LAB #8 - MACHINING EXPERIMENT II

MILLING

Objective(s): To understand the principles of milling operations and learn to operate a semi-automatic milling machine safely and analyze the surface finish and dimensional accuracy of the machined parts.

Equipment and materials:

1. Semi-automatic milling machine.
2. 6-inch digital caliper, analog micrometers.
3. High-speed steel end mill (0.5 in diameter), Fly cutter (2.5 in, 3 in, 5 in diameter).
4. Edge finder – 0.2-inch diameter.
5. Angle blocks – 10 degrees.
6. Aluminum 6061-T6 bar workpiece – 3 x 2 x 1 in.
7. SAFETY GLASSES.

Procedure:

A. Secure the workpiece in the table of the milling machine using the vice.

Preparation:
a. Thoroughly clean the surfaces of the milling machine vise and the parallels.
b. Secure the workpiece in the vise on the parallels, ensuring that about 1/2 to 3/4 inch of the workpiece extends beyond the right edge of the vise.

Machining:
a. Lower the end mill so that the cutting edge covers the entire depth of the workpiece.
b. Turn on the machine. Carefully touch the end mill to the rough sawn edge of the workpiece.
c. Using the calculated spindle speed (RPM) and feed rate, remove material in 0.010-0.015-inch passes until the edge is fully machined square, smooth, and ready for use.

Repositioning/Final setup:
a. Remove the workpiece and thoroughly clean the milling machine vise.
b. Place parallels under the workpiece to raise the surface to be milled above the level of the vise jaws.
c. Place the workpiece back in the vise without any overhang.
d. Tighten the vice securely.
e. Tap the workpiece lightly with a dead blow hammer to seat it properly. Do not retighten the vise afterward, as this could alter the position of the workpiece and the tightness of the parallels.

B. Machine Setup

1. Check the diameter of the cutter with a set of calipers.
2. Calculate RPM for cutting speed = 140 ft/min (end milling), 600 ft/min (fly cutting)

$$RPM = \frac{V(ft/\min) \times 12}{\pi D(in)}$$

3. When using the Milling machines.
   Set the speed of mill by adjusting the RPM knob while the spindle is ON and reading the speed on the analog meter. "H" and "L" scales represent high and low gear ranges.
   Note: do not cut at a speed lower than 500 rpm while in the high gear range.
   (Check with the lab instructor to verify your setup before you continue).
4. Calculate feed (in inch/min) for a feed per tooth = 0.003".

   Feed= feed per tooth x no. of cutter teeth x RPM =

   Note: *Power feed: speed is displayed on the digital readout. Adjust with the dial while moving away from your part.*
   If power feed is not available, use handles for manual feed.
   Note: *The direction of feed is optional on a vertical milling machine.*

C. Face milling the top surface – FLY CUTTING

1. Before starting the machine, be sure the cutting tool is clear of the workpiece. If not, lower the workpiece by lowering the table using the knee.
2. Raise the table using the knee to a SUITABLE work height. Lower the Quill(tool) until it touches the top surface of the part and lock it in place. Do not raise the table into the tool, instead only raise the table to a suitable work height before lowering the tool to the workpiece.
3. Zero Digital Read Out (DRO) of both the Quill and the Z-axis. Set the Dial on the knee to zero. Be aware of the slop or slag in the dial on the knee.
4. Move the table with the x and y axis handles to position the workpiece under the cutter.
5. Lower the table by 0.020", by turning the knee handwheel counterclockwise. (Use right hand thumb rule to determine the direction of the axes)
6. Move the table longitudinally so that the cutter is clear of the workpiece at either end.
7. Turn the machine on.
8. Raise the table to -0.030" (notice there may be "slop" or a gap in the dial on the knee) (1 graduation is 0.001") from zero so that the final thickness of the workpiece is 0.970"±0.002". Use multiple 0.010" passes and monitor the thickness of the workpiece.
9. Engage the power feed with the lever. Note: Table travels in the direction of the feed engagement lever.
10. After workpiece has been milled on the top surface, disengage power feed and turn spindle off.
11. Lower the table, remove the cutting and deburr part edges with a file.

