DOVETAIL CUTTER EXPLANATION

I wanted to make a 60degree dovetail cutter so I could build a micrometer boring head. Hopefully, I'll be able to also add a trip pin feed so that I can face under power also.

I had some 1/2" drill rod available, but I only had a 1/4" and 1/2" collet for my old Gorton mill.

See Cutter Step 1.jpeg

I decided to make a 1/4" shank cutter, but with the shank axis 1/8" eccentric from the bar stock axis. This would provide the largest possible dovetail, given the bar stock size and my work holding. I mounted the rod stock 1/8" eccentric in a 4-jaw chuck mounted on my mill. I made a 1/2" to 3/4"-16 adapter to permit this. I carefully took many light cuts with a honed tool to reduce the diameter to create the shank, holding the lathe tool in the mill vise. I stopped 2 or 3 thousandths large, and finished up with a file to within 1/2 thousandth of 1/4" diameter over the length of the shank. A tool post grinder would be nice for this, but I haven't built one yet.

See Cutter Step 2.jpeg

I then mounted the 1/4" shank in the 1/4" mill collet. I machined the 60degree angle on the end, taking light cuts with a sharp tool. I was able to swing the head of the mill to create the angle. I don't a have a compound set up for mounting on the mill table yet. This is an interrupted cut when made eccentric to the rod stock so don't hurry things.

See Cutter Step 3.jpeg

I then removed the cutter from the mill and clamped it in the vise. I split the business end with an end mill. Get as close to center as possible, but err on the side of taking off a _little_ too much or you won't end up with the cutting edge as the furthest point from the cutter axis. The most eccentric "point" of the angled bit should end up as the tip of your tool. BE CAREFUL when splitting the point to leave the correct half of the cutter intact, or you'll have to mill with the spindle in reverse!!

See Cutter Step 4.jpeg

Lastly, I ground the clearance angle on the end. Note that it's a compound angle, providing clearance both radially and circumfrentially. I just eyeballed it, cleaned it up a bit with a fine file, and then honed both faces a smidge with a diamond stone.

See Cutter Step 5.jpeg

I heat treated it with a plumber's torch, a Bernzomatic propane job. I dumped some 20 weight motor oil into a tuna can, heated the cutter's business end to a nice cherry red (observed with the shop lights _out_), then dunked it into the oil. I let it stay in the oil for a while until I was sure it was below any useful tempering temperature. Then I fished it out, re-polished the flat face of the cutter, and heated the shank a bit more gently. I held on to the middle of the shank with channel locks. When the colors started to run across the polished face (it'll be pretty subtle, observed with the shop lights "**ON**". I watched closely and dunked it back into the oil when I saw the straw-yellow just slip over toward brown. This might be a little soft for general work, but it has a small shank so I want a bit of toughness, and mostly I'll be cutting aluminum or free machining steel with it, so I think it'll work out OK.

So far, I've tested it on a piece of 6061 AL.It gums up a bit (0 degree rake and sticky AL contribute) but does make a dovetail cut. I havent made the actual boring head yet, so can't comment on the long-term durability of this cutter. Remember to feed a single flute cutter SLOWLY. There is only one cutting edge. The feed should be half that used for a 2-flute cutter of the same diameter when turning at the same speed, so as to keep the chip load the same.

Obviously, any number of special shapes can be made this way with a little thinking. Also, note that I was able to accomplish all machining on my mill. That's probably not really good practice (make lathe parts on a LATHE!)but it did work ;)

Kevin W. Olson