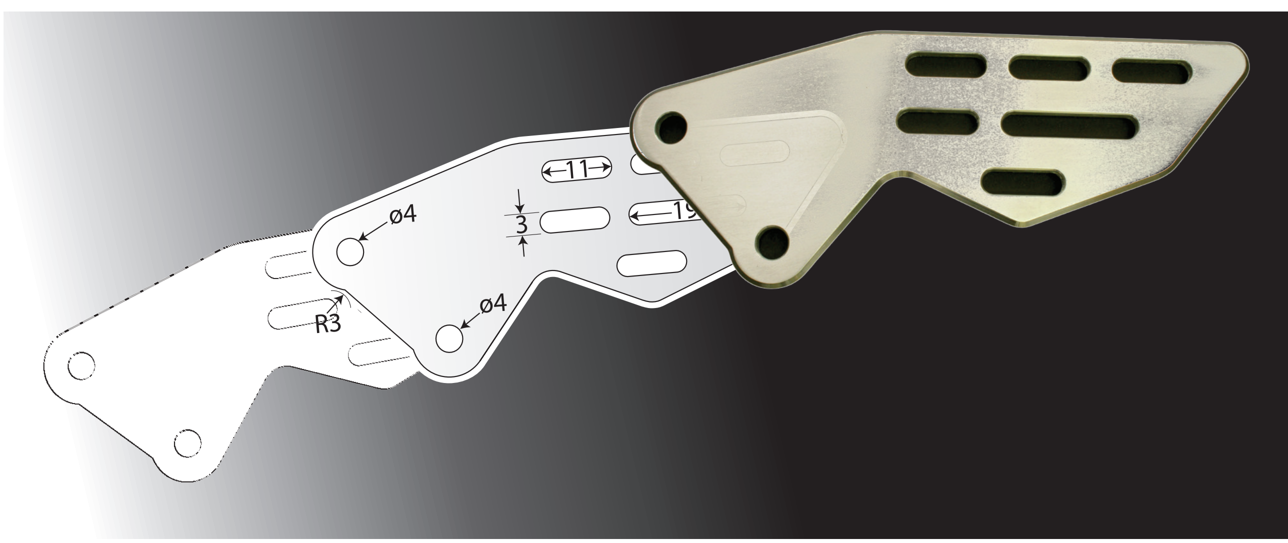
CNC in the Workshop



Part 4

Includes the section on "Backlash", which was formerly at the end of Part 3.

Part 4

In this part of the series, we consider some of the more mechanically-related practical matters such as backlash feeds and speeds for your mill.

Note: The initial section on Backlash was originally printed at the tail end of Part 3, in MEW 205.

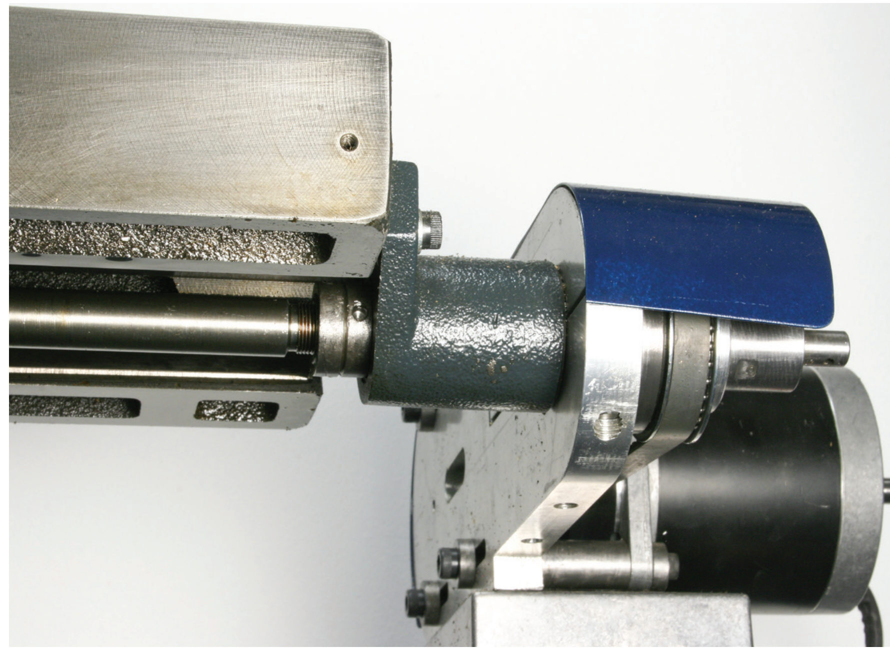


Photo 22: The collars to the left of the shaft support casting can be adjusted to monimise backlash.

