

Machining Operations Cheat Sheet

Turning, milling, drilling: speeds, feeds, removal rate

ENGINEERING
CHEAT SHEET

Key equations

Quantity	Equation	Notes
Spindle speed	$N = V / (\pi D)$	Textbook form; V cutting speed, D workpiece or cutter diameter
Turning feed rate	$f_r = f N$	f in mm/rev
Turning time	$T_m = L / f_r$	L = cut length
Turning MRR	$MRR = V f d$	d = depth of cut
Milling feed rate	$f_r = N n_t f$	n_t teeth, f per tooth
Face milling time	$T_m = (L + 2A) / f_r$	$A = \sqrt{(w(D - w))}$ if cutter centered over work
Milling MRR	$MRR = w d f_r$	w = width of cut

Key takeaways

- Three conventional processes: turning, drilling, milling. Single-point tools turn and bore; multi-point tools mill, drill and saw.
- Three relative motions: primary (cutting), feed, and depth-of-cut setting. Roughing removes bulk; finishing sets size and surface.
- Up (conventional) milling: cutter turns against feed, tends to chatter, needs good clamping.
- Down (climb) milling: cutting force holds work down, longest cutter life, good for finishing; needs a rigid setup and avoids scaled surfaces like castings.
- Drilling: deep holes exceed about 10:1 length to diameter. Reaming, tapping, counterboring, countersinking, centering, spot facing follow.
- Check (mine) for the post's face-milling problem, assuming a centered cutter: $N \approx 637$ rpm, $f_r \approx 25.5$ in/min, $A \approx 1.41$ in, $T_m \approx 0.58$ min, $MRR \approx 7.6$ in³/min.
- The post's worked solutions are images; formulas here are textbook forms, so compare with the post's answers.