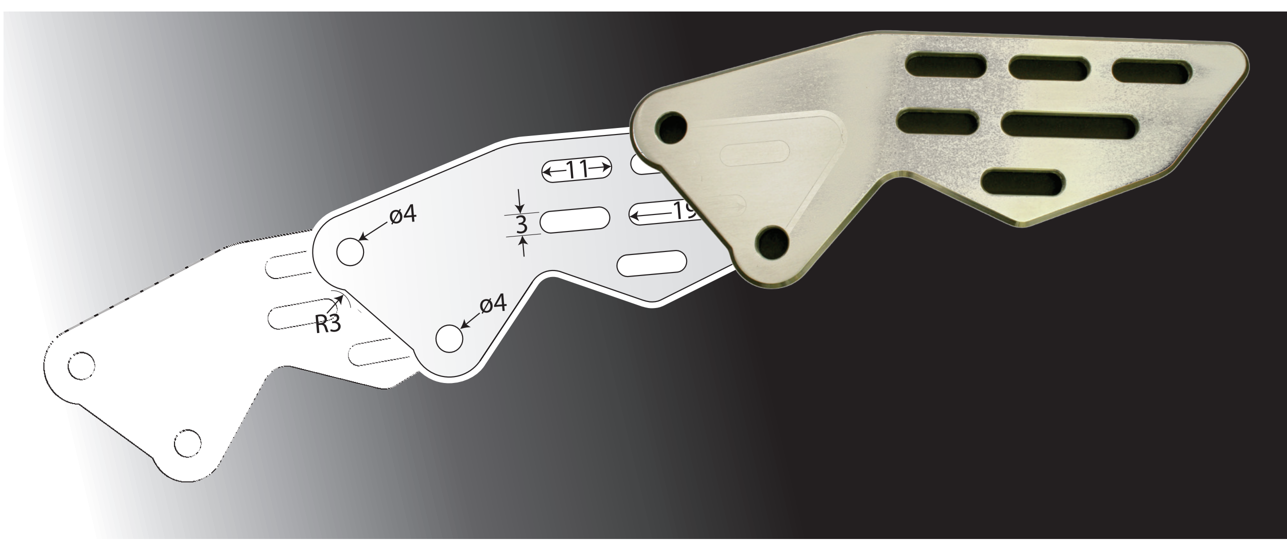
CNC in the Workshop



Part 10

Part 10

In this part of the series, we continue looking at circular paths and making the example workpiece.

