TRAMMING A RONG FU MILL/DRILL

by

Norm Berls

Point of View

- I’m a novice and, this is a voyage of discovery.
- As I encountered problems, had to engage in a series of supporting minor projects to achieve the main goal.
- Presentation contains many details that I find interesting.
- Hope they interest everyone.

The Rong Fu Mill/Drill

- 1.5 horsepower motor.
- Multiple speeds via compound belt drive.
- Head swings side to side and moves up and down on smooth vertical column.
- Work table has T slots and moves forward-backward, left-right on dovetail ways.
- Spindle will hold collet or drill chuck.
- 0.5 inches practical limit of cutting tools.
- 0.050 inches practical limit for depth of cut.
- Circular column makes head prone to twisting.

HISTORY AND CONTEXT

- Bought mill at Wholesale Tool 30 years ago.
- Cost: $1,300
- At the time was the lowest quality and cheapest available.
- Only recently have begun attempting precision work.
- Discovered mill was not trammed… actually, discovered what tramming meant.

Why tramm?

- Goal: precision metal cutting and drilling.
- Want to hold tolerance of 0.001 inches or less.
- Often surfaces on a workpiece must be perpendicular to each other.
- Solution: Set spindle of mill exactly perpendicular to the work table. This is called tramming the mill.

About the Mill/Drill

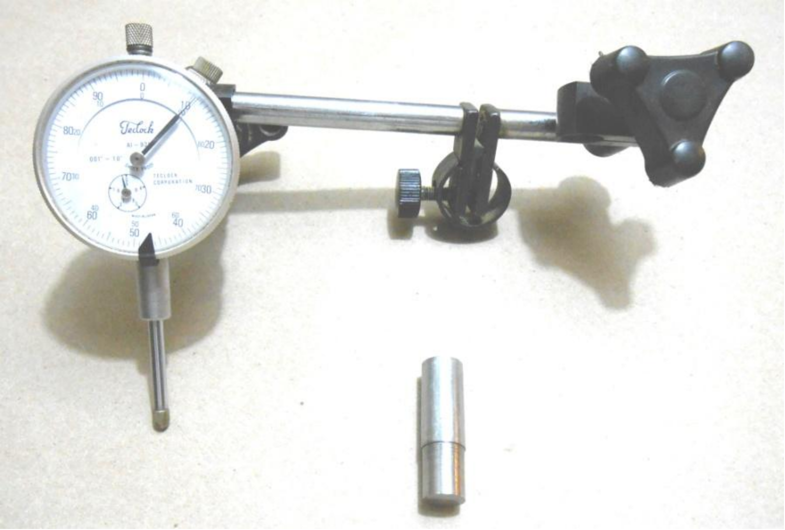
- Mill/Drill base and column appeared to be one piece.
- Loosening four bolts at base of column revealed that base and column are separate.
- Contact between column and base had been puttied and painted over.
- Here the column has been broken loose and is standing above the base.

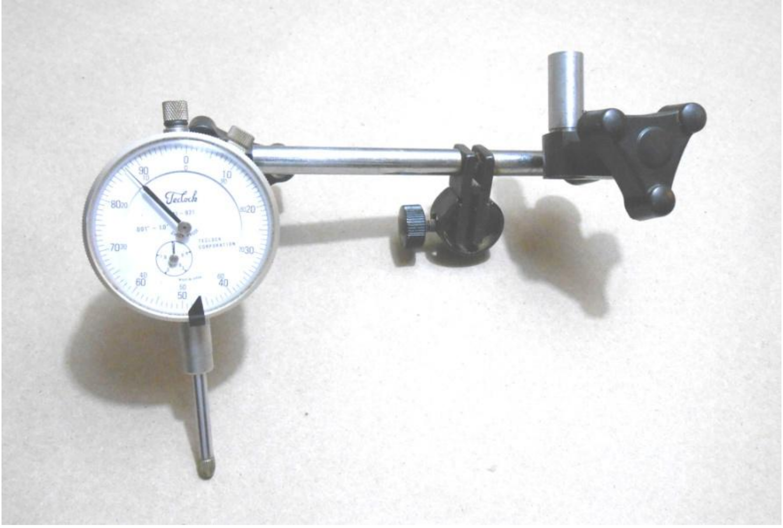
How To Tramm

- Placing carefully selected shims between column and base should tramm the mill.
- Example tramm procedure available on Internet.

Tramming with a dial indicator

- Made an adaptor to fit indicator into ½ inch collet.



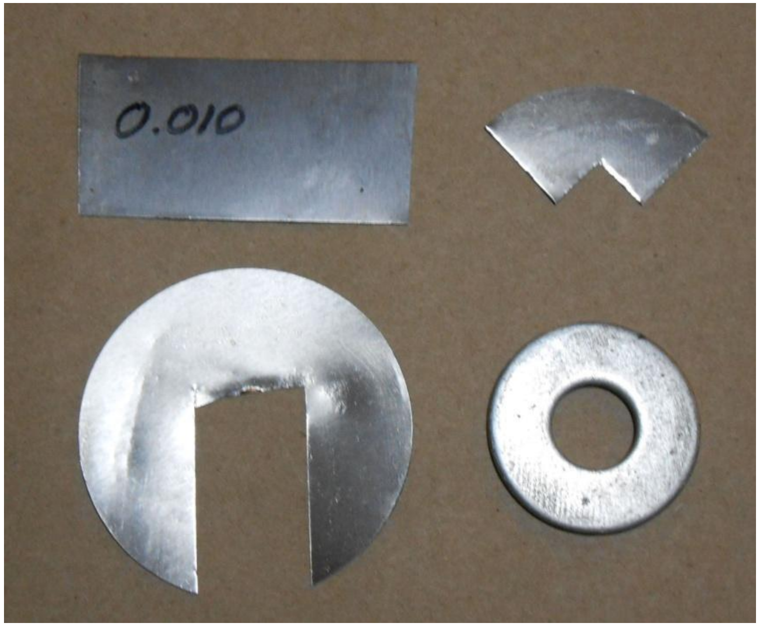


Tramming with a dial indicator

- Leveled the work table with a machinists level and shims under the feet of the mill.
- Freed the belt drive on the spindle.
- Spindle could freely rotate to any position.
- Fixed dial indicator in spindle.
- Measured distance from the dial indicator to the work table in left-right (X) and front-back (Y) directions.
- When the X distances and the Y distances are all equal, the mill is trammed.

