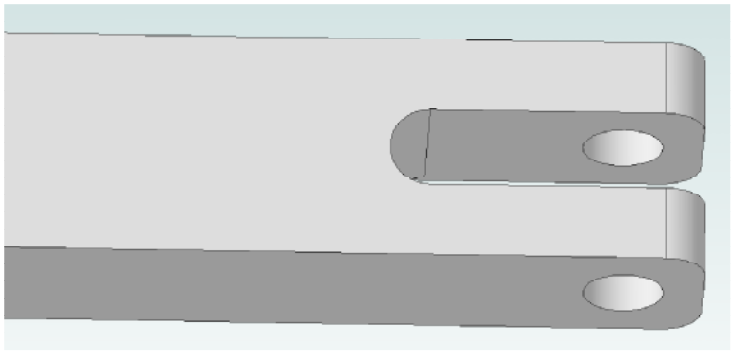
Designing and Milling a Slot, version 2

By R. G. Sparber



One of the cool things you can do with a milling machine is to cut slots. At first glance, you might think it is simple – just lower the end mill down past the bottom of the bar being cut and feed in. Well, that will get you a slot. It will also unnecessarily wear out your end mill.

Another side effect of this straight forward method of cutting a slot is that the resulting width of the slot may not be equal to the diameter of the end mill. The stresses on the cutter can cause the end mill to bend which makes the slot wider.

In general we do a rough cut which takes a lot of metal quickly but is not precise. Then we make a finish cut that is around 0.01" deep that gives us the needed precision and smooth finish. If you are just running the mill at full depth and full width, there is no room for the finish cut.

So we get to our first "rule" of slot design – don't call for a slot width equal to or less than the diameter of your end mill[2].

From the above discussion we also get our second "rule" – leave about 0.01" after the rough cut for the finish cut.

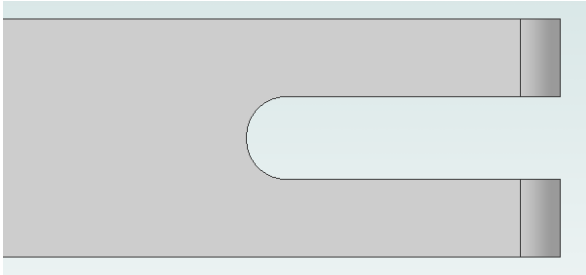
And while I'm at it, let me toss in a third "rule" – remove as much metal as possible with means other than end mills.

[1] You are free to copy and distribute this document but not change it.
[2] Narrow slots are usually cut with a slitting saw.

This last rule can entail lots of methods. If you can cast the part, then you are certainly being very efficient since getting to the rough shape involves no waste.

But even if you are using bar stock, you might be able to rough saw an opening. In the case of an end slot as shown above, drilling a hole at the far end of the slot and sawing the sides can be quick and low cost.

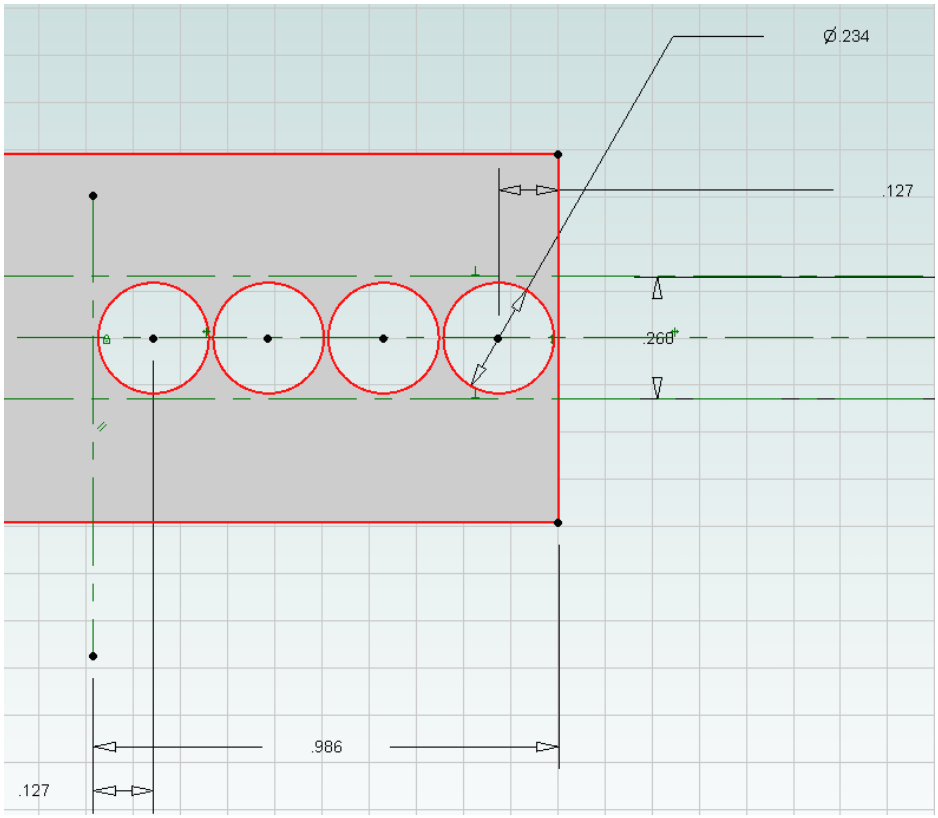
The method I will explain here involves drilling first and then milling. Drills are cheap, easy to sharpen (if you own a Drill Doctor®) and fast. With a little planning, you can get great results.



