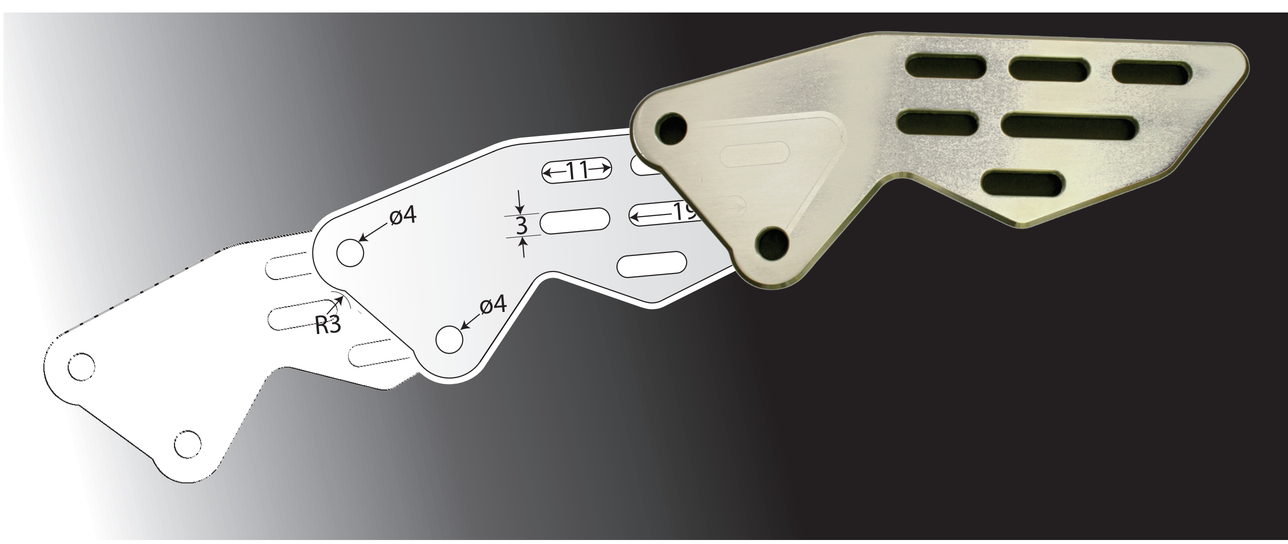
CNC in the Workshop



Part 5

Part 5

In this part of the series, we continue work on the Vice Plate.

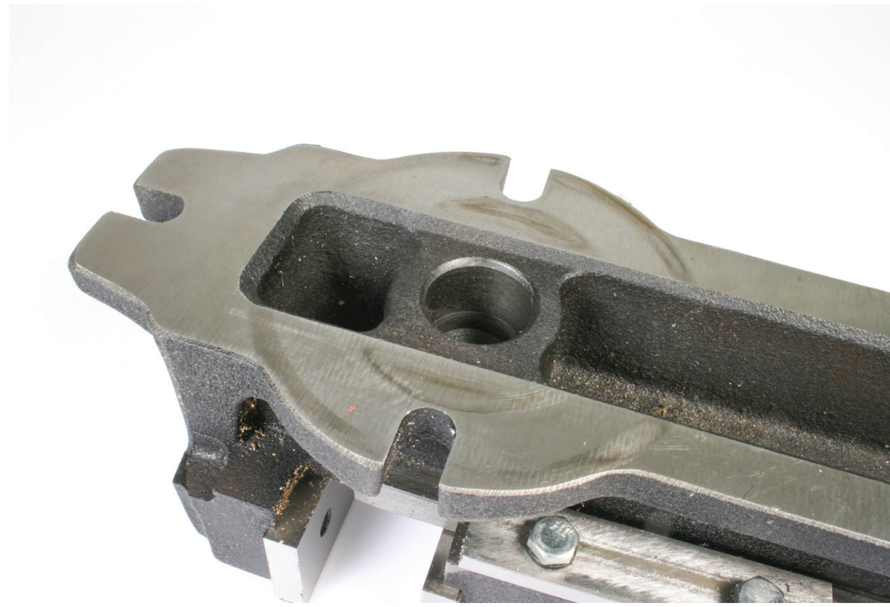


Photo 31: The underside of the Vertex K4 vice has a 25mm diameter hole to allow it to pivot on a boss.

