FUNDAMENTALS OF ADVANCED MACHINING

AUTODESK®
FUSION 360™

JORGE CASAS

MACHINE THEORY

Knowing your Machine

PROGRAMMING IN CAM FOR DIFFERENT MACHINES

When programming in CAM, understanding the capabilities and limitations of the specific machine is essential. Mills differ significantly in spindle power, rigidity, work envelope, and tooling options, which directly impact toolpaths, speeds, feeds, and overall strategy.

Let's use an example.

- **Tormach 770M:** 10,000 RPM, 1.5 HP, smaller work envelope, less rigid; best for lighter materials.
- **Haas Super Mini Mill:** 10,000 RPM, 15.0 HP, larger work envelope, higher rigidity; handles harder materials.

- Use lighter cuts and adaptive toolpaths for the **Tormach**.
- **Haas** can manage aggressive cuts and deeper tool engagements.

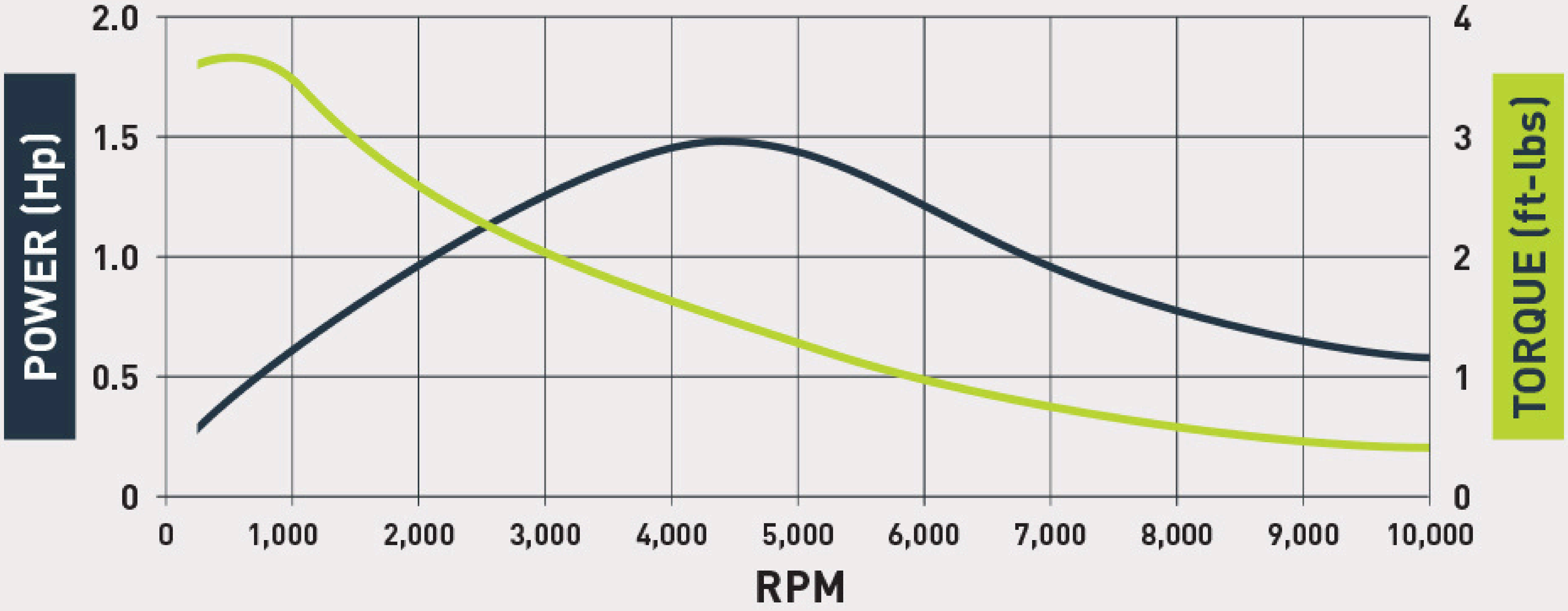
TORQUE CURVES

Torque curves represent the relationship between a spindle's speed (RPM) and the torque it can deliver. Understanding torque curves is essential for programming in CAM, as it helps optimize cutting conditions for different materials and operations.

