

## **Machining Rules of Thumb**

### **Speed, Feed, and Depth of Cut**

1. Machining Speed in rpm =  $(CS \times 4) / \text{dia}$ , where CS = cutting speed in surface feet per minute =  
300-800 for aluminum  
200-400 for brass  
40-70 for high carbon steel  
30-80 for stainless steel
2. Drilling Speed in rpm =  $(CS \times 4) / \text{dia}$  where CS = cutting speed in sfm =  
300 for aluminum  
120 for brass and bronze  
90 for low carbon steel  
70 for cast iron  
50 for alloy steel
3. Counterboring / Countersinking Speed =  $1/3$  drilling speed
4. Machine Reaming Speed =  $1/2 - 1/3$  drilling speed
5. Facing Feed on a Lathe for Roughing =  $0.005-0.015'' / \text{rev}$ , for Finishing =  $0.003-0.005'' / \text{rev}$
6. Machining Feed on a Mill in inches per minute =  $(\text{chip load}) \times (\text{number of flutes}) \times (\text{speed in rpm})$  where chip load is  $0.005''$  for roughing and  $0.002''$  for finishing
7. Machine Reaming hand feed = two times drilling hand feed
8. When Hand Tapping, turn in  $1/4 - 1/2$  turn, then out  $3/4$  turn
9. When Cutting Threads with a Die, turn on 1 full turn, then back  $1/2$  turn
10. When Boring, drill  $1/32 - 1/16$  undersize, then bore
11. When Flycutting, take off  $0.025 - 0.100''$  per pass with a spindle speed of 1500-2000 rpm. Faster will leave a better surface finish, but use 1200 rpm or so with delrin and other plastics because otherwise the material will melt. Also, having a nice radius on the tip of your tool will improve surface finish.
12. Reaming stock allowance according to reamer diameter:  
 $1/32 - 1/8'' = 0.003-0.006''$   
 $1/8 - 1/4'' = 0.005-0.009''$   
 $1/4 - 3/8'' = 0.007-0.012''$   
 $3/8 - 1/2'' = 0.010-0.015''$   
 $1/2 - 3/4'' = 1/64-1/32''$   
 $3/4 - 1'' = 1/32''$
13. Depth of cut on soft steel = up to  $1/8$  inch

## Fasteners

1. Minimum thread engagement = diameter of screw (1.5 diameters is better)
2. #0 machine screw diameter = 0.060", add 0.013" for each # above  
e.g. diameter of a #6 machine screw = 0.060" + (6 x 0.013") = 0.138".
3. The diameter of a bolt head is 1.5 times the diameter of the screw
4. The diameter of a counterbore is .0313" (1/32) greater than diameter of the bolt head
5. The length of the bolt head is equal to the diameter of the screw
6. The practical strength limit for threads is 75%

## Miscellaneous

1. Single-cut files give the smoothest surface finish
2. Use chalk on a file for better action, less clogging
3. Always use cutting fluid when reaming except with brass or cast iron
4. Taper pins taper at .25" per linear foot
5. Use square keys for shafts < 6.5 in. diameter < use rectangular key
6. Large lead angles can cause chatter on a lathe
7. When grinding chip breakers, grind the divot first, then grind the face up to the divot, leaving 0.010" gap between divot and face