The following instructions apply specifically to the popular Vertex K4 vice, but can easily be adapted to most other makes and models of vice. The Vertex has a rotating base with a large projecting shaft, which locates in a large hole in the underside of the vice (photo 31). If your vice does not have that feature, you will still be able to use a vice plate with your own vice, it's just a little bit more tricky to set up on the plate the one and only time this will need to be done.

The sizes I used are shown in fig 21, but these can be altered to suit your own mill and vice. The 152mm dimension is the result of buying steel 6 inches wide. The 165mm more or less accommodate the main part of the base and the front lug. The essentials are that the horizontal and vertical spacing of the 4 outer holes is the same as the distance between the centres of the T slots in your mill table. Mine are at 100mm centres, so that they match the slots when the plate is laid across the table fore-and-aft, or at right angles to that. That allows the plate to line up with the vice jaws parallel to the X axis or parallel to the Y axis. With the vice jaws parallel to the X axis and the jaws open fairly wide, the centre of the opening will correspond approximately with the centre T slot. In fact, the lugs on the bottom of the vice casting will largely determine the positioning, because with 100mm centres the casting will foul the bolts which secure the plate to the table if the vice is much further forward or back from this position. Experiment with your own vice, but make sure that the Y axis travel allows a cutter to reach the front and rear of a typical decent-sized job.

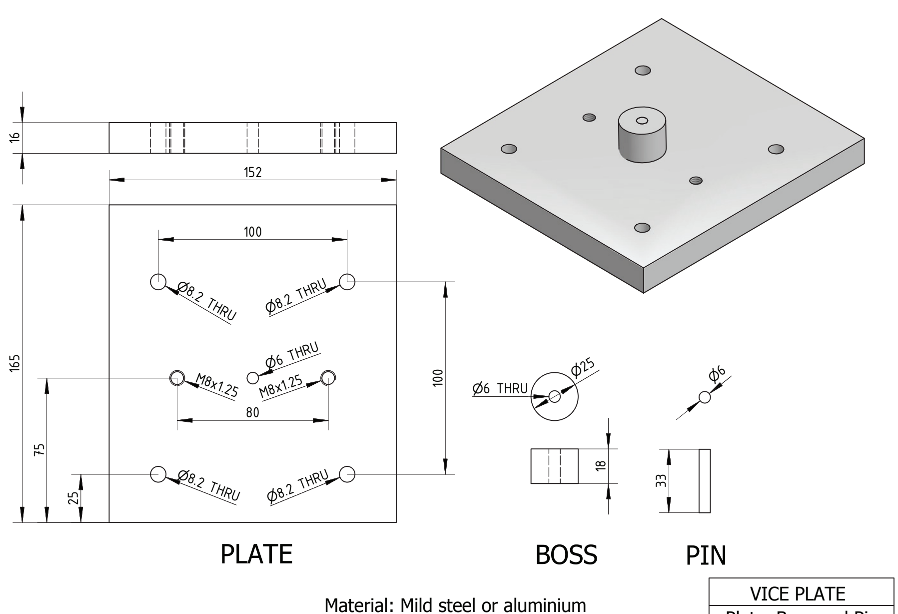


Fig 21: Dimensions of the Vice Plate.

