Machining Speeds and Feeds

Material Speed Factor - SFPM

SURFACE FEET / MIN – Hi-Speed Steel

(Carbide = 2 – 4 X Hi-Speed Steel)

A) Stainless Steel = 35 SFPM-take a cut / 0.015" min

B) Tool Steel = 50 SFPM-includes 4140

C) Cast Iron = 75 SFPM-get below scale-no lubricant

D) Mild Steel = 100 SFPM

E) Copper = 100 SFPM

F) Brass and Bronze = 200 SFPM

G) Aluminum = 300 SFPM

H) Wood = 3000 - 5000 SFPM

I) Plastic = 3000 – 10,000 SFPM

(1ft.=304.8 mm, 1 inch=25.4mm)

SPEED – Lathe – Mill *(metric = $SmmPM/\pi D$)*

1) RPM = 4 X SFPM / Dia." (r/min=318.3xm/min/D)

$$\frac{surface\ feet\ per\ minute\ \times 12"}{inches/revolution} = \frac{sfpm\ \times 12"}{\pi d} \cong \frac{sfpm\ \times 4}{d}$$

2) Threading Dies rpm = 0.20 (Calculated rpm)

3) Knurling rpm = 0.20 (Calculated rpm)

4) Reamer rpm = 0.25 (Calculated rpm)

5) Tap rpm = 0.25 (Calculated rpm)

6) Counter Bore rpm = 0.50 (Calculated rpm)

7) Counter Sink rpm = 0.50 (Calculated rpm)

8) Center Drill rpm = 0.65 (Calculated rpm)

9) Parting Tool = 0.65 (Calculated rpm)

10) Drill rpm = 0.75 (Calculated rpm)

11) Band Saw – 1.5 x SFPM

12) Grinding wheel surface speed =

60 mph or 88 ft/sec

Tap drill size

1) Fractional bolt & metric tap drill size =

Maj. Dia. – Pitch

2) Machine screw Maj. Dia. =

(screw # X 0.013) + 0.060

3) Basic Pitch Dia. =

Nominal size – (pitch X 0.6495)

FEED – square chip=best

Lathe - IPR

1) Roughing feed = 1/8 depth of cut

Depth of cut - ? (0.030" or more)

2) Finish feed = 1/6 depth of cut

Depth of cut – 0.007"-0.015"

Last two passes approx. =

3) Knurling feed = 0.015-0.040 "/rev

4) Parting feed = 0.002-0.005 "/rev

Mill - IPM

5) Chip load/flute = 1% diameter of cutter

(never < 0.001" or > 0.010 ")

6) Cutter feed =

(RPM) (#flutes) (chip load/Flute)

7) Maximum depth of Milling cut =

10% diameter of cutter

(2 flute=2 dia., 4 flute=2.5-3 dia. max)

Drill - IPR

8) Drill feed = 0.0075 (Dia.)(#flutes).

Reamer - IPR

9) Reamer feed = (Chip load/flute) (#flutes)

(Chip load = 0.001 - 0.004 "/flute)

10) Reamer Pilot hole

97% of finish size – never < 0.003" under

.97(reamer dia)

Grinder

11) Wheel cut = 0.002" or less typical (finish)

Feed = 0.010 – 0.025" typical

(Too fast feed causes large wheel radius)

12) Table speed

Too slow = more heat in material

Too fast = break down wheel

13) Dressing – 0.005 - 0.010" typical pass

Male thread – 1% undersize

0.020" grind stock when heat treating

Sq. key = 25% of shaft diameter