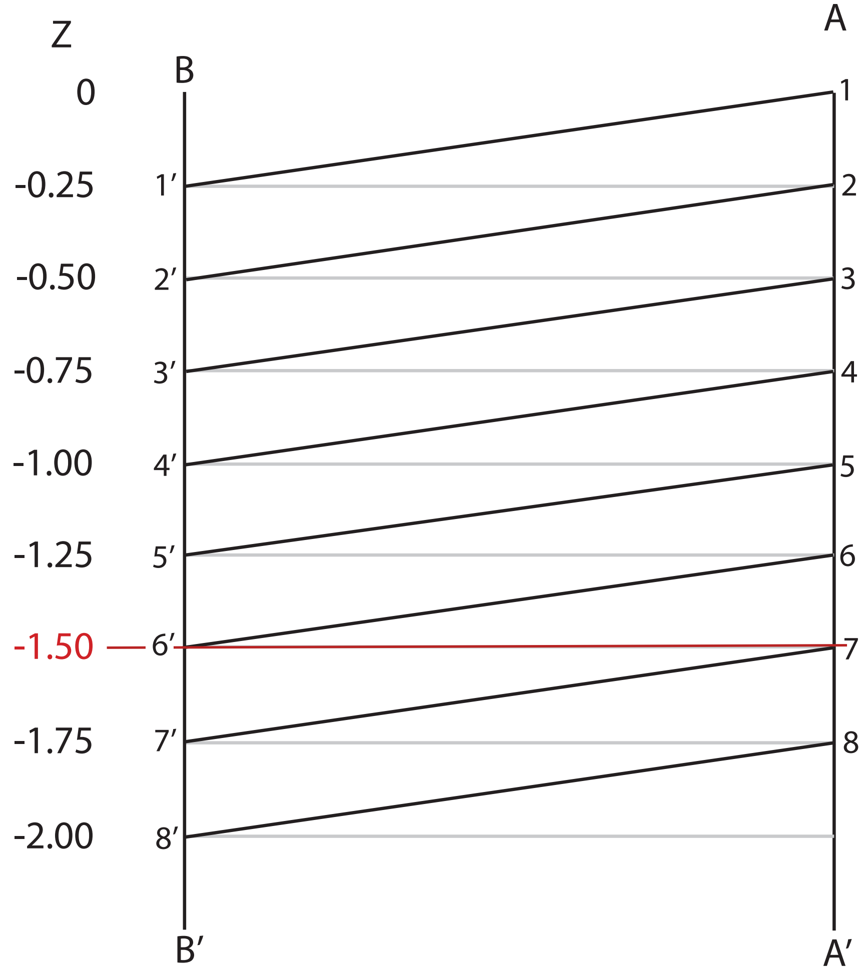


Fig 48: The spiral cuytter path unwrapped..

THINKING ABOUT RAMPED CUTS

Let's just take a moment to think about ramped cuts in a slightly different way. Imagine drawing the path of the CP around a cylinder. It would look a lot like a screw thread, wouldn't it? Now think about slitting that thread, vertically, and unwrapping it like a sheet of paper, then drawing the path on paper. Fig 48 shows what it would look like. It might take a moment to visualise that. Line A-A′ is the start of a cut each time around. In fact that is the starting point for each complete revolution of the cutter. Line B-B′ is the end of the cuts. But because the end is the same place as the start of the circle, A-A′ and B-B′ are actually the same position on the CP path (we unwrapped the thread from around a cylinder, after all, so that pair of lines are the two sides of a single cut down the cylinder).

For the first pass, the CP starts at point 1 and descends at a uniform rate as it moves around, finishing at 1′. But 1′ is the same point as 2, where the next pass starts.

Cut from 2 to 2′ which is the same as 3 … and so on, with the CP descending during each pass.

The horizontal lines give a visual indication of depth, so 1′ and 2 are at the same depth; 2′ and 3 are at the same depth; and so on.

At the end of the first pass, the CP is at point 2 (same as 1′). Now look at the material in front of the cutter (fig 49). That's the full height of material from that first cut, because the cutter descended slowly from Z0 to Z-0.25 and didn't remove anything much from the very start of the cut. It's the same at the end of any pass and the start of another. So although the cutter is descending during the cut, it faces a constant depth of cut, except on the first and last cuts.

Now look at what you might think was to be the last pass. It starts at point 6 and ends at 6′, where Z is -1.50. That won't release the waste, because of the descending cut, and the blue triangle shows the material left in the workpiece. That's why we need a final cut. That final cut can be one more tapering cut G2 X18 Y21 Z-1.60 I-4 J0 followed by a level cut G2 X18 Y21 Z-1.60 I-4 J0 to ensures the cutter cuts right through all the way around, with a bit of extra depth (0.1mm) just to make sure the cutter cleans up the bottom edge of the circle.

Or it could be G2 X18 Y21 Z-1.51 I-4 J0 then G2 X18 Y21 Z-1.51 I-4 J0 if you feel you want to work with more accuracy. The method is the same.

Taking an alternative approach, the cuts from 5 onwards could be:

```
G2 X18 Y21 Z-1.49 I-4 J0
G2 X18 Y21 Z-1.49 I-4 J0
```

Those commands cut by ramping down to -1.49, then cleaning up the bottom of the slot with a level cut at -1.49, leaving a thin skin all the way around the waste to hold it in place. It all depends on how flat the workpiece is; how uniformly thick it is; and how accurately Z0 was set at the top surface of the work.