BACKLASH

Before we rush off and cut metal, we need to deal with some important matters, beginning with backlash. On a perfect machine, there would be no backlash in any of the feed screws, so giving a command to move from one point on an axis to another would always result in the Controlled Point (CP) arriving exactly at the second point. Backlash is often present in the real world, though, largely because of necessary clearance between the flanks of the lead screw nut and the lead screw itself.

Fig 15 shows a cross-section through the nut and the lead screw. In the upper diagram, the flank of the thread of the lead screw is pressing against the thread in the nut, pushing the nut to the right. There is a little clearance between the rear flank of the lead screw thread and the following face of the thread in the nut, and that results in backlash. If the rotation of the nut is reversed, the lead screw will rotate a little without the nut moving, until the rear flank of the lead screw has taken up the clearance and can press against the mating face of the thread in the nut. That is lost motion, and, where the lead screw is being turned by a stepper motor, some steps are used just to take up the clearance (i.e. the backlash). Unless the computer is told that steps are lost when the direction of motion reverses, it will not rotate the leadscrew enough for the CP to reach the intended position.

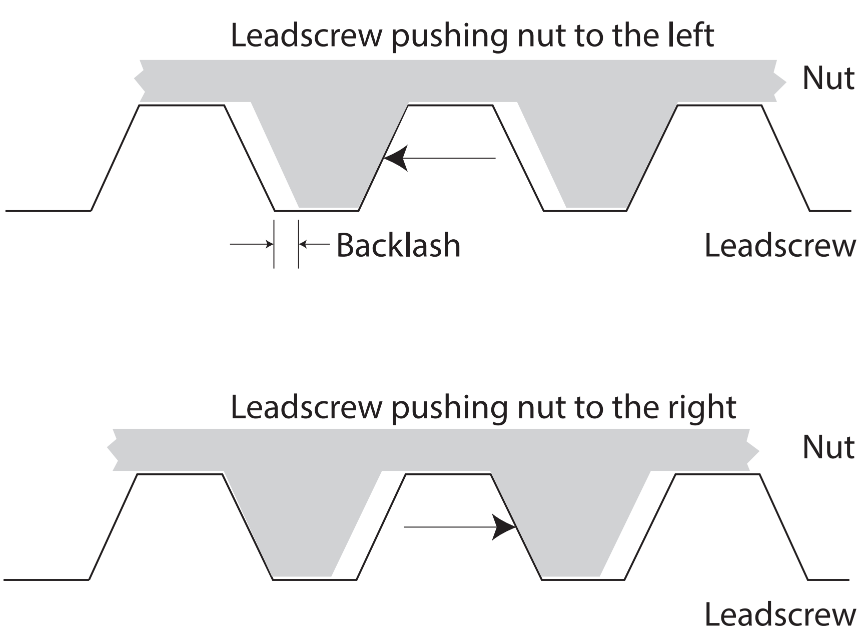


Fig 15: The effect of backlash on a leadscrew pushing a nut..

Backlash acts as a built-in error in an axis, so that when a reversal occurs, the distances moved are always short, by the amount of the backlash.

The remedy for this situation is to measure the backlash on each axis, and enter the values into Mach3. Then, when reversing a leadscrew, Mach3 can add some extra steps to take up the backlash, before taking the steps to move the CP to the intended position. It's the kind of thing that's a nuisance for a human to do, but easy for a computer to do each and every time it reverses the direction of the leadscrew.

Before measuring the backlash on any axis, make whatever adjustments you can, to eliminate as much backlash as you can. Remove any slackness caused by the adjustment collar at the bearing where the lead-screw exits the table (photo 22) if there is one on your machine. If the axis nut is adjustable, adjust it now, to eliminate as much lost motion as you can without introducing excessive friction.

Set up a flat faced object at right angles to the X axis (for example). Photo 23 shows an angle plate clamped to the table. It has been squared across the table using an engineer's square, and that is sufficiently accurate for this job.

Set up a plunger type Dial Test Indicator (DTI) with a ball-ended foot, mounted on a stationary part of the mill (the head, the column, or the base) and touching the face of the angle plate (photo 24).

Jog to move the CP in the positive X axis direction (i.e. to the right) so that the DTI is roughly mid-travel; zero the Mach3 X axis DRO; then zero the DTI scale (probably by rotating the bezel).