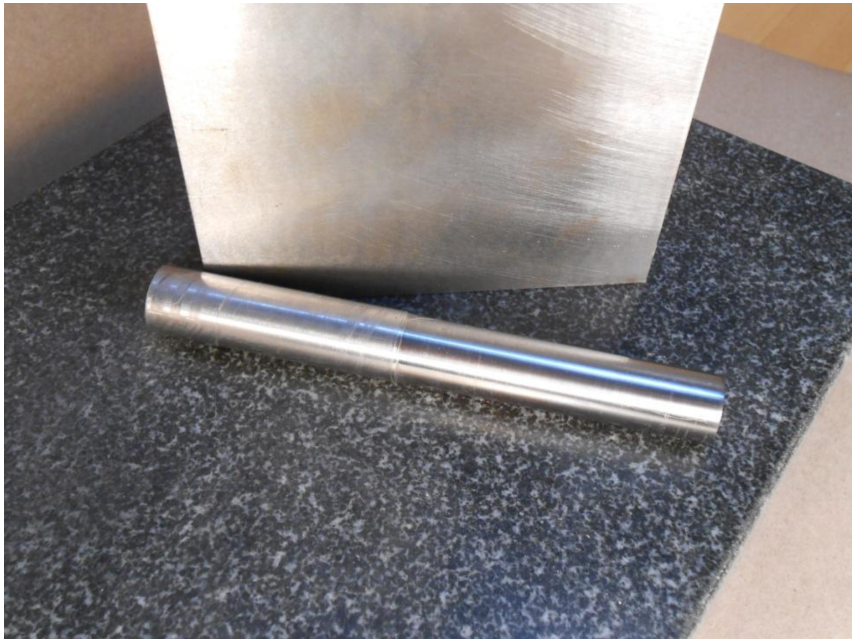
Shimming the mill

- First tried standard steel shims. No success.
- Tried multiple alternatives over period of one man month. No success.
- Tried torque wrench. Sequenced and incremental torquing. No success.
- Best result was both Y distances and +X were equal. -X was not equal.
- Conclusions:
  - 1) Work table is not flat
    (error 0.002 – 0.004).
  - 2) Column is probably not straight (error 0.002).
  - 3) Need new approach to tramming.



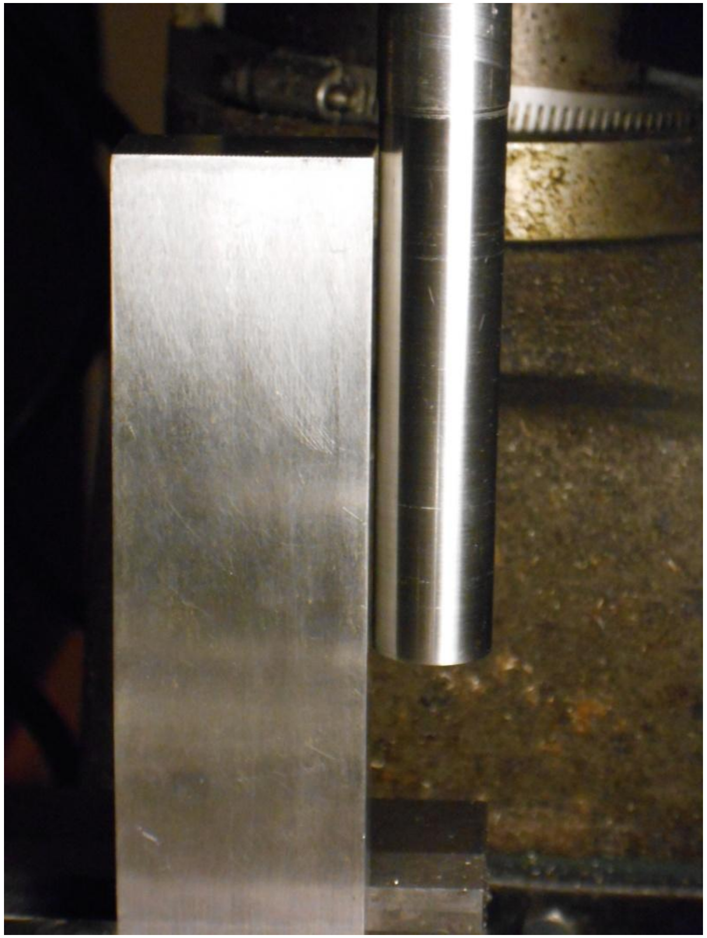
Made a mandrel

- Made a ¾ mild steel mandrel with my lathe.
- Turning necessary to remove runout.
- Polished mandrel with silicon carbide abrasive paper backed by foam sanding block.
- Lathe ran at 2000 RPM while polishing.
- Silicon Carbide Abrasives: 150, 220, 400, 600 grit.



Approximate Tramming

- Mounted vise on mill and squared it.
- Clamped machinists square in vise.
- Mounted mandrel in splindle.
- Tried to shim so that mandrel was as close as possible to parallel with the square.
- Results unsatisfactory.
- Conclusion: tramm of this mill cannot be achieved with shims.