- **High Torque at Low RPM:** Machines designed for hard materials, like steel or titanium, often provide higher torque at lower spindle speeds, ideal for heavy cuts or slow feeds.
- **Low Torque at High RPM:** Machines optimized for softer materials, such as aluminum or plastics, deliver less torque at high speeds, suitable for high-speed machining and fine finishes.

Inform decisions on feed rates, depths of cut (DOC), and toolpath strategies.

770M/MX HIGH BELT (120V)
POWER & TORQUE CURVES

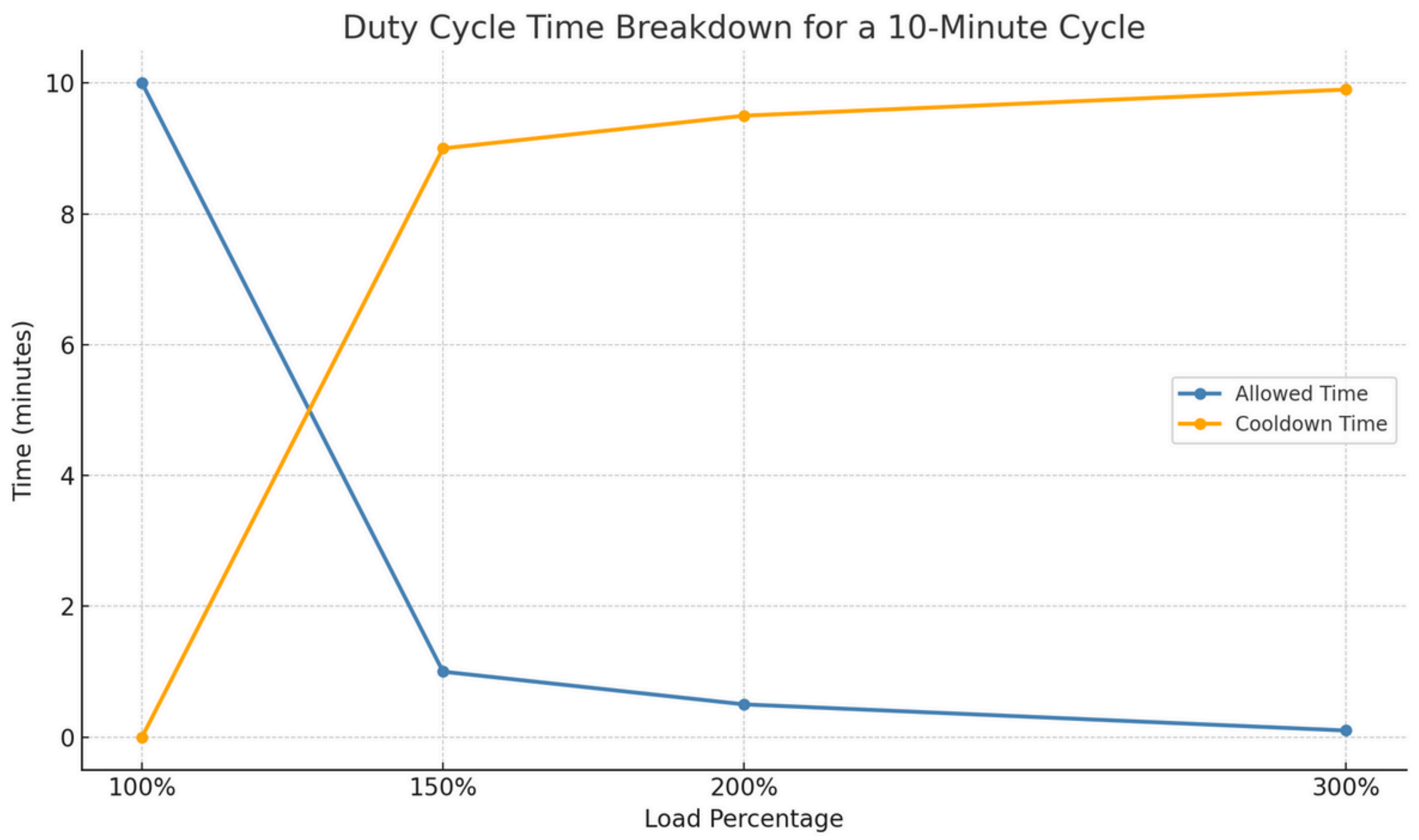


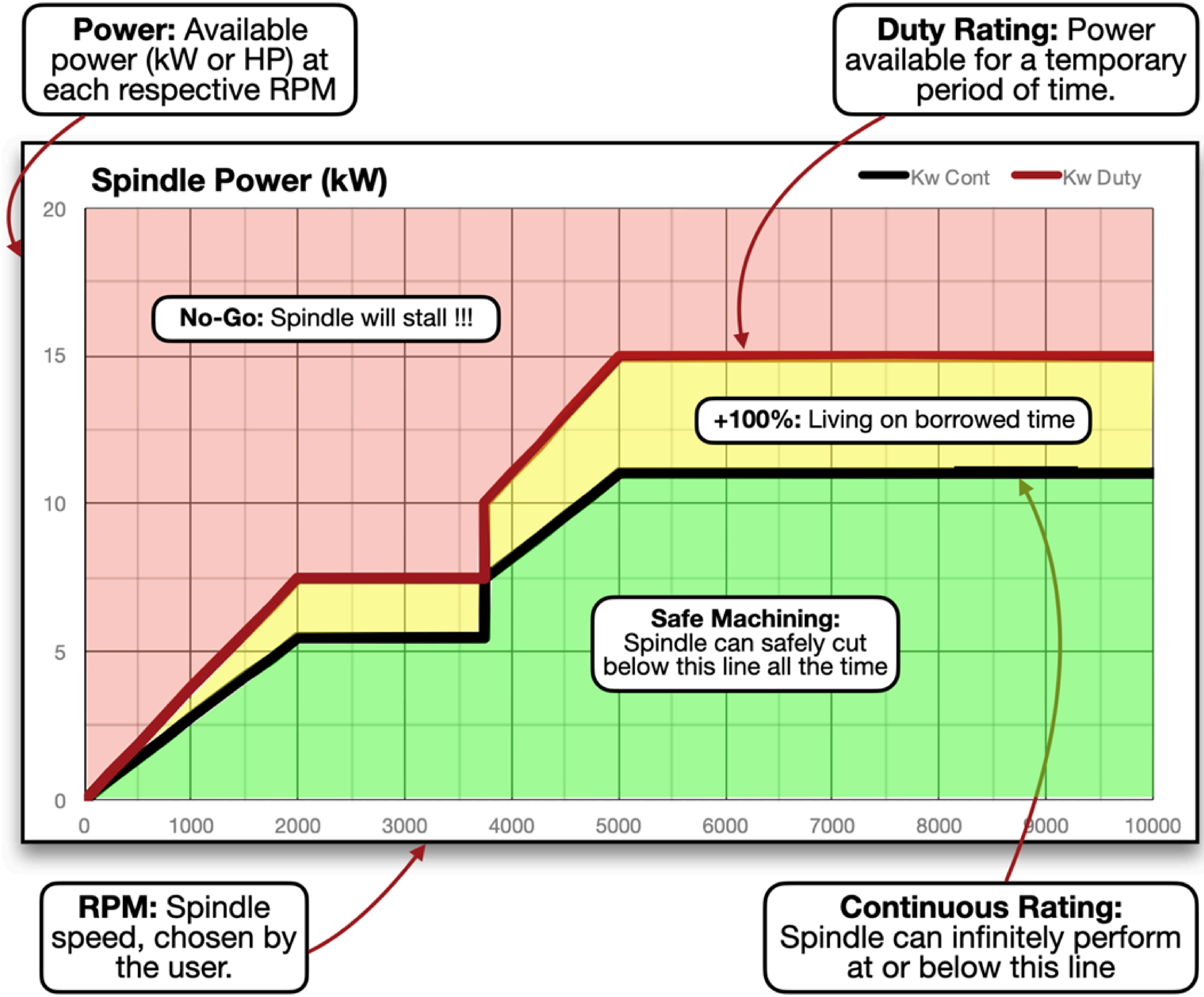
DUTY CYCLE

The duty cycle refers to the percentage of time a machine's spindle or motor can operate at a specific load without overheating or causing long-term damage. For example, a spindle with a 100% duty cycle at full load can safely run at its maximum rated torque or power continuously for the entire cycle (e.g., 10 minutes in a 10-minute cycle). However, if the spindle operates at a 150% load, the duty cycle might only allow for 10% of the cycle (e.g., 1 minute at high load), followed by a cooldown period of 9 minutes at or below 100% load to prevent overheating.