Using MDI mode, move the CP to X10 (G0 X10). The DTI should now read 10mm travel.

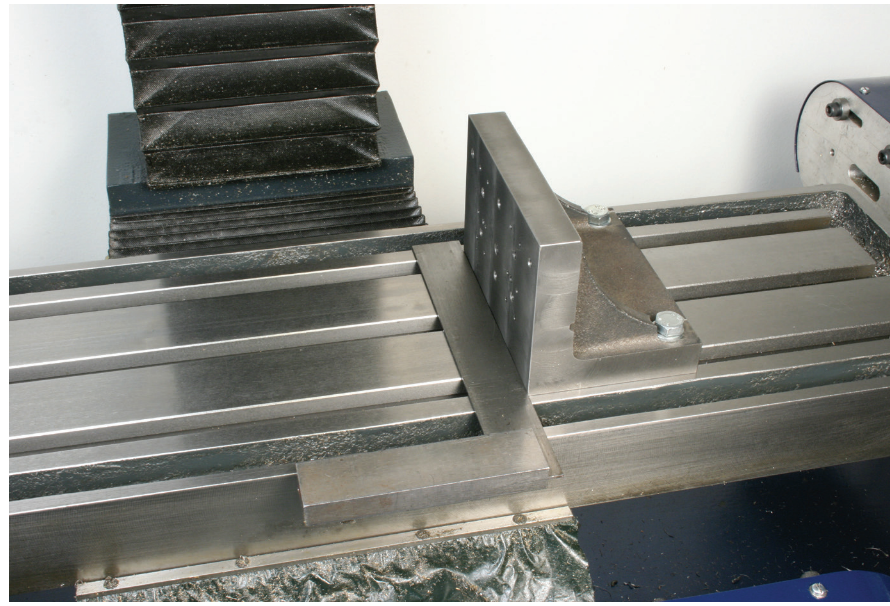


Photo 22: The collars to the left of the shaft support casting can be adjusted to monimise backlash.

The initial move before zeroing the DRO and the DTI took up any backlash. The G0 X10 should then have moved the CP to X10. If the DTI doesn't read 10mm, something is wrong, mechanically. Check the DTI points along the X axis, and that the face it is touching is at right angles to the X axis. If the DTI reads significantly less than 10mm (e.g. 5mm) you have something wrong with your Mach3 setup, and may have entered a wrong pulley ratio.

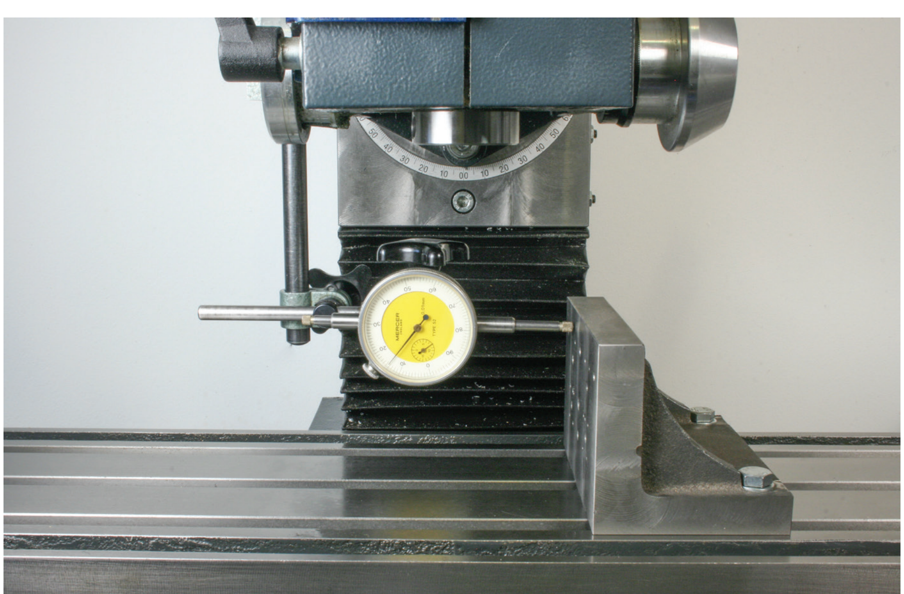


Photo 24: Using a DTI to measure X axis backlash.