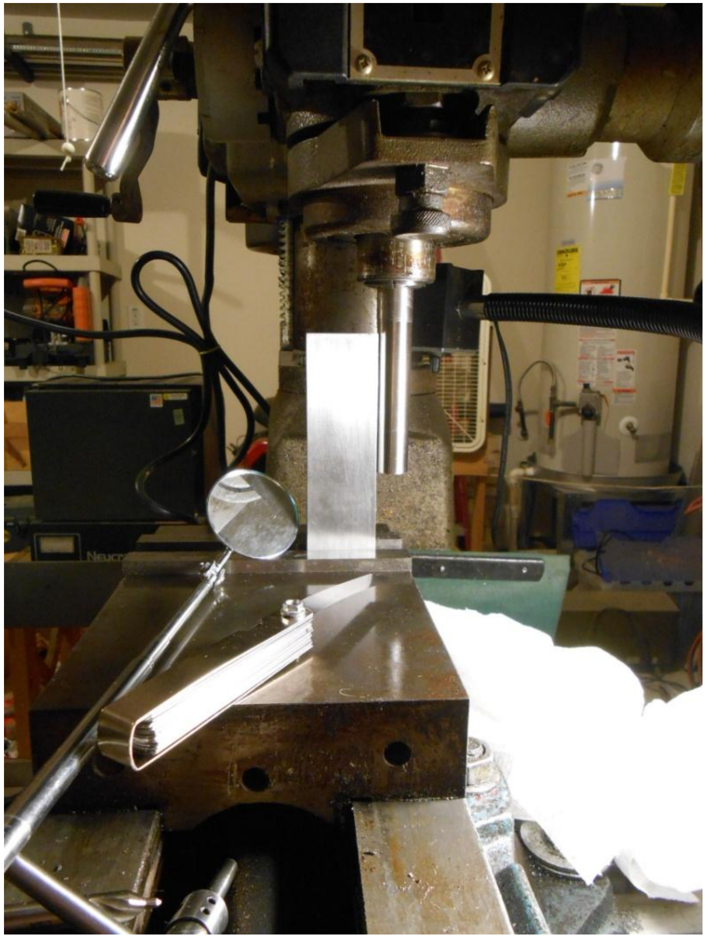


An alternative to shimming

- Cut four pieces of 5/8 all-thread rod 5 inches long each.
- Cleaned up threads with a die.
- Screwed rods into base of mill and locked them into place with a jam nut.
- Screwed second nut onto each rod.
- Set column on top of second nut.
- Screwed third nut on top of column.

Yet another attempt to tramm

- Set a light behind the mandrel and square to illuminate the gap between mandrel and square.
- Adjusted jam nuts to minimize and equalize light showing through gap.
- Checked gap with feeler gage.
- Smallest gage (0.0015) would not slide between mandrel and square… in both X and Y directions.
- Out of tramm by less than 0.0015 inches over distance of four inches… for this position of vise and mill head.
- Approximate Success!



Realities of jam nuts

- Preliminary X and Y positions achieved by adjusting pairs of jam nuts (left-right) (front-rear).
- Fine adjustments made at just one of the four corners at a time.
- Tightening top nut will shift tramm even when lower nut not changed.
- Final torque achieved by creeping top and bottom nuts towards each other.
- Conclusion: jam nuts are easier to control and work with than shims.

Retest Tramm

- After using the mill and moving the head, decided to retest the Tramm.
- Tramm found to be in error in both X and Y dimensions.
- Purchased HSS tool steel rods in multiple sizes.
- Used ¾ inch rod to re-tramm mill.
- No light showed between square and HSS rod.
- Tramm was apparently perfect.

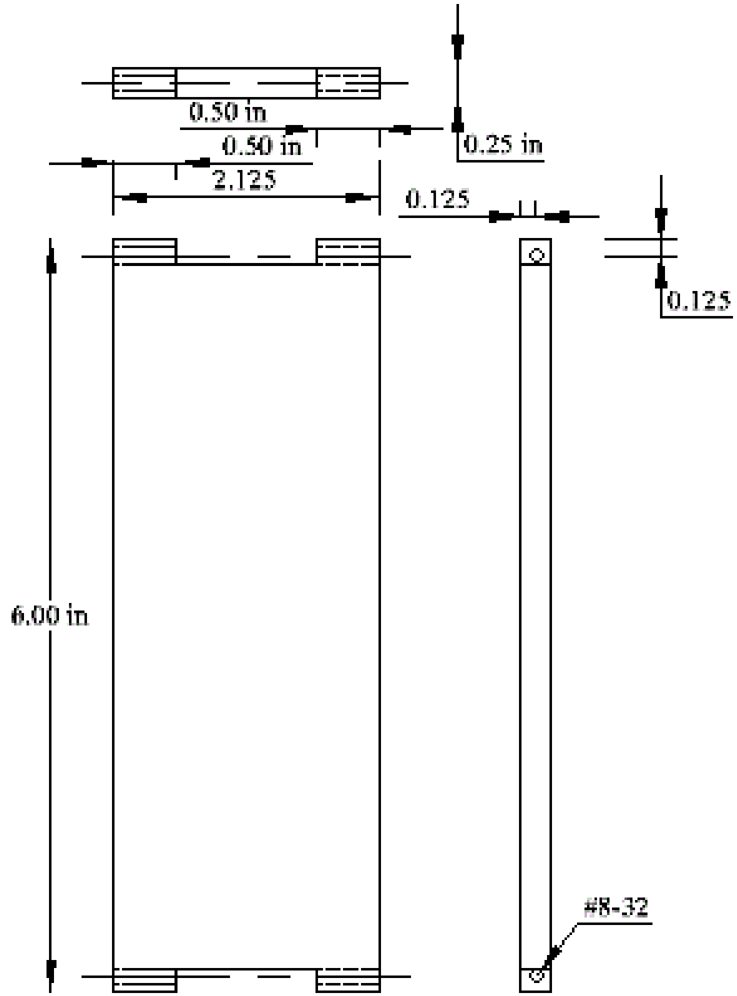
Future Things To Try

- Spindle square available on Internet for circa $100.
- 5/8 threaded rods for jam nuts were almost too short at 5 inches… 6 inch rods would be better.
- 5/8 rods have to be threaded at 11 TPI (NC) to match mill base but, upper part could be threaded at 18 TPI (NF) to give finer control.