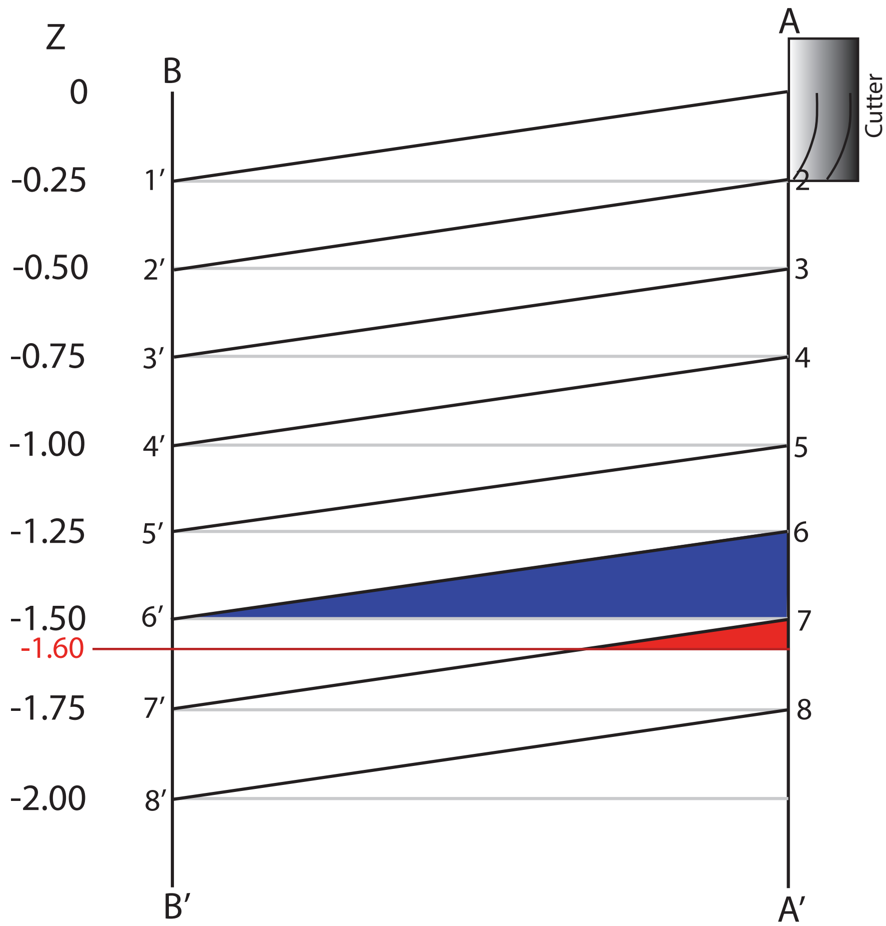


Fig 49: The cutter 'sees' the full height of the last cut at the start of the next cut..

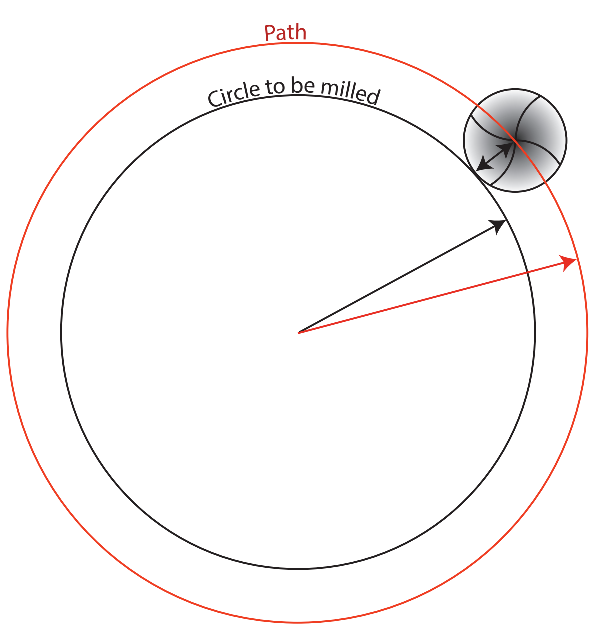


Fig 50: To machine the outer circle, the CP must lie outside the circumference, displaced by half the cutter diameter..

To create the large circle, the cutter should be on the outside of that circle, so the CP will follow a path a little outside the circumference. The radius of that path will be: radius of the circle + radius of the cutter (fig 50).

For a conventional milling action, the cutter should travel anticlockwise around the circle, so that will mean a G3 command.

Take the cutter out to the right hand side of path: G0 X47 Y26

The current point is X47 Y26; and the end point will be the same.

I = 24 – 47 = -23

J = 26 – 26 = 0

So the appropriate command is: G3 X47 Y26 Z~ I-23 J0 where ~will be replaced by a number to provide a depth of cut.

