Milling Machine Cutting Speed & Feed Rate Practice Calculator

Table 60.1 – Recommended Milling Machine Cutting Speeds

Use this chart to determine cutting speed recommendations for different materials and cutter types. Then apply the RPM formula to calculate spindle speed.

RPM Formula: RPM = (SFM × 3.82) ÷ Cutter Diameter (in inches)

Material	HSS Cutter (ft/min)	Carbide Cutter (ft/min)
Alloy Steel	40–70	150–250
Aluminum	500–1000	1000–2000
Bronze	65–120	200–400
Cast Iron	50–80	125–200
Free Machining Steel	100–150	400–600
Machine Steel	70–100	150–250
Stainless Steel	30–80	100–300
Tool Steel	60–70	125–200

Cutting Speed Practice Problems:

1. Calculate the RPM for a 1/2" HSS end mill cutting aluminum at 600 SFM.
2. Calculate the RPM for a 3/4" carbide end mill cutting stainless steel at 250 SFM.
3. Calculate the RPM for a 1" HSS cutter machining cast iron at 70 SFM.
4. If your carbide tool runs at 400 SFM on bronze, what is the RPM for a 5/8" cutter?
5. BONUS: Explain how SFM affects tool life and surface finish.

Note: *Always start on the lower end of the SFM range when using new tools or uncertain setups. Increase gradually to find optimal cutting conditions.*

Table 60.2 – Recommended Feed per Tooth (High-Speed Cutters)

Use this table to find recommended feed per tooth (f_z) values for various cutter types and materials. These are used to calculate feed rate (IPM).

Material	Face Mills (in/mm)	Helical Mills (in/mm)	Slotting & Side Mills (in/mm)	End Mills (in/mm)
Alloy Steel	.006 / 0.15	.005 / 0.12	.004 / 0.1	.003 / 0.07
Aluminum	.022 / 0.55	.018 / 0.45	.013 / 0.33	.011 / 0.28
Brass & Bronze (Medium)	.014 / 0.35	.011 / 0.28	.008 / 0.2	.007 / 0.18
Cast Iron (Medium)	.013 / 0.33	.010 / 0.25	.007 / 0.18	.007 / 0.18
Free Machining Steel	.012 / 0.3	.010 / 0.25	.007 / 0.17	.006 / 0.15
Machine Steel	.012 / 0.3	.010 / 0.25	.007 / 0.18	.006 / 0.15
Stainless Steel	.006 / 0.15	.005 / 0.13	.004 / 0.1	.003 / 0.08

■ Feed Rate Practice Questions

Formula: Feed Rate (IPM) = $f_z \times Z \times RPM$

Where:
f_z = feed per tooth (inches/tooth)
Z = number of flutes
RPM = spindle speed

Feed Rate Practice:

1. A 4-flute HSS end mill runs at 1200 RPM with f_z = 0.003 in/tooth. What is the feed rate (IPM)?

2. A 2-flute carbide end mill at 3000 RPM uses f_z = 0.005 in/tooth. What is the feed rate (IPM)?
BONUS: If switching to 4 flutes with the same chip load, what happens to feed rate and chip thickness?

3. Explain why increasing the number of flutes (Z) at the same RPM and f_z increases feed rate.

4. A machinist doubles feed rate but keeps RPM constant. What effect does this have on surface finish and tool wear?

5. You are cutting aluminum with a 3-flute carbide end mill. SFM = 1200, Diameter = 0.5”, f_z = 0.004 in/tooth.
(a) Calculate RPM using Table 60.1 and formula.
(b) Calculate Feed Rate (IPM). Use the chart on page 1 to confirm SFM.

6. If tool diameter is reduced but SFM stays the same, how does that affect feed rate and why?

Answer Key – Speed & Feed Practice

Cutting Speed Section:
1. RPM = (600 × 3.82) ÷ 0.5 = **4584 RPM**
2. RPM = (250 × 3.82) ÷ 0.75 = **1273 RPM**
3. RPM = (70 × 3.82) ÷ 1.0 = **267 RPM**
4. RPM = (400 × 3.82) ÷ 0.625 = **2448 RPM**
5. Higher SFM increases cutting temperature, potentially reducing tool life but improving surface finish if properly cooled.

Feed Rate Section:
1. Feed = 0.003 × 4 × 1200 = **14.4 IPM**
2. Feed = 0.005 × 2 × 3000 = **30 IPM**

Bonus: 4 flutes → 60 IPM; chip thickness same, load distributed across more flutes.

3. More flutes = more teeth cutting per revolution → higher feed at same f_z.
4. Doubling feed increases chip load and risk of tool wear; finish quality decreases.

5a. RPM = (1200 × 3.82) ÷ 0.5 = **9168 RPM**

5b. Feed = 0.004 × 3 × 9168 = **110 IPM**

6. Smaller diameter requires higher RPM to maintain SFM; total feed can increase if f_z and Z remain constant.