Tramming Reconsidered

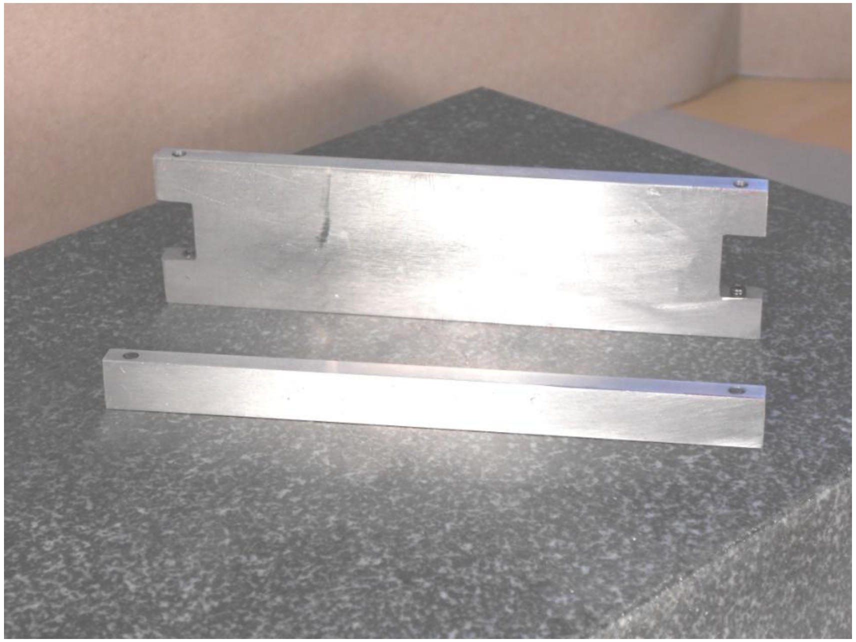
- Every time the head is moved up or down to switch between collet and drill chuck, the tramm of the mill is in question.
- What matters is the orientation of the spindle to the workpiece.
- Theory: If you can’t exactly tramm the mill, tramming the workpiece will work just as well.
- Adjustable parallels would allow tramming of workpiece in a vise.
- Precision mandrels would allow workpiece to be set parallel to the spindle.

Adjustable Parallel Drawing



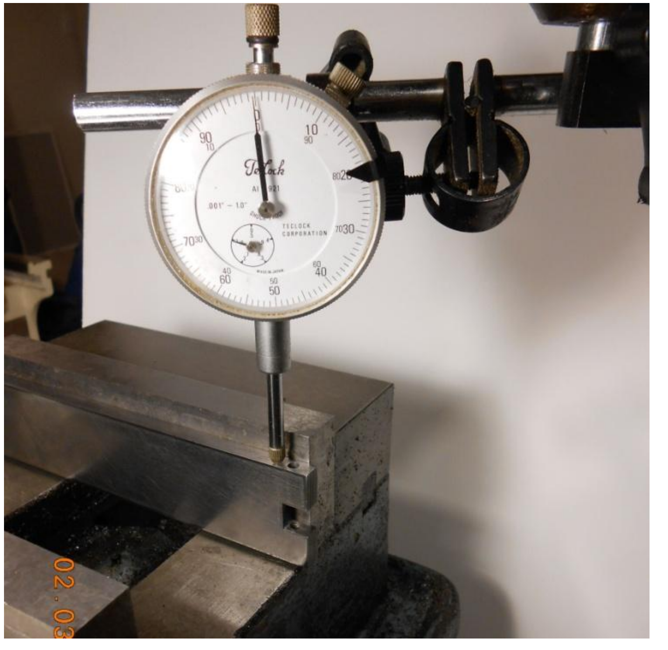
Adjustable Parallels

- Parallel sets in made in 1/8 inch increments.
- Sizes range from 0.5 to 1.625 (shown).
- Made from ¼ inch precision ground low carbon steel (1018).
- Adjustable parallels involve a vertical set screw in either end.
- ½ inch by 8-32 set screws.
- Either end of parallel can be raised or lowered.



Setup with adjustable parallel

- Insert adjustable parallel in jaw vise.
- Clamp dial indicator in spindle.
- Fine adjust spindle to make dial indicator read zero.



Setup with adjustable parallel

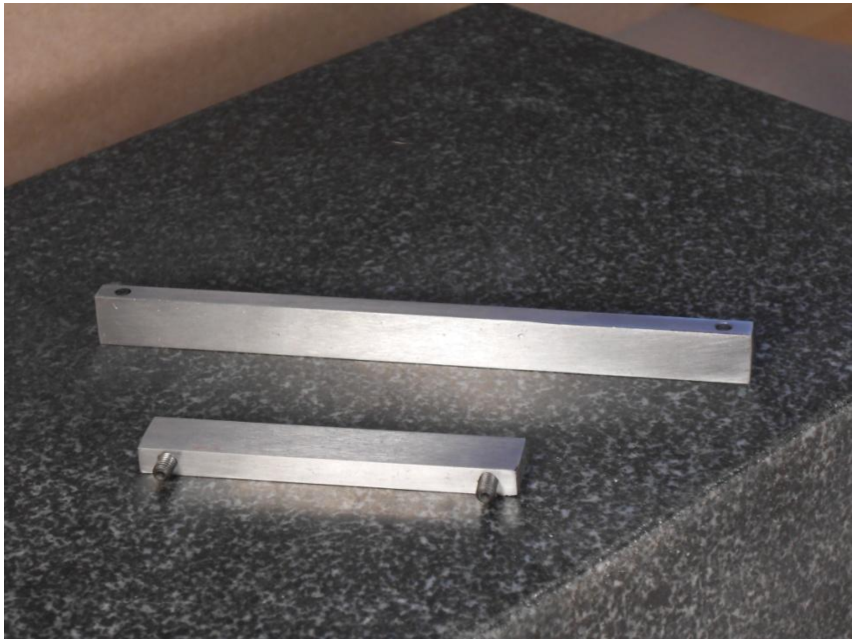
- Move vise so that dial indicator is at other end of parallel.
- Parallel is 0.065 inches low at this end.