- **Continuous vs. Intermittent Operation:** Lower-duty cycles may run continuously and indefinitely, whereas high-duty cycles are better suited for intermittent heavy machining that may require breaks for cooling.
- **Spindle Longevity:** Exceeding the duty cycle can lead to overheating, reduced spindle life, or motor failure.
- **Monitoring:** Always monitor spindle load meters and avoid sustained operation near 100% load for extended periods.

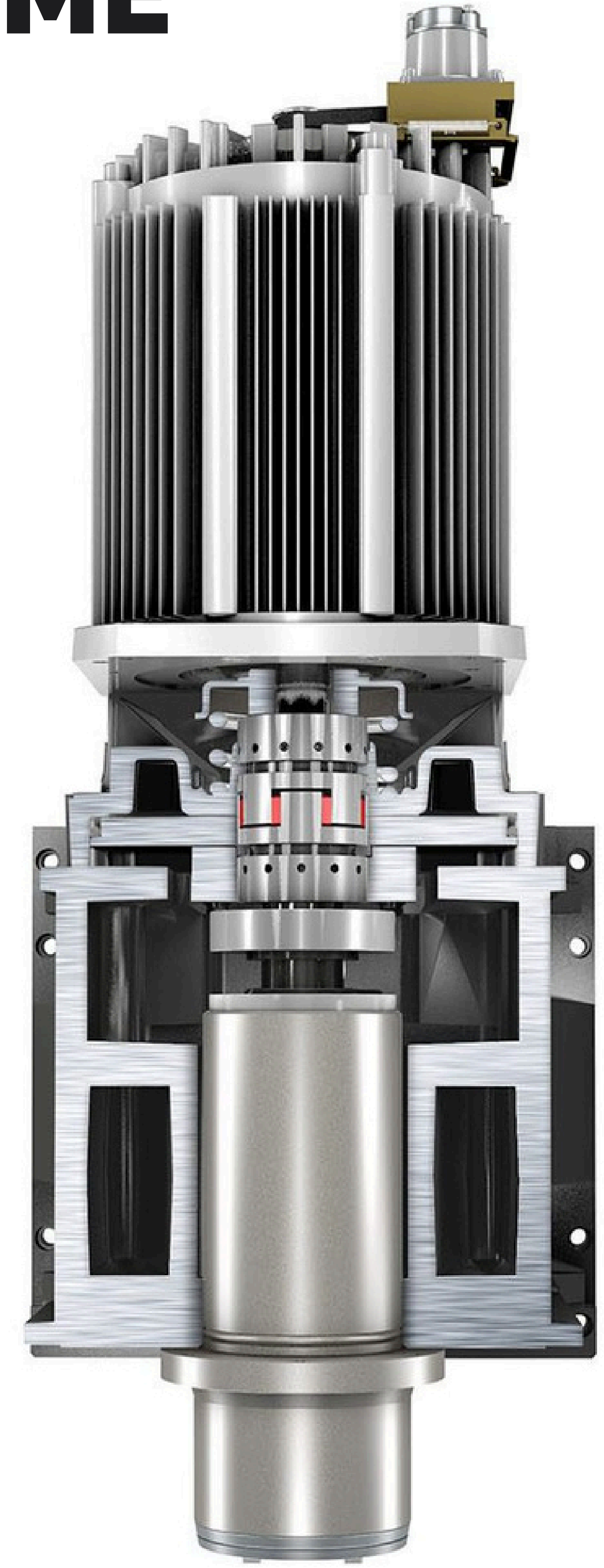
Knowing these limits allows you to program toolpaths that maximize material removal while protecting the spindle from damage or overheating. Machines with poorly matched duty cycles to your application will require slower operations, reducing productivity.




Power: Available power (kW