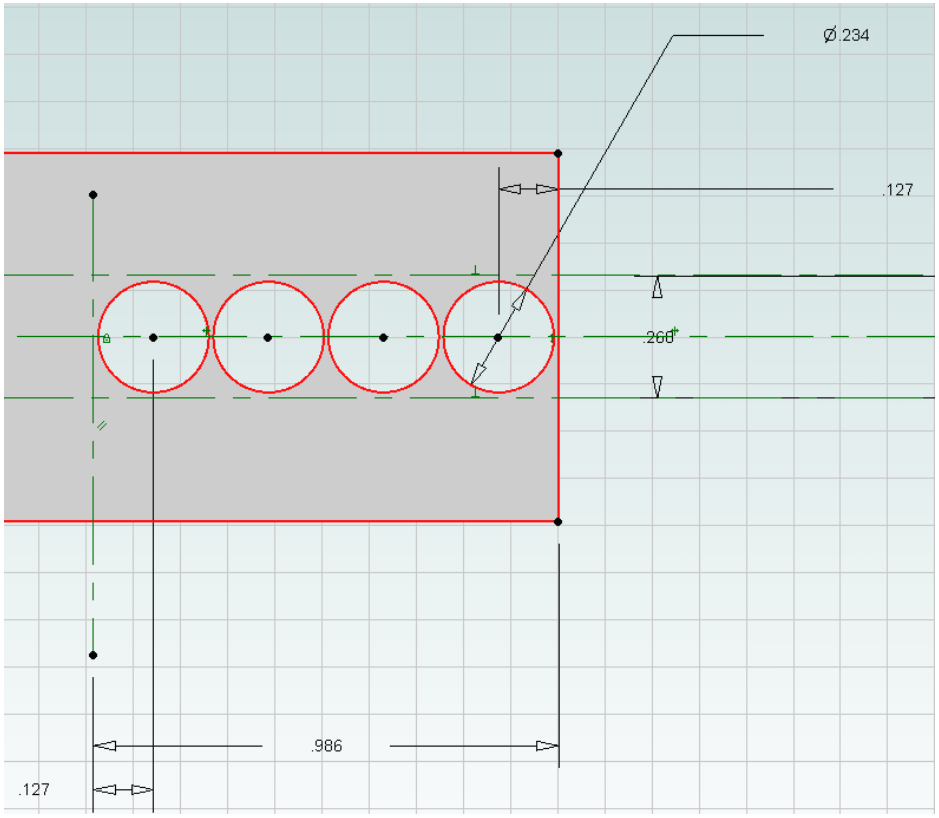
I want to drill a series of holes along the centerline of the slot. If the holes overlap, I risk having "side break through" between adjacent holes which can seize up the drill or deflect it. If I leave about 0.01" between holes, I don't have this problem.

I can choose a drill that will give me about 0.01" between hole and finished slot width. There is nothing magical about the 0.01". It could be from about 0.005" to 0.015". A lot depends on how accurately you can place the holes.