Setup with adjustable parallel

- Insert allen wrench in set screw.
- Adjust parallel upward 0.065 inches.
- Dial indicator shows parallel is trammed with respect to spindle.
- Used ½ inch parallel to setup cut of 1.625 inch parallel.
- 1.625 parallel height varied 0.0002 to 0.0003 inches end to end!

Yet another parallel

- First set of parallels were 6 inches long to fit big vise.
- Needed another set 3.2 inches long to fit a smaller tool makers vise.



Why two vises?

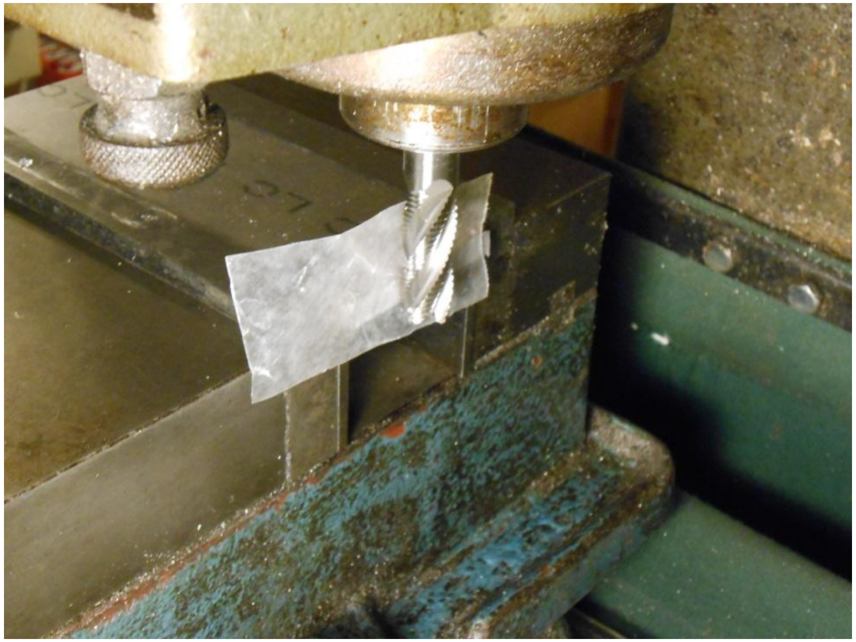
- I prefer to use a vise to hold a workpiece. Faster setup. Less complicated setup.
- Two vises allow workpiece to be oriented in direction of either X or Y axis of work table.
- Here a large 6 inch vise slot is oriented parallel to X axis of work table.
- And a small 3.0 inch vise is positioned inside the jaws of the 6 inch vise.
- The 3.0 inch vise slot is oriented parallel to Y axis of work table.

Why two vises?

- Power feed allows continuous movement of workpiece.
- Stalled end mill leaves marred finish.
- Power feed makes milling easier and yields higher quality finish.
- Power feed works only in X direction of the mill.
- Two vises allow workpiece to be oriented to make maximum use of power feed.

Making an adjustable parallel

- Used hack saw to cut workpiece to near final length.
- Squared both ends of parallel.
- Used wax paper as feeler gage to setup end mill to touch workpiece.
- Moved workpiece to rear to free wax paper.
- Adjusted workpiece for a 0.010 cut.
- Made the cut.
- Same for other end.
- Used a steel ruler to measure workpiece.



Yet another view of end squaring.

Milling a parallel to correct height

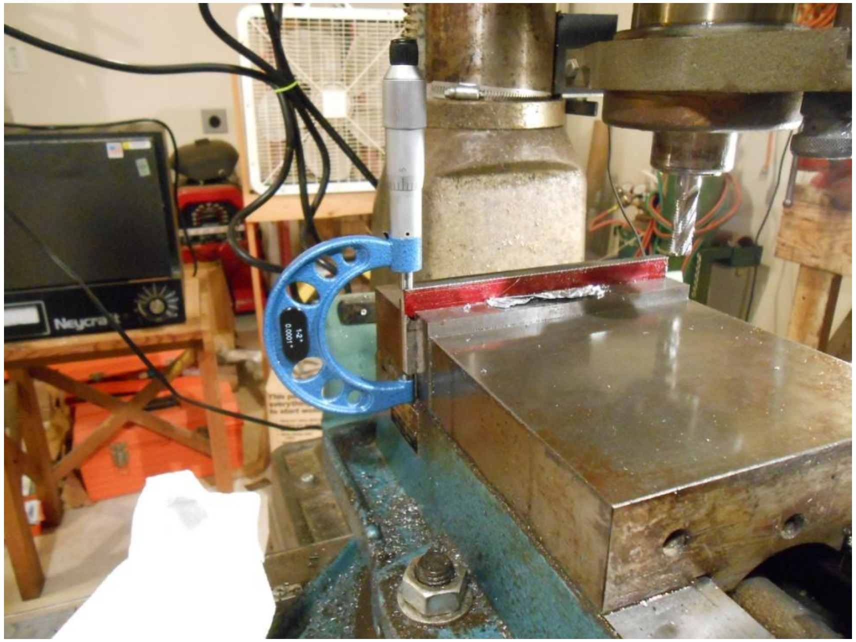
- Positioned ½ inch end mill at top of work piece.
- End mill just touched top.
- Moved end mill to side and lowered it 0.050 inches.
- 0.050 is about the maximum depth mill will cut.

Milling a parallel to correct height

- First cut underway.
- Power feed is moving workpiece under end mill.
- Using ½ inch roughing end mill.
- Next lower end mill by another 0.050 inches.
- Repeat until most of metal removed from workpiece.