Assuming the DTI reads 10mm, command the CP back to X0 (G0 X0). Ideally, this should make the DTI read 0 again. Any difference will be the backlash. Make a note of the deviation from 0, which will be the reading on the DTI.

Repeat the whole procedure, taking the CP to the left, first, then advancing to half travel and zeroing there, before moving to the right, and back to zero. After repeating a few times, you should have a fairly accurate backlash figure.

Enter that figure using Config > Backlash. The "units" are millimetres or inches; whatever you set up Mach3 to use.

Repeat for the other axes.

CUTTING SPEEDS AND FEED RATES

G0 and G1 commands allow us enough control over the Controlled Point (CP) to be able to take some cuts and make some handy items. In preparation for a cut we need to be able to choose an appropriate spindle speed and feed rate.

Spindle speed is the rotational speed of the spindle, in revs per minute (rpm).
Feed rate is the linear speed at which the CP moves past the work, in millimetres per minute (mm/min) as shown in fig 16.

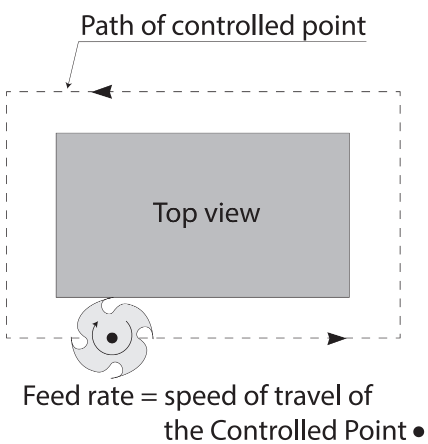


Fig 16: The Controlled Point moves past the work at a given feed rate.

Both spindle speed and feed rate are calculated in exactly the same way as for a manual mill.
The first important aspect is the cutting speed i.e. the speed at which the rotating cutting edge meets the work (fig 17). This differs for different materials, and some common cutting speeds are listed in table 1.

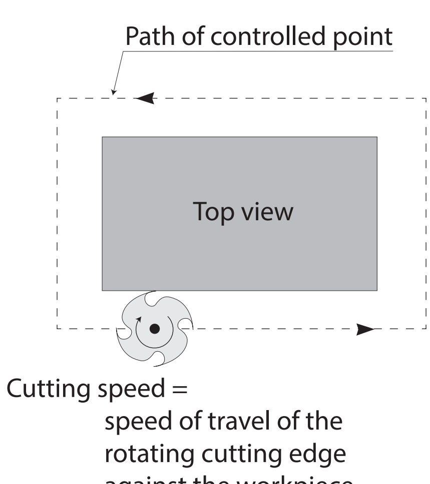


Fig 17: Cutting speed is measured at the periphery of the cutter, where the cutter meets the work.

Cutting speeds for common materials

Material	Cutting speed (M/min)
Mild steel (230M07)	30
Aluminium (L111/6068)	75
Brass (CZ121/CW614)	100
Cast iron	20
Stainless steel (300 series)	25

Table 1: Typical cutting speeds for common materials.

Spindle speed is directly related to the cutting speed and the diameter of the cutter:

Spindle speed (rpm) = (cutting speed (M/min) x 1000) / (3 x cutter diameter (mm))

To save having to do the arithmetic, there is a speed table and a spreadsheet for calculating speeds on the support website. Or you can use proportion, because in the case of steel (30M/min),
a 10mm cutter should rotate at approximately 1000rpm.
A 20mm cutter would rotate half as fast, at 500rpm, and
a 5 mm cutter twice as fast, at 2000rpm, and so on.

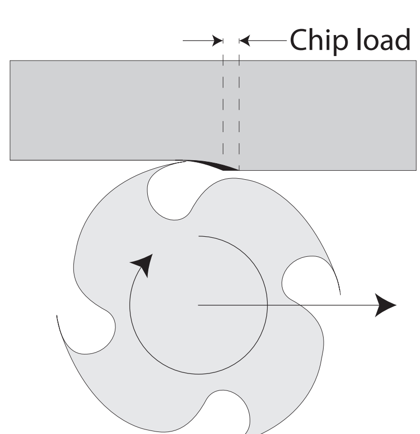


Fig 18: Chip load (conventional milling)