Cut a square piece of thick steel into a rectangle or square large enough to span the working surface of the table from front to rear. Leave a little additional material on the length and breadth (say 1mm on each). I chose 16mm steel because the plate needs to be sufficiently thick to take two tapped holes for bolts to secure the vice to the plate. Sawing that thickness of plate certainly works the biceps, even if its only to push the On-Off switch on the bandsaw. Clean up the top and bottom surfaces with a file, or scare the living daylights out of it with a surface grinder (grinding a little from both faces to avoid distortion). If you want a deluxe-beyond-belief approach, use some ground flat aluminium tool plate but be prepared to re-mortgage the workshop first.

To be able to machine the edges and drill the holes in the plate we need to give some thought to how to position the plate and the clamps on the table. From front to back, the plate will be approximately the same size as the table. With a cutter in the chuck, and moving the Controlled Point (CP) almost as far back as it will go (i.e. a positive Y movement of the CP), the cutter is unlikely to be able to reach the back edge of the plate. You can test this by jogging a cutter almost as far back as it would go in the Y direction (moving the CP in the positive Y direction), lowering it, then checking where it is in relation to the rear edge of the table. In most cases it will not reach the rear edge.

Because it is not easy to set the rough rear edge of the material parallel to the X axis, we will use the left edge of the material as a temporary reference edge and set it squarely across the table. I chose to place the as-manufactured edge of the material parallel to the Y axis, so that when the rough end was machined parallel to the X axis it would be at right angles to the left edge of the material. Use as much packing under the plate as you can find, because we will be drilling down through the plate. I used some 18mm MDF, which is flat and of uniform thickness. You could use a series of spacers provided they are all the same thickness, but they will need to be placed in positions where they are not under a hole position, yet directly under suitable clamp positions, all of which is a bit bothersome. I keep some engine tappet pads handy for this kind of job (photo 32), and they are in frequent use. I also use some blocks of mild steel, cut to a handy size. Those find all sorts of uses in the workshop, and you can finish them nicely, if you are that way inclined. Mine are off-saw, but have been deburred with a fine file.

Where the material is on thick packing, an ordinary square can't be used to set the edge of the material in position, but you can set an angle plate or a lump of material with a flat face square across the table (photo 33) and butt the plate up against that, sliding the plate close to the cutter in its rear position (photo 34). You can remove the angle plate or block once the steel plate has at least one clamp in place to prevent it from moving. Two would be even better. Arrange the clamps so that they do not prevent a cutter from travelling along the rear face of the material (photo 35).

Clamps should be substantial; clamping studs should be a decent size; and you should use proper T-nuts which fit the slots in your mill table snugly (photo 36). Make sure the studs cannot pass through the bottom of the T-nuts. Where a stud can be screwed through the bottom of a T-nut, it can exert considerable upwards force, jacking the T-nut against the underside of the T-slot with the real prospect of considerable damage to the slot, including distortion of the top surface of the table, or shearing a lump out of the T-slot.

Photo 32: Engine tappet pads for use as spacers.

Machining an edge

Put an end mill in the milling cutter chuck. Use a 12mm or ½" end mill if you have one. If not, use the largest you do have, and we will compensate for any differences in diameter, as we go along. Mill chucks vary in design, efficiency and cost. The most common are probably the plain end mill holders (photo 37), the Pozilock-style collet chucks (photos 38 and 39), and the ER collet chucks (photos 40 and 41). Any of those will do, for this job.

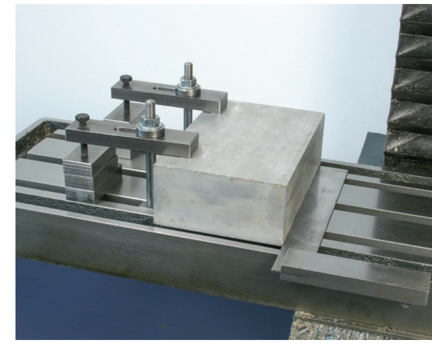


Photo 33: Setting a flat-faced block parallel to the the Y axis by squaring off the front of the table.