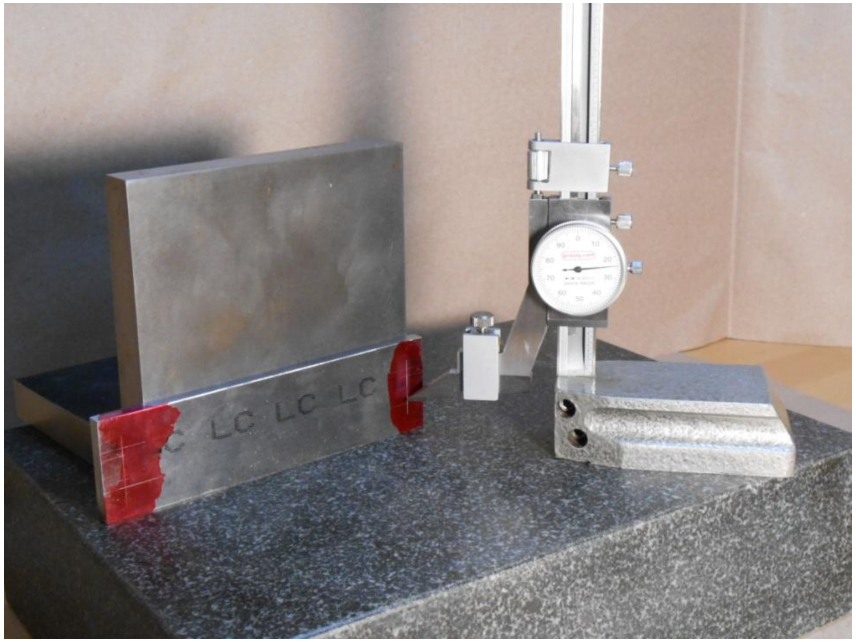
Milling a parallel to correct height

- When remaining metal < 0.050 inches, start measuring height with micrometer.
- Limit cuts to ½ remaining metal.
- 0.050 remains: cut 0.025
- 0.025 remains: cut 0.013
- 0.012 remains: cut 0.006
- When 0.001 remains, finish with silicone carbide abrasive paper.



Notch cutting an adjustable parallel

- After parallel cut and finished to correct length and height.
- Need to cut rectangular notch in either end of workpiece.
- Paint workpiece with Dykem Red.
- Use granite surface plate.
- Set workpiece against block square.
- Scribe cut limits with height gauge.



Milling a notch

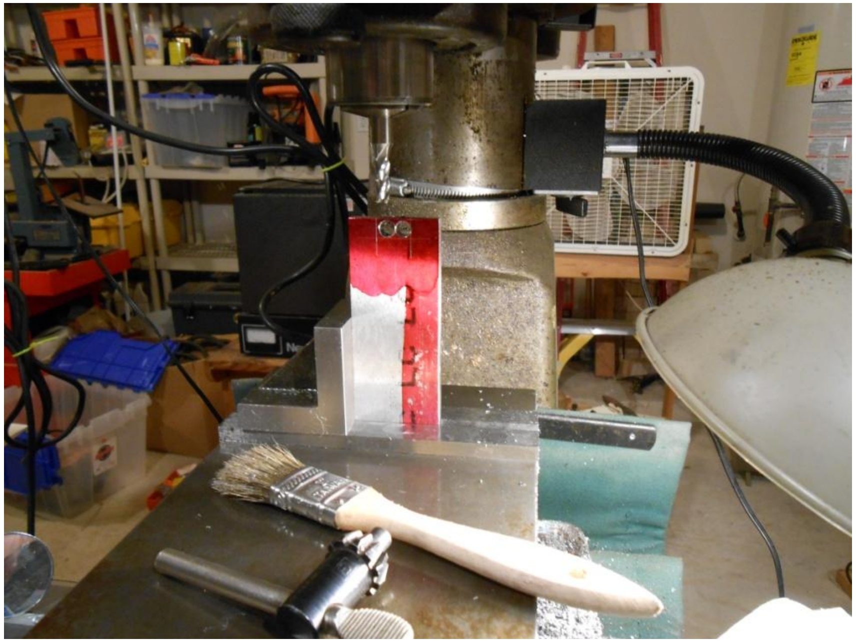
- Setup workpiece.
- Drill two holes in area to be removed.
- Drill small dimple with centering drill.
- 118 degree drills will drift without lead dimple.

Milling a notch

- Prefer to remove as much metal as possible with drill.
- Drills are cheaper than end mills.
- One hole drilled, need one more.

Milling a notch

- Setup to mill notch with ¼ inch end mill.
- First cut slot through center.
- Work on left side of slot from front to back… to make sure cutting edge of end mill advances into the cut.
- Work on right side of slot from back to front… to make sure cutting edge of end mill advances into the cut.



Setup to drill set screw hole

- Setup work piece on top of adjustable parallel.
- Object: drill set screw holes exactly vertical.
- Using optical center to position chuck over scribed center.
- Discovered $200 optical center was off center in both X and Y.
- Tested optical center and have example showing inaccuracy.
- Needed new method to center drill bit.

Setup to drill set screw hole

- Put HSS point from scribe into chuck.
- Centered it over scribed center using small magnifying glass.
- A tedious and somewhat inaccurate procedure.

Youtube demos edge finder

- Have had edge finder for 6 months but did not know how to use it.
- Found a demo on Youtube that showed how to use an edge finder.
- Changed to using edge finder for setups.
- Here drill chuck is in motion and edge finder has just kicked out to the left.
- Continuing to scribe parts on surface plate: it makes a nice position check against the edge finder.

Making a lead dimple

- Use center drill to dimple work piece.
- Small dimple should center up on scribe marks following positioning off edge of workpiece.
- Check with magnifying glass.
- Drill dimple deeper.
- 1/8 drill has 118 point angle. Will drift without substantial lead dimple.

Drilling a 1/8 inch hole.