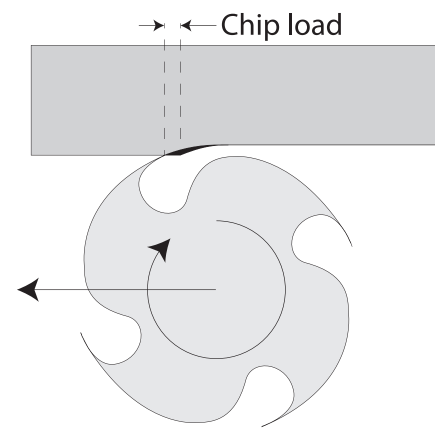


Fig 19: Chip load (climb milling)

Feed rate, on the other hand, depends on the number of teeth on the cutter, and the chip load per tooth. Chip load is the amount of material cut per tooth (figs 18 and 19). Don't confuse this with the depth of cut which is the distance the cutter is advanced into the work. The way in which the depth of cut is measured depends on the orientation of the cutter in relation to the work, but for this project we will be concerned with the radial depth of cut, as shown in fig 20. Chip load typically varies between 0.02mm for a finishing cut and 0.125mm for a roughing cut. Chip load varies with cutter geometry, rigidity of the mill, the length of the cutting edge in contact with the work, and the power of the spindle motor, so you should experiment a little on your own machine. 0.02 to 0.125mm is a reasonable starting point, though.
Feed rate(mm/min) = spindle speed(rpm) x no. of teeth on the cutter x chip load per tooth(mm)

Using a spindle speed of 1000rpm, a cutter with 4 teeth, and a chip load of 0.025mm, feed rate = 1000 x 4 x 0.025 = 100mm/min

In days of old, we used to be expected to do this kind of calculation in our heads, as a warm-up before breakfast, but to relieve the pain the support website lists some useful feed rates. Softies.

That's enough of the theory and setup for now. Although there is more to learn before we can make full use of automated methods on the CNC mill, we have enough to get going, so let's cut some metal.

VICE PLATE

We can do a lot by clamping a workpiece to the mill table, but on many small mills the head does not come close enough to the table for short cutters to reach the work. This is more an inconvenience than a real problem, but it is a nuisance all the same.

It is probably more common, and certainly more convenient, to grip work in a vice. The challenge with that is that the vice must be secured on the mill table with the rear jaw parallel to the X axis. This is best done using a lever dial test indicator (DTI) against the rear jaw (photo 25). The DTI can be secured to any convenient stationary point (like the head, or the vertical column). If you only have access to a plunger DTI, you can clamp the flat face of an object against the rear jaw, then run the plunger against that face (photo 26).

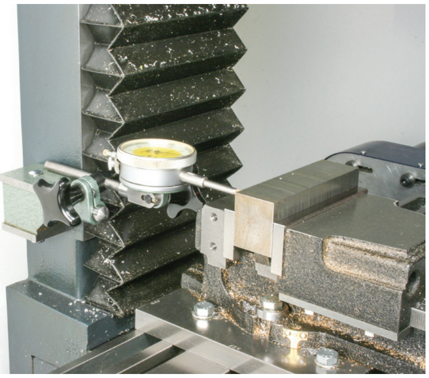


Photo 26: Using a plunger-type DTI against flat material gripped against the rear vice jaw.

Secure the vice to the table, using proper T-nuts and bolts through large washers. I use home-made T-nuts and M8 hex set screws.

Set the DTI stylus or plunger foot close to the left end of the jaw or face, zero the DTI using the rotating bezel, then zero the Mach3 X axis DRO and the Y axis DRO. Select a distance which will take you just short of the right end of the jaw (photo 27). To make adjustment reasonably easy it helps if the DTI travels roughly the same distance from the centre of the jaw to the right end as it was to the left end of the jaw. Using MDI mode, enter a G0 command to move that distance along the X axis e.g. G0 X90.

The DTI will no doubt show a deflection. Loosen the vice and gently rotate it so that you lose half that deflection, then re-tighten. This is easiest if the vice has a rotating base. The safest thing to do next is remove the DTI. Travel back to the start using G0 X0. Remount the DTI, zero it, and repeat the traverse to the other end of the jaw. Repeat the adjustment. Keep repeating the procedure until there is no deflection when the DTI travels from one end of the jaw to the next. Fully tighten the vice securing bolts (and the clamp bolt on the rotary base, if that is fitted). Check the alignment again, just to be sure, then remove the DTI.