In the same way that we ramped the cutter down as it cut its way around the circle, we can do the same here; and the sequence might begin:

G3 X47 Y26 Z-0.25 I-23 J0

G3 X47 Y26 Z-0.5 I-23 J0

and so on. Finish the sequence in the same way as for the earlier circles.

MACHINING THE WORKPIECE

How will we hold the material so that it is secure both during and after it is cut?

The best way to secure a thin sheet is to attach it to a piece of more substantial backing material, then hold that securely using clamps or a vice. For this project, we will use thicker material, held in the vice and acting as a fixture (photo 74). You will need a piece

Photo 74: The block sitting on parallels and gripped in the vice..

of material no harder than the workpiece, and I suggest a piece of aluminium at least 50 x 50mm, and as thick as you can find. The sides need to be thick enough to allow the vice to grip the material securely, so 10mm or more is best. If you can't find that thickness, use 6mm or thicker, but attach it to a block of material which can be held in the vice.

I happened upon a piece of 15mm thick aluminium which cleaned up to 60 x 60, which is ideal. I have a small collection of pieces of aluminium with thicknesses from 8mm upwards, and 50 x 50 or larger, which I use for this kind of job. If you are making clock wheels, or anything else of that sort, you will find a ready use for this way of holding work, and although aluminium is no longer cheap, it is well worth laying in a little stock of sheet and/or block.

The corners of the sheet are outwith any machining operations, so those corner areas could be used to secure the sheet to the fixture by using machine screws. The lower the profile of the screw head, the better, as a rule, but you can use any convenient size. Allow enough room for a cutter to pass close by and, for this project, no part of the screw head should be closer than 22mm, measured from the centre of the largest circle at (24, 26), and hole co-ordinates are suggested in the next section. Use soft headed screws, rather than high tensile fasteners, in case the cutter touches the screw head. At 2mm diameter it will just squeeze past, and a larger diameter cutter will certainly take a bite from the head.

The large circle will eventually be cut free of the main sheet, so it needs to be held securely so that it doesn't fly off. With a small 2mm cutter and relatively shallow cuts, double sided tape covering the whole of the underside of the sheet would hold the various circles as they are cut through, so that's one solution. It does involve a bit of cleaning up afterwards, though, and it is never 100% secure. A more secure alternative is screws and clamps, and for that we will need two tapped holes at points (14, 21) and (34, 21). Any convenient size will do, and M3 is a good choice for all the securing screws. One other possibility is to leave small tabs of material in the grooves cut at the edges of each circle. That is possible now, but involves a bit of thought in programming. Later, we will see that a CAM program can do that easily.

Prepare the workpiece

Now we need to prepare the workpiece by drilling 4 holes near the corners (shown as 4 black dots in fig 45) which will allow us to secure it to the fixture . Clamp the workpiece to a backing suitable for drilling into (not the fixture), and make the left edge of the material approximately parallel to the Y axis (fig 75) .

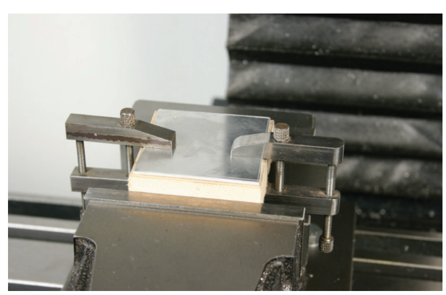


Photo 75: Sheet on a backing, ready to have the corner holes drilled..

Set the Work Origin at the front left corner of the workpiece, perhaps by taking the CP to that position and entering 0 into the X and Y DROs. If your material is a little larger than 50 x 50, it is a good idea to set the Work Origin a little in from the corner, to give yourself some leeway.

You already know how to position the CP and drill a hole, so using MDI commands, or by writing a simple program, drill M3 clearance holes (3.2 or 3.5mm) for the securing screws, at (5, 5), (5, 45), (45, 5), (45, 45). If you intend using the fixture again, for this or any other job, the positions of those holes in the workpiece will always be the same, so you might want to write a simple program to drill them, and save it for future use. You could also create a small plate which could be used as a jig to drill a blank quickly, in the drilling machine. That's handy if you are making several identical items. It also means you can easily cope with slightly different sizes of sheet, by visually centring the jig over the sheet before drilling.