D. END MILLING of slots

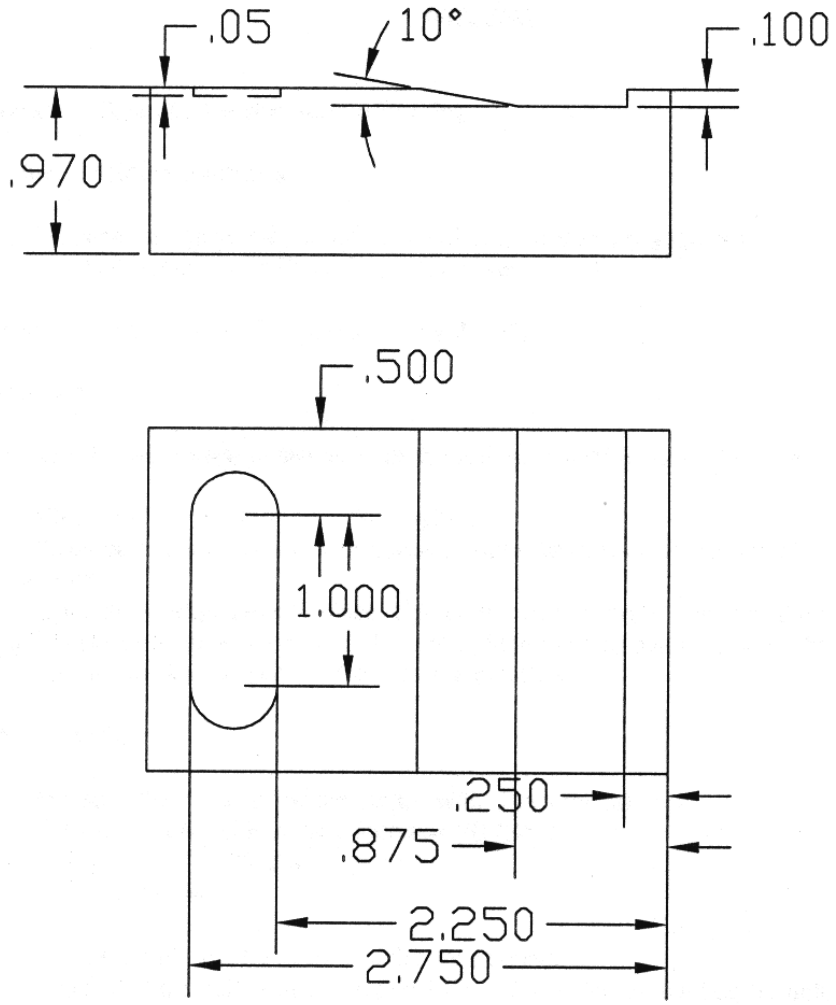
1. Replace the fly cutter with an edge finder (run the edge finder at 1000 rpm)
2. Touch off on the edges of the workpiece to set the origin of the workpiece (the lab instructor will demonstrate the procedure). Consider the RADIUS of the edge finder and offset the zero point accordingly.
3. Remove the edge finder and Jacob's chuck and replace it with a 1/2" end mill using an appropriate collet.
4. Using multiple depth passes, machine slot 1 as per the part print to a final depth of -0.100". Your roughing passes should be roughly 0.030-0.050" in depth with a finish pass of about 0.010". Your cutting speed should remain at 140 ft/min. Note: It is good practice to measure your part before taking the final finish pass to depth.
5. Make the slot 2 as shown in Figure using the same procedure in C.2-C.10. The depth of slot should be 0.050".

E. Slope machining

1. Use a correct angle block to set up the workpiece to be tilted at 10^0.
2. Use the 1/2" endmill to generate the slope using multiple path cuts.

Results:

1. Tabulated dimensions of the workpiece.
2. Calculate the cutting conditions used for each feature: cutting speed and feed rate.



All dimensions must be within tolerance of 0.002". The angle of the slope should be within tolerance of 0.1 degrees.

Table 1: Cutting conditions.

Operation	Tool	Feed direction (x, y, z)	Feed rate (in/min)	Cutting speed (rpm)
Fly cutting				
Elongated hole slot				
Straight slot				
Angle cut				

Table 2: Dimensions of machined part.

Desired Dimensions (in)	Actual Dimensions (in)
0.970	
0.050	
0.100	
0.250	
0.875	
2.250	
2.750	