

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/πD*)

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

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

- 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