Prepare the fixture

Square up all four edges of the material you will use for the fixture (see MEW 209).

Set the material on parallels in the vice. If the material is sufficiently thick, set it so that its top surface is at least 3mm clear of the tops of the jaws (fig 52). Then face the block. If you are using thinner material, set it as high as you can but it need not project above the tops of the vice jaws; and omit the facing operation for this job.

To set the work origin we need a reference for X and Y which locate the front left corner of the work. Furthermore, we will drill some holes, then manually tap them, so it might help if the workpiece could be removed then replaced in exactly the same position. The logical reference for X is the left edge of the material, and the logical reference for Y is the front edge of the material. As the vice sits, in the conventional position, the left edge of the material locates against fresh air (photo 74) and the front edge is against a movable vice jaw, which is not much better. One solution is to turn the vice through 90o so that the fixed jaw can be the X reference, and, if you made the vice stop in MEW 208, 209 and 210, that can act as a fixed reference for Y (photo 76). That seems like a workable

Photo 76: Block located against two fixed edges..

solution for most workpieces like this.
Use one of the methods from MEW 210 to set X0 at the left edge and Y0 at the front edge of the work, or the surfaces against which the work will locate. Note that there is a small notch in the vice stop right at X0 Y0, so use the edges instead.
Assuming your material is a little over 50 x 50mm, the first thing we should do is move to the centre of the material and set the Work Origin X0 Y0 there. Notice that the Work origin can be reset in a new position any time we need.
Using your smallest centre drill, make a tiny mark there, so that we can come back to this point and use it to set a Work Origin next time we use this fixture.
Now move out to the left front corner of the 50 x 50 area where the workpiece will be held, by moving to X-25 Y-25, and set the Work Origin to X0 Y0 there.
The purpose of all this fiddling around with the Work Origin is simply to get the 50 x 50 area placed reasonably symmetrically within the boundaries of the surface of the fixture plate.

Drill 2.5mm holes for the M3 screws at the corners (avoiding drilling into the parallels…) at (5, 5), (5, 45), (45, 5) and (45, 45).
Drill 2.5mm holes for the M3 clamping screws at (14, 21) and (34, 21).
Drill all those holes again, using a 3.2mm drill, approximately 1mm deep, in preparation for the tapping operation.
Tap all the holes manually at this stage, to complete the work on the fixture (photo 77).

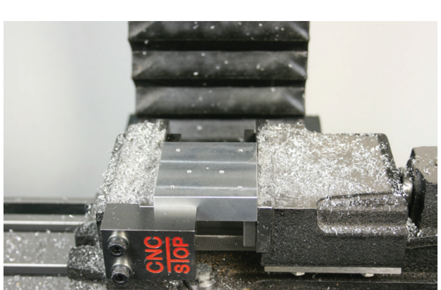


Photo 77: Fixture block drilled, tapped and ready for the workpiece..

Organise two large clamp washers to sit over the two smaller holes when they have been cut out. These can be plain and simple, or you can turn two washers 18mm diameter and 3mm thick, with a little spigot 1mm thick and 9.9mm diameter, to aid location (photo 78). Each washer has a clearance hole (3.2mm) in the centre.

Photo 78: Clamp washers..

Put the workpiece on the fixture

After tapping the holes, put the fixture back in the vice, and attach the workpiece using screws through the four corner holes.

Set the speed and the feed rate
For a 2mm diameter cutter, the theoretical spindle speed would be in the region of 15000rpm. A more achievable speed on most machines might be 3000rpm. At that speed, the theoretical feed rate for a 3 flute end mill would be approximately 900mm/min. I suggest you start as near to 4000rpm as you can, and use a feed rate of 100mm/min. Increase the feed rate if you feel the cutter is capable of taking the load.
One problem is that many grades of aluminium are quite “sticky” and tend to clog the cutter, leading to breakages. I normally lubricate the cutter with turps, but it is rather smelly, and a viable alternative is lip salve, as discovered by Mark Noel (MEW 210). I applied it using a stirring stick for coffee, and it worked well.

Create two programs

Create a program to machine the large arc and the two smaller circles. Take the time to follow the structure suggested in MEW 209, with a title and comments; an Initialisation Block; and the Main program.
Save the program; test it; and modify it as necessary until you are happy with the toolpath shown in Mach3.
Create a second program to cut the outer circle.