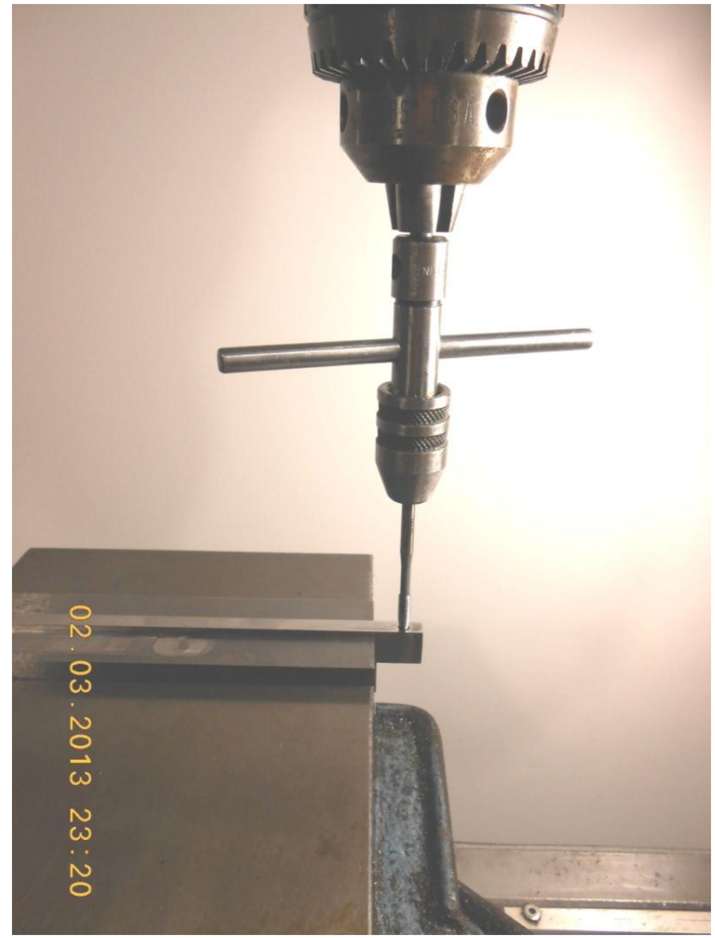
- Drilling 1/8 inch hole.
- Drilled through one side only.
- Flipped over workpiece to drill other side.

Drilling final hole before tapping

- #29 drill (0.136) required for 8-32 tap.
- Drill #29 hole through both top and bottom lead holes.

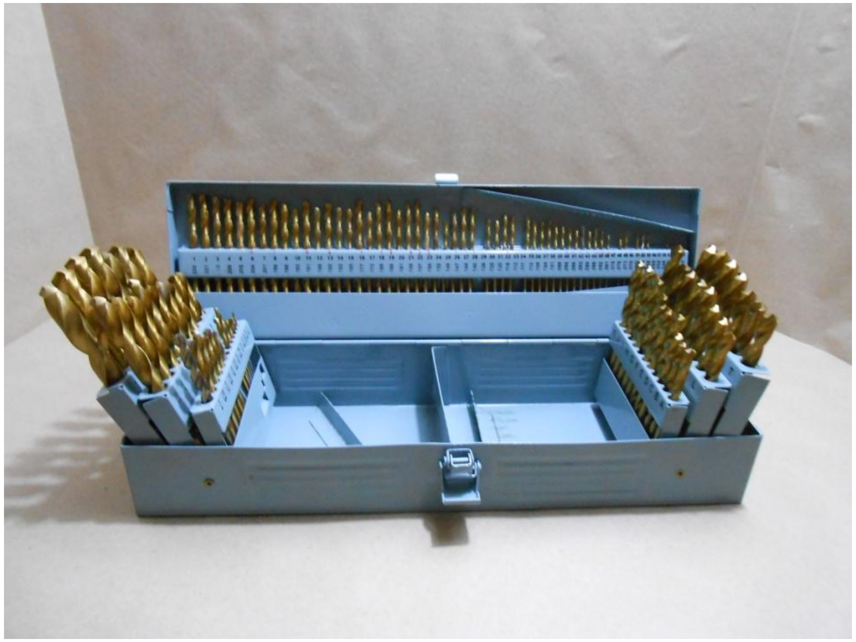
Supporting Drill Chuck Nice For Tapping.

- I like T shaped tap wrenches with a center post that fits in drill chuck.
- Less likely to break taps this way.



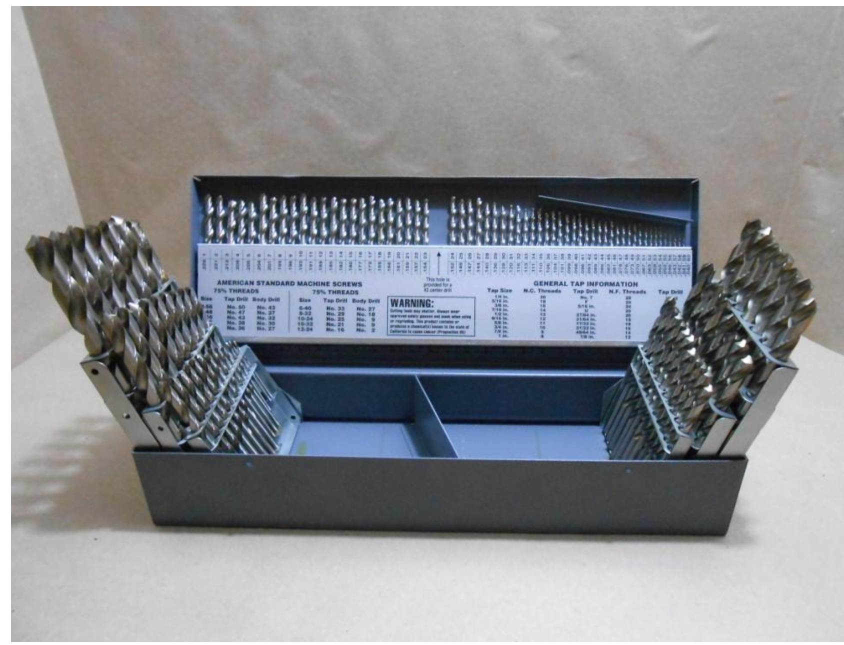
Tapping problems

- Attempt to tap 8-32 threads resulted in broken tap.
- Investigation revealed #29 drill was too small by 0.004 inches… cheap drills ($75).
- Only 10% of numbered drills correct. Most too small by 0.002 to 0.004 inches.
- Only 4 of 26 alphabetic drills correct. Errors of 0.006 and more.
- None of the fractional drills were correct.
- ½ inch drill was too small by 0.015.