Now, often the length of the slot is not all that important. If you afford it, pick a slot length that is 0.01” from the last hole.

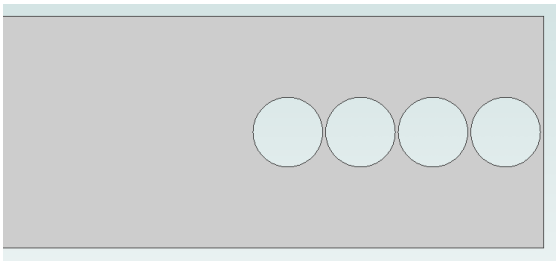


Here you see my hole placement for the above slot. I wanted a finished slot width of 0.260”. Leaving 0.01” top and bottom, this gives me an ideal drill size of 0.260” - 0.01” = 0.240”. If I go with a $\frac{15}{64}$” drill, the hole will be around 0.234”. That gives me ideally 0.013” for my finish cut.



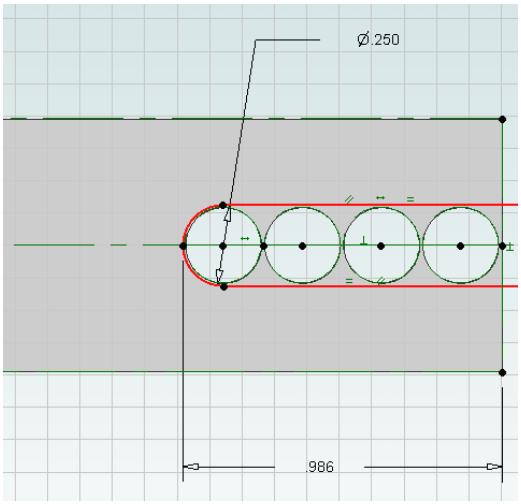
The length of my slot had to be greater than 0.8” and less than about 1.2”. If I drill four holes and add 0.01” beyond the last hole, I get a finished slow width of 0.986”. That is fine.

It might look rather odd and arbitrary to specify a slot length of such an odd number but hopefully you can see that it makes fabrication easier.



Here you see the bar after I drilled my series of holes. This technique is commonly called “chain drilling”.

“Doc” of metal_shapers said that he has done a lot of chain drilling with the holes spaced $\frac{3}{4}$ of a diameter apart. I can see how this could work if the drill is large enough in diameter not to bend into the adjacent hole. The advantage with this spacing is that there is no web to cut out.

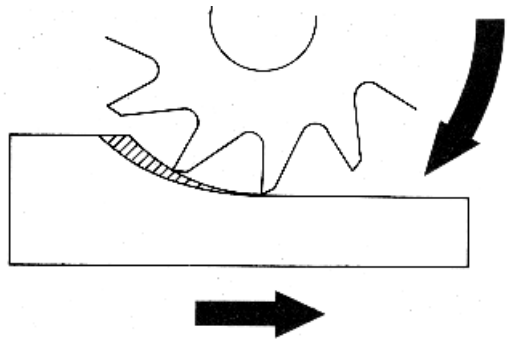


The red line shows where my 0.25” end mill will cut. I will feed in to the finished length of 0.986”. This is a bit tight on the sides because the finished width is 0.260” giving me only 0.005” for my finish cut. Alignment of the holes and the end mill is critical.

It is my understanding that the best type of end mill to use here is called a 2 flute end mill in the USA. In the UK it is called a slot drill.

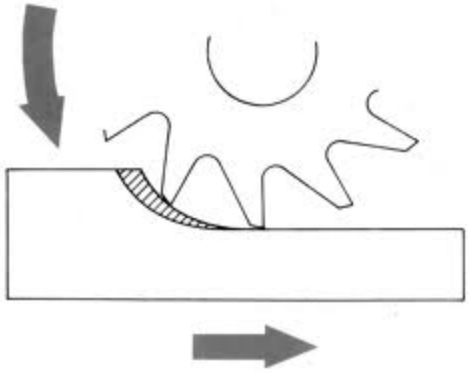
I only own a 4 flute 0.250” end mill and it did work OK. However, the next time I place an order at Enco, I’ll order a 2 flute version.

Before I proceed with the finish cut, let me present two terms.