This process is absolutely necessary every time the vice is secured to the mill table. Frankly, it's a pain. There are various ways around this, with the most popular being a tongue fitted to the underside of the vice so that it can be replaced in the same position every time. I'm not a particular fan of that arrangement. Instead, I use a plate with two adjacent reference edges, each of which can be set at right angles to the front edge of the table, using a square. The vice is bolted to the plate and adjusted, once, so that its rear jaw is parallel to the X axis. After that, it is simply a case of placing the vice plate on the mill table, and squaring it off the front edge (photo 28), then nipping up the bolts. To set the vice with its jaws parallel to the Y

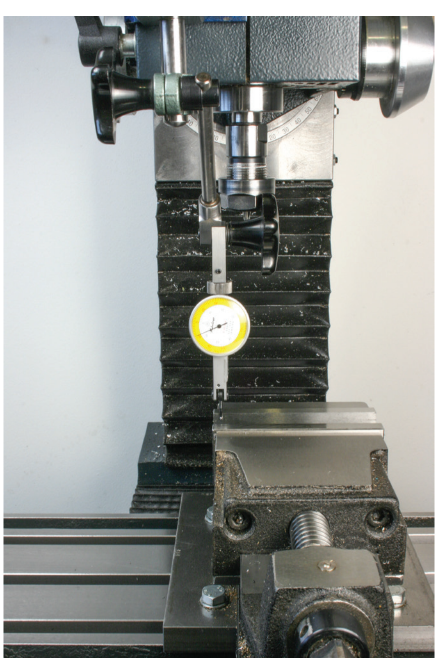


Photo 25: Lever-type DTI against the rear vice jaw.

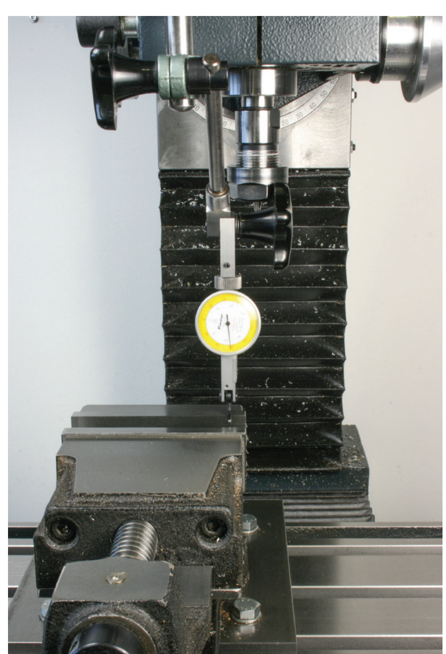


Photo 27: Lever DTI near the other end of the vice jaw.

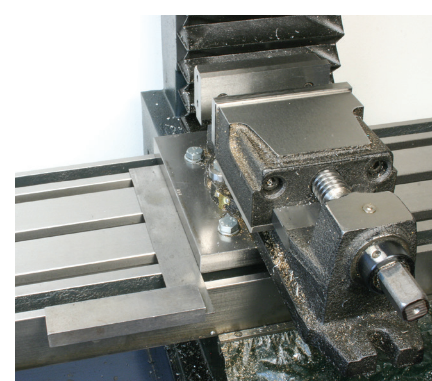


Photo 28: Lever DTI near the other end of the vice jaw.

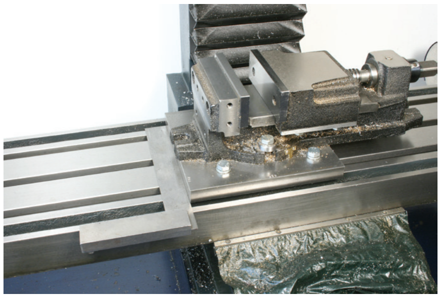


Photo 29: Using a square against the mounting plate to set the vice jaws parallel to the X axis..

axis, square the other reference edge across the table (photo 29), then tighten down. The accuracy of this method does depend on the front edge of the mill table being parallel to the X axis, but that can be checked with the DTI. Secure the DTI holder against a non-moving part of the mill, with the lever or plunger against the front edge of the table, then move the table left and right and note any deflection (photo 30). If there is no deflection, you can use a vice plate.

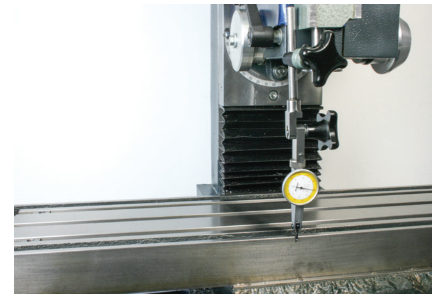


Photo 30: Checking the front of the mill table is parallel to the X axis..