Needed new set of drills

- Could not tell quality of drills in tool catalog.
- Needed to see the drills and measure them with micrometer to insure quality.
- Off to Rex Tools.
- Generic USA drills were nearly all within 0.0005.
- R drill was incorrect.
- Cobalt alloy… good for tool steel as well as mild steel.
- 135 degree point angle… will cut directly down without slipping off to side.
- $600.



Hard learned lessons about drilling

- Never trust the labels in a box of drill bits.
- Always measure drill bits with a micrometer before using them.
- It is easy to confuse drill bits when multiple bits are in the work area.
- Too small a hole leads to broken taps.
- Start holes too small and gradually drill them bigger.
- Use cheap drills for initial (smaller) holes.
- Finish with high quality drills or reamer.

Hard learned lessons about tapping.

- Run the drill through the hole to be tapped to insure chuck is centered exactly over hole.
- Use T handle tap holder secured in drill chuck.
- Do not expect to complete threads in one pass.
- Be sensitive to resistance. Stop and reverse when you sense serious binding.
- Taps can break during reverse movement out of hole. If you sense resistance, go forward again just a little, then reverse again. This will break off swarf that is still attached to the workpiece.
- Clean tap with paper towel or chip brush.
- Clean hole with pipe cleaner.
- Resume tapping and expect to have to back out again.

Forgetting Is Easy

- I tend to forget which way the work piece will move when the drive screw is turned… so…

A Handy Oil Can

- From Micro Mark tools for model makers.
- Intended for glue.
- Works well for oil.
- Here with Pennzoil 5W-20

Tapping Operations in Review

Case Hardening: The Plan

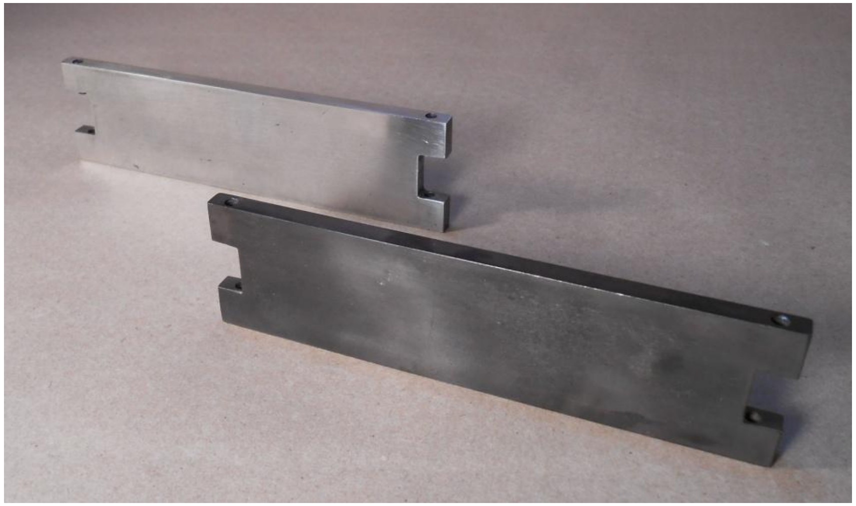
- The adjustable parallels are made from low carbon steel (1018).
- Case hardening would help protect the parallels during hard usage.
- Internet formula for case hardening mixture:
  - 13 parts Carbon
  - 3 parts BaCO3 (Barium Carbonate)
  - 2 parts NaCO3 (Sodium Carbonate)
  - 1 part CaCO3 (Calcium Carbonate)
- Cover part in compound and seal in stainless steel foil pouch.
- Heat to 1750 F and hold there for 8 hours.
- Quench in 10% brine solution.
- Re-polish and protect with paste wax.

Setup For Case Hardening

- Neycraft electric furnace.
- Fire bricks
- Long pliers
- Weldors gloves
- Bucket for 10% brine solution.
- Work done outside because of carbon monoxide considerations.

After Quenching

- Case hardening left hard tarnish on parallel.
- Rotary wire brush removed some but, not all tarnish.
- 150 silicon carbide abrasive removes tarnish but takes a lot of work (30 minutes +).
- Best removal procedure involved:
  - 150 silicon carbide abrasive
  - Flitz metal polish
  - Steel block
  - Pressure.
- Finished polishing with 150, 220, 400 and 600 grit silicon carbide.



After Polishing

- Polishing the adjustable parallels was a lot of hard work.
- Purchased a 4x36 belt and 8 in. disk sander.
- Finished polishing the parallels by hand and 3 hours later the needed abrasives for the belt-disk sander arrived.

Did It Work?

- File that readily cut 1018 skated across case hardened 1018.
- Tested untreated 1018 vs. case hardened 1018 with simple hardness test files.
- Untreated 1018 tested out to < 40 Rockwell.
- Case hardened 1018 tested out to around 50 Rockwell.

Result: A Box of Adjustable Parallels

- 32 adjustable parallels
- 14 pounds
- 5-6 months work
- $150 worth of steel
- $600 worth of drill bits

Conclusions

- Mill / Drill is stable enough and reliable enough to justify installing a DRO on it.
- Use of the adjustable parallels will not be an impediment because, even on a perfectly trammed mill, you still have to do the setup correctly or all is for naught.
- May consider a coolant system in the future.

Where The Tools Came From

Source	**Tools**
Wholesale Tool	Rong Fu mill/drill
Brownells	Neycraft furnace.
MSC	Big vise, surface plate
Grizzly	Small machinist vise, height gauge
Northern Tool	Cheap drill bits
Rex Tools	High quality (USA) drill bits
Micro Mark	Oil Can, Squares
Home Depot	1/8 inch drill bit
Enco	End Mills, Ground 1018 Steel Stock and most other tools

The End