

## Eyepiece Retainer – Manufacturing Procedure

- 1) **Grab a piece of  $\varnothing 1.0''$  OD 360 brass round bar with a length of 1.5"-12". Cut a piece from larger stock on bandsaw if necessary. (9:51)**

Tools used: Combination square

### Lathe Operations:

Mount stock in a 1" collet in a 5C collet chuck on the lathe with 1" stick-out.

- 2) **Face one side to clean. (The stock piece may have been used before you, so you may need to remove the remnants of the previous machining operations.) (11:39)**

Tools used: 6" rule, HSS turning tool, digital readout

- 3) **Spot  $\varnothing .56$  hole. (12:25)**

Tools used: #4 HSS center drill, drill chuck

- 4) **Drill  $\varnothing .56$  hole .44" deep. (12:52)**

Tools used: 9/16" HSS twist drill, drill chuck

- 5) **Turn major diameter of 3/4-16 UNF 2A threads .44" from end. (15:37)**

Tools used: HSS turning tool, digital readout, 0-1" micrometer

- 6) **Cut  $\varnothing .65''$  X .25" wide thread relief groove .2" from end of part. (17:19)**

- Visually align right side of part-off tool to end of part using 6" rule as a straight edge, zero digital readout, position right side of tool .2" from end of part, feed tool in X axis until it lightly touches off on major diameter for 3/4-16 threads, set digital readout to actual measured value of major diameter, plunge to  $\varnothing .65''$ , retract tool, move tool to .325" from end of part with carriage handwheel, plunge to  $\varnothing .65''$ , retract tool.

Tools used: 6" rule, part-off blade with carbide insert, digital readout, dial caliper

- 7) **Cut .04" X 45° external chamfers on major diameter of 3/4-16 UNF 2A threads. (20:23)**

Tools used: HSS chamfering tool, digital readout

- 8) **Break inside edge .015" max. (21:20)**

Tools used: HSS chamfering tool, digital readout

- 9) **Cut 3/4-16 UNF 2A threads. (21:38)**

- Visually align vee form of carbide threading insert to part using center gage, feed tool in X axis until it lightly touches off on major diameter for 3/4-16 threads, set digital readout to actual measured value of major diameter, set quick change gear box on lathe to cut 16 TPI threads, position tool at the start of the threaded section and dial in a cut of .010" on the X axis using the DRO, engage the halfnut at the appropriate position indicated by the leadscrew chasing dial to take a first pass on the threads, stop the tool inside the thread relief groove, double check the lathe is cutting the correct TPI threads using a thread pitch gage, take successive cuts of .010" per pass to a diameter of approx.  $\varnothing .690''$  (.015" over the minor diameter), measure the pitch diameter using a pitch micrometer, *make sure tool is in the same X axis position where it took the last pass* and enter the measured pitch diameter for the X axis in the DRO, take successive cuts until the measured pitch diameter is between  $\varnothing .703''$ -.708".

Tools used: Threading tool with carbide insert, digital readout, center gage, thread pitch gage, 14-18 TPI pitch micrometer

- 10) **Part off at .19" length. (35:03)**

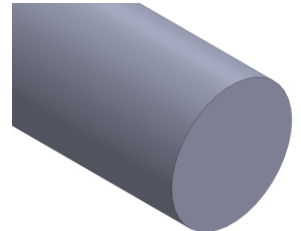
Tools used: 6" rule, part-off blade with carbide insert, digital readout

WATCH THE VIDEO!

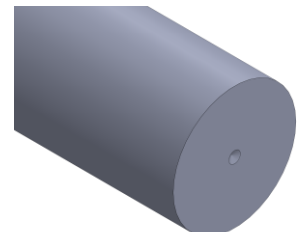


<https://youtu.be/Ap8o-toxKag>

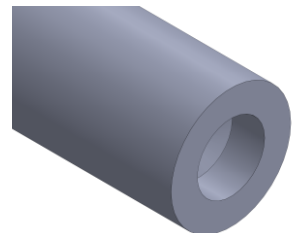
#2



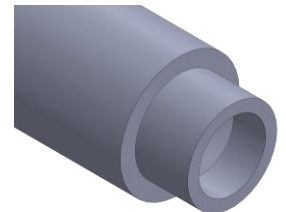
#3



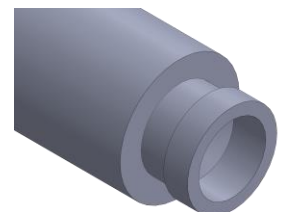
#4



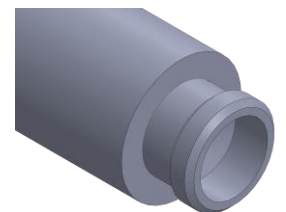
#5



#6



#7



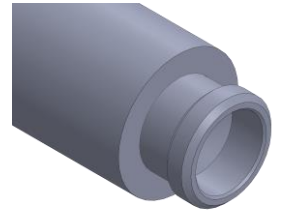
## Milling Machine Operations:

Install mill vise on table and ensure it is properly aligned to the table travel.  
Clamp part in vise on a 1 1/2" parallel with flat surfaces between jaws.

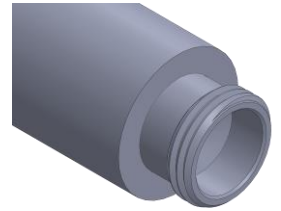
### 11) Cut .063" wide X .031" deep slot. (36:14)

Tools used:  $\varnothing$ 1.5" X 1/16" thick slitting saw, 1/2" R8 collet, digital readout

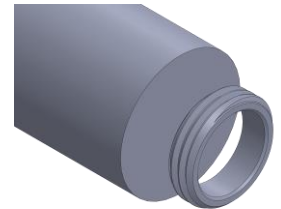
#8



#9



#10



#11