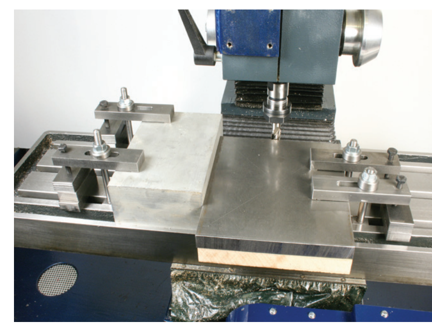


Photo 34: Butting a plate against a flat faced block previously set parallel to the Y axis..

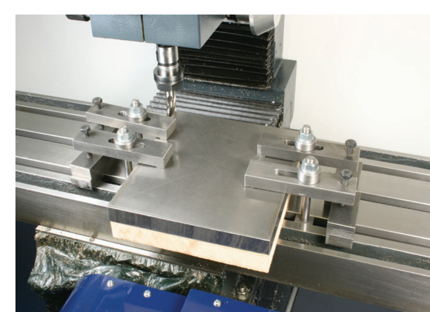


Photo 35: Clamps in positions where they will not obstruct the cutter machining the rear face..

Photo 36: Clamps studs and T-nuts..

Photo 37: Plain end mill holder..

Photo 38: Pozilock collet chuck..

Photo 39: Component parts of the Pozilock chuck.

Photo 40: ER collet chuck..

Photo 41: Component parts of the ER collet chuck.

The plain holders use a grub screw to secure just one size of shank, and are available with 6, 10, 12 or 16mm holes which correspond nicely to the common end mill shank sizes.

The Pozilock collet chucks have a set of (usually) 4 collets to grip shafts of 6, 10, 12 or 16mm diameter (or imperial equivalents) which have a thread at the end. Both metric and imperial threaded-shank milling cutters all have a 20TPI Whitworth thread and a dimple in the end face of the shank. The dimple and the threaded portion are an essential part of the way in which the chuck operates.

Selecting a collet which corresponds to the diameter of the cutter shank, screw the shank part way into the collet; insert the collet in the chuck; rotate it to make sure it is fully seated; and screw the nose nut onto the body of the chuck. Leave just a little slack on the nose nut, then rotate the cutter shank to take up the play. Put a spanner on the flats at the rear of the chuck body (up next to the tapered shank in the mill taper) then use the large collet nut spanner to fully tighten the nose nut. Tightening the nose nut forces the collet and cutter shank up against a conical pin which locates in the dimple in the end of the end mill shank, stopping it from moving backwards. The nose nut then closes the collet around the shank, gripping it securely.

The ER collets do not require a thread on the cutter shank. Slip the collet into the nut by holding it at a slight angle then pushing and twisting. The nut has an elliptical flange just behind the front face, and the collet will click in and be held there, loosely. Assemble the collet-and-nut onto the body; put the cutter shank into the collet; then tighten the nut. The collet nose nut simply closes the collet to grip the shank by squeezing the rear taper of the collet into the body of the holder, and the front taper against the inside of the nut. The slits in the collet allow it to squeeze inwards, gripping the shank. Note that the shank should be far enough into the collet so that it is gripped by both the front and rear of the collet as it closes.

Do NOT use a drill chuck to hold cutter shanks; the cutters can work loose under heavy loads and/or vibration. Drill chucks are excellent for holding drills and reamers, but are not suitable for anything else.

Speaking of end mills, photo 42 shows that not all end mills are created equal. The end mill on the left is a conventional cutter with 4 flutes and spiral cutting edges. Good end mills are expensive, and are available in a range of materials (High Speed Steel, or solid carbide are the usual materials) with optional coatings. Coatings improve the cutting performance of the end mill, although whether this is noticeable under anything less than industrial conditions is debatable. The more important aspect is whether the end mill has a sharp cutting edge and the length of time for which it will remain sharp. In the photograph, the middle cutter is a Dormer XS1 and despite use over a period of time it remains sharp enough to draw blood from a loving caress.

Photo 42: Roughing and finishing end mills..