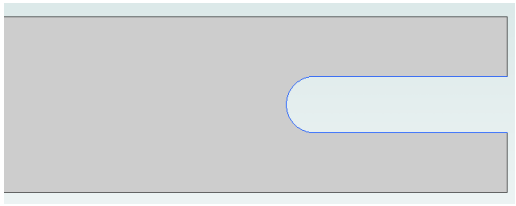


Here the material is fed into the cutter as the cutter is pushing against it. This is called “*conventional milling*”. It tends to prevent the cutter from grabbing material and digging in too deep.

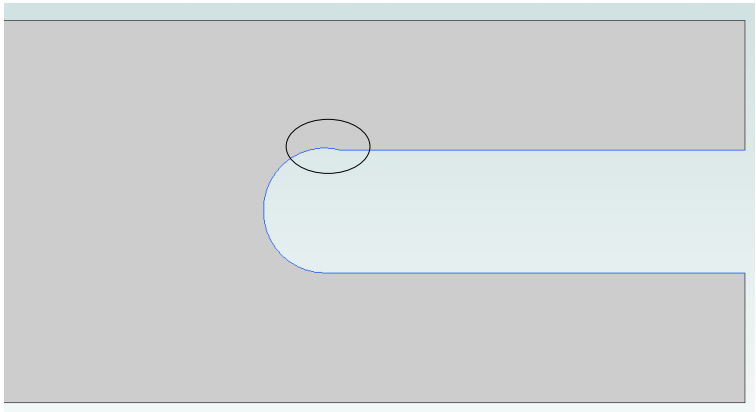
The opposite of conventional milling is “*climb milling*”. The material is being fed into the cutter and the cutter is trying to climb ***onto*** the material and pull more in. This can cause the mill table to violently slam back and forth in the backlash area of the leadscrew if the cut is too deep.



There is some disagreement over the best way to feed the end mill in the slot for the finish cut. Dave of metal_shapers says to climb mill because it will tend to push the cutter away from the material and prevent it from cutting too deep. I have always tried to use conventional milling in order to maintain a smooth table feed. I suggest you try it both ways on your machine and pick the one that works best.



Here is the slot after I going along the center line and stopping at 0.986”.

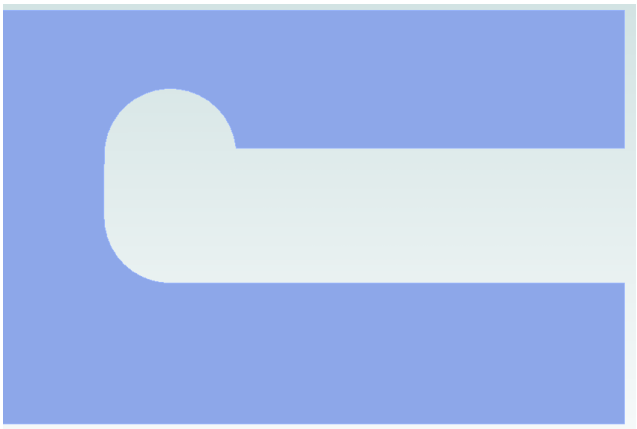


With the end mill all the way to the left, I feed towards the far wall (up in this picture) until I reach the finished side of the slot. You might be able to barely see the start of the cut inside the circle.

This is where you appreciate specifying the slot from the center line. The slot is 0.260” wide but is also ±0.130” from the center line. So if I’m using a Digital Read-Out (DRO), I set zero at my centerline, dial in the tool offset of 0.250”, define the back face as my cutting surface, and feed in to 0.130”. It is easier to do than say.

The end mill is then fed out of the slot using conventional milling. The last step is to change the tool offset so the opposite face is defined as the cutting surface. Then feed to -0.130” and feed back into the slot to 0.986” again using conventional milling.

The result should be a nice formed slot with a radius that matches the end mill. It does take some practice.



A problem I had a lot when starting out and still have occasionally is that I turn one of the feeds the wrong way. The result is something like this. I'm sure you can furnish your own monologue to go with this picture.

Acknowledgements

Thanks to Dave of metal_shapers for explaining how he would mill the slot. Thanks to Corey of valleymetal for pointing out that there could be confusion between conventional and climb milling. Thanks to "doc" of metal_shapers for presenting an alternate hole spacing for chain drilling.

I welcome your comments and questions.

Rick Sparber
Rgsparber@aol.com
Rick.Sparber.org