fixed jaw casting. These are simple tee nuts with short bolts to allow support under

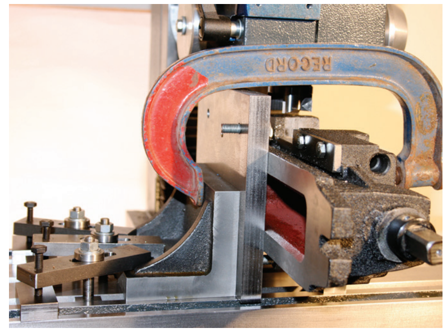


Photo 70: Support the vice on its side against an angle plate.

the cut (photo 71). Having two jacks allows controlled adjustment to bring the jaw seat

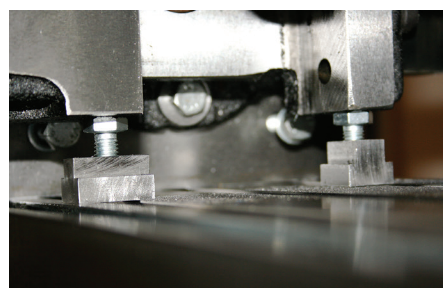


Photo 71: Support the vice casting using jacks.

vertical.
Take a light cut across the side of the vice, where the stop will be fastened. Then drill the holes and manually tap the threads. The hole positions are indicated in fig 37 but these should be checked with reference to the stop, and you may wish to drill and manually tap one hole, bolt the stop in place, and check the position of the second hole before drilling and manually tapping that hole too. There would be nothing

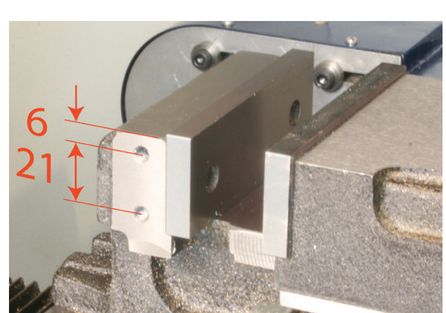


Fig 37: The hole positions.

wrong with drilling the holes in the drilling machine. An appropriate cutting speed for cast iron is approximately 40M/min and a 5mm drill (tapping size for M6) should revolve at around 2700rpm.
An alternative approach is to sit the vice on a substantial block, right way up, and tilt the head of the mill at 90o so that the spindle lies parallel to the table. Depth of cut is then applied using the X axis. If you are going to use this (equally awkward) setup, take the opportunity to skim the rear face of the vice casting (parallel to the rear jaw) because there are some interesting accessories which can be made to fit this face. You will need some tall tee bolts for this setup. You will also want to spend a happy hour or two realigning the mill head so that the spindle is vertical, once the job is over.
Or you could just clean up the side and end faces with a scraper and/or a file, because hand methods are sometimes the easiest choice. I machined mine with the mill, just because of the challenge.

Add a flourish

Before you tick the job off as complete, let's just add a little touch to impress our friends by machining some lettering in the outside face of the stop. Photo 72 shows the kind of

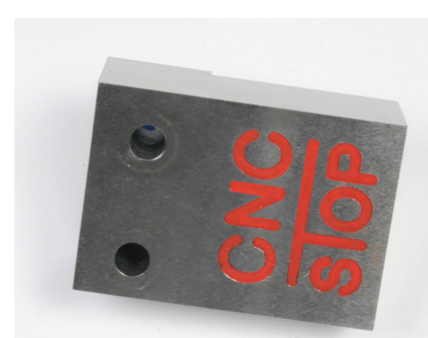


Photo 72: Lettering cut in the side of the stop.

thing. To create the lettering you will need a program which can be downloaded from the support website, and a small cutter. The ideal cutter is a centre cutting 2mm end mill with 3 flutes, but 5/64 inch would do; and a

2 flute slot drill or a 4 flute end mill would also work just fine provided they are centre cutting. The program expects to create the lettering in the position shown in fig 38 and you should note that the Work Origin needs

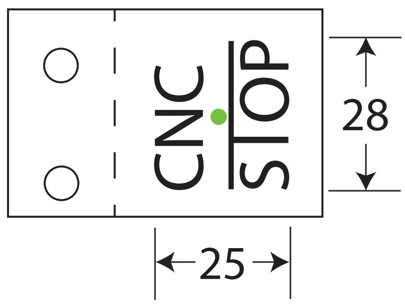


Fig 38: The position of the lettering.

to be set to the centre of the rectangular area within which the lettering will be cut. The lettering will be cut in the orientation shown i.e. rotated 90 degrees anti-clockwise, lying on its side. The program expects a spindle speed of 5000rpm but will work at a slower speed. Just set the spindle speed to the maximum your mill can achieve. Cutting takes place at a feed rate of 50mm/min. This is much slower than theoretically possible, for two reasons: firstly, slowing the spindle speed requires that the feed rate be dropped proportionately; and secondly, small diameter cutters are fragile, so it is wise to go much more slowly than theory indicates. There will be a tendency for the cutter to rub, but that's still better than hearing it snap.

Load the program into Mach3 and check the toolpath shows the lettering. Look at the comments at the start of the program. Then set the work origin. You could stick a label in the centre of the area to be lettered, mark the position of the Work Origin with a fine pencil cross and use a pointer in the chuck, or a spotting scope, or a laser to move to that position and set the DROs to X0, Y0. Or you could use a touch point sensor or a wiggler to find the sides then use MDI commands to move the CP to the centre of the lettering area and set X0, Y0 there. Set Z0 at the top surface of the material. That gives you your Work Origin. Set your spindle speed manually, or let the program set it automatically, then run the program.

Use copious lubrication. For steel, apply neat cutting oil by pump or toothbrush.

Once the lettering has been cut, degrease the workpiece carefully, then add a little paint, in your favourite colour, to the grooves. Wipe off any excess and let the paint dry, before admiring it in a flattering light.

Nice job.