The right hand cutter is a roughing end mill, designed to remove a greater quantity of material quickly and exerting less force as it does so, because of the interrupted cutting edges. This cutter will not give a fine finish, but it is surprisingly good. The cutting edges of this cutter have remained sharp despite being abused to the point of turning an attractive shade of blue under the heat generated from over enthusiastic use. Like the finishing cutter, this was an expensive investment, but one I would repeat. This

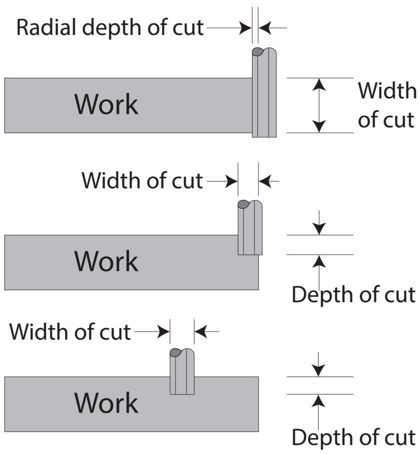


Fig 22: Terminology used to refer to end mill cuts..

kind of cutter is sometimes called a "rougher" or a "ripper".

Neither of these end mills are end cutting, so they cannot be plunged straight down into a workpiece. They do have teeth on the end, but the centre portion has no teeth and cannot cut. These end mills are useful for side cutting, cutting a slot or a step (fig 22). To cut directly down into a workpiece, an end mill needs at least one over-centre tooth like the right hand cutter in photo 42 so that the whole of the area beneath the end of the cutter is swept by a cutting edge. All things being equal, choose an end cutting end mill if you are buying a new one.

The number of cutting edges on an end mill depends to some extent on its diameter, with larger cutters having typically 4 teeth and smaller cutters having perhaps 3. Cutters with 2 cutting edges are usually termed slot drills and these normally have 2 cutting edges on the bottom, with most being end cutting. The term "slot" describes the use of the cutter for milling a slot (although you can do that with a 3 or 4 flute end mill too), and the term "drill" may be because this kind of cutter can cut vertically into the work, like as drill; or perhaps because it has the same number of cutting edges as a drill). For our purposes, the difference between a slot drill and an end mill is largely academic. I have often read that a slot drill will cut a slot with a more accurate width than a 4 fluted end mill, but I have seen no evidence of that.

Put your end mill into the collet chuck and tighten to hold it securely.

I will assume a 12mm diameter cutter, but it won't matter.

Bearing in mind the need to avoid obstructions, jog to take the cutter close to, but just beyond, the farthest edge of the plate, with the cutter spanning the whole depth of the edge, as shown in the top illustration in fig 22 and photo 34. Jog the cutter just clear of the left edge of the plate (photo 35), then zero the X and Y and DROs in Mach3. For this kind of job it is usual to set the Z axis to 0 where the bottom of the end mill is approximately level with the top surface of the plate. That's not essential for this job, but let's do it anyway. Jog the cutter upwards until the end of the cutter is approximately level with the top surface of the plate; then zero the Z axis DRO.

Appropriate speeds and feeds for a 12mm diameter end mill might be approximately:

Spindle speed 800rpm

Feed rate 80mm/min (for a chip load of 0.025mm)

Assuming the cutter rotates clockwise (as viewed from above, looking down the axis of the spindle) then for conventional milling the cutting edges of the end mill should scoop into the work as it passes from right to left (fig 18). The chip of metal removed by each tooth starts thin, increases as the tooth continues to rotate and move forward, and finishes at maximum thickness (the chip load). In climb milling (fig 19), the chip starts at maximum thickness and reduces as the cutter turns and moves. The chip load in that case is the thickness at the start. We will use conventional milling for this job. More about climb milling later.