Run the programs

Run the first program, then retract the tool to a safe height, and put the two clamp washers in place (photo 79). Then run the second program.

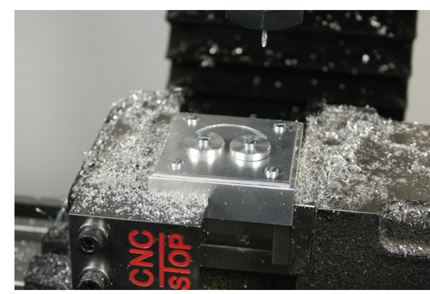


Photo 79: Clamp washers in place, ready to hold the work when it is separated from the man sheet..

Backlash marks

When you remove the work, look carefully at the finish on the inside face of the small circles and the outside face of the larger circle, particularly at the points shown in fig 51. If there are little bumps or grooves at those points, it is probably the result of backlash. Those are the points at which slide motion reverses for one of the axes, and at that point backlash takes its toll. If the backlash settings within Mach3 don’t quite match the actual physical size of the backlash, the feed screw will take up a little too much or a little too little backlash, and you will see the result in the surface finish. This is also where ballscrews suffer from the disadvantage of low friction, because the slide may float free of the screw momentarily as the screw takes up the backlash. It’s a pain. The ideal

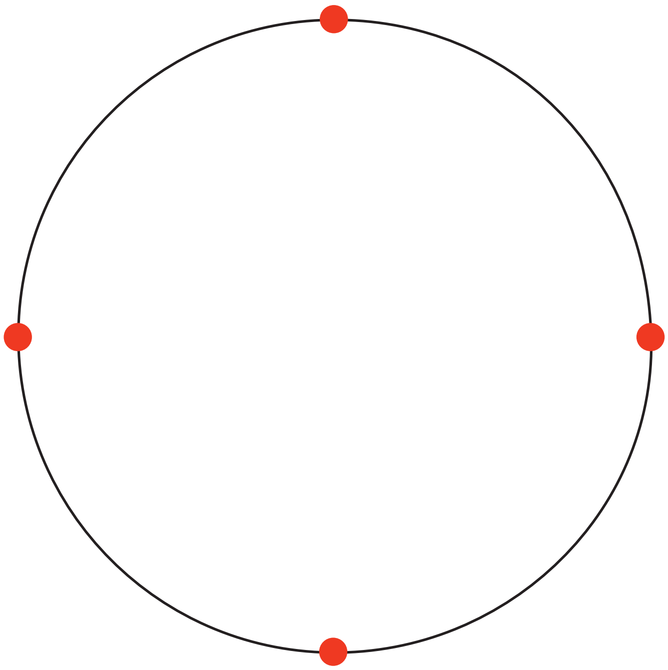


Fig 51: Backlash points shown in red..

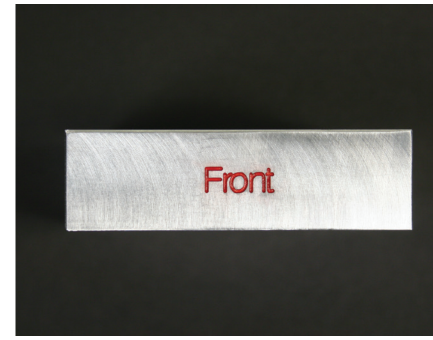


Photo 80: An obvious indication of the orientation of the block..

is, of course, zero backlash and pre-loaded feedscrews.

In the real world, we can but dream. If the marks are significant, you may want to re-measure the backlash and correct the backlash values within Mach3 (using the Config > Backlash menu).

A final flourish

Remove the finished disc and deburr the edges. Paint the face yellow, then fill the arc with black paint. Repeat after me, in a very cheesy voice, "Have a nice day".

I'm sticking mine on my lapel.

Finishing the fixture

That's a handy fixture we've made, and it will find plenty of uses for other jobs. It would be a good idea to mark the front face somehow, so that it can be put back in the same orientation each time it is used. Marking can be anything from a simple pair of punch dots (two in case one looks like a random mark) to a more easily read legend (photo 80).

If you fancy the legend, there's a program and some instructions for that, along with complete programs for this current project, on www.cncintheworkshop.com