We will be cutting across the full depth of the face, so there are two important factors to consider. The first is that, generally speaking, for an end mill used as a profiling cutter (cutting on the side rather than on the end teeth) the maximum width of the face should be no more than 3 times the diameter of the end mill. 12mm diameter x 3 = 36mm width of cut, so we are well within that limit. The tendency to chatter increases with the width of cut, though, so that will affect the finish, and the speeds we can use. Theoretically, we could use a cutter of 16/3 = 5.3mm diameter (because 3 x 5.3 diameter = 16mm width of cut), but the rigidity of the tool drops off rapidly as the diameter is reduced, and that makes chatter more likely. The thinner tool will also be more prone to bending.

End mills can cut in different ways, using either or both the side cutting edges and/or the end teeth. In this configuration, where the end teeth are not involved, what's important is the *radial* depth of cut (fig 20), so called because to put on the cut the cutter is fed radially into the work. In our situation, we will move the cutter radially *before* it meets the work.

Depth of cut 0.2mm per pass.

Jog the cutter just beyond the right end of the edge (photo 43) and note the reading on the DRO. Assuming the width of the plate is 152mm, the DRO might read X170 Y0.

Photo 43: End mill just clear of the right hand end of the rear face of the workpiece..

At the MDI screen, lower the cutter using G0 Z-18 so that the end of the cutter is just below the bottom surface of the plate and spans the face to be cut. We will trim the face by repeating the following sequence (shown in fig 23) until the face has been cleaned up. Starting at position 1:

apply a cut by moving in –Y (position 2)
cut along the rear face, by moving in –X (position 3)
remove the cut by moving back to Y0 (position 4)
rapid move to the start of the cut by moving in +X (position 1)

Turn the spindle on. If your spindle is controlled by Mach3, type
S800 M3
otherwise set the speed and turn it on manually. M3 makes the spindle turn clockwise.
Set the cutting feed rate using
F80

The first two repetitions of the sequence are:

```
G0 Y-0.2
G1 X0
G0 Y0
G0 X170

G0 Y-0.4
G1 X0
G0 Y0
G0 X170
```

At each repetition, the cut adds another -0.2 to Y

Don't worry if the first couple of cuts don't remove any material. The cutter will eventually come close enough to the edge to begin cutting.
When the face has been cleaned up, stop. Turn the spindle off (M5); or turn it off manually. I took 8 cuts, because that cut edge was significantly squint.
Release the clamps, and spin the plate 90 degrees anti-clockwise, then flip it over, making the front the back, and vice versa. Why? I used the as-manufactured edge of the material as one temporary reference edge. It makes sense to clean up that edge now, referencing it to the newly cut edge. Turning the plate clockwise brings the freshly cut edge approximately parallel to the Y axis, but the former reference edge ends up at the front of the mill table, where it can't be reached by the cutter. The "flip" puts it at the rear, leaving the new reference edge where it needs to be, parallel to the Y axis. Not precisely parallel, though, so use the angle plate or flat faced block in the same way as before, to square the reference edge of the plate across the table. From the Program Run screen, jog the cutter to roughly the centre of the rear side, and zero the Y axis DRO. Gently slide the plate to touch the cutter, then clamp it in that position. Jog the cutter a small amount away from the plate. Remember it is the CP which moves, so jog in the positive Y direction. Then use the same method as before, to clean that face up, starting your cuts at the right hand end of the plate. It shouldn't take much at all, because it should be at right angles to the current reference face. I took one cut of 0.2mm.
You now have two reference faces at right angles to one another. Mark them clearly.
You can clean up the other two, if you wish. They will look so much better.
Photo 44 shows the finish resulting from a sharp cutter and the suggested feeds and speeds. As a matter of interest, you can work the machine much harder, and mine will take a cut of 1mm at the same speed and feed rate, if you can stand the mental strain. It's a real test of your machine, though, and unnecessary for this kind of job.

It's laborious, typing all those commands, so you will be ready to learn some more efficient techniques for dealing with that, shortly. For now, though, typing commands in MDI mode does allow step-by-step control over the mill's movements and actions.

Photo 44: surface finish from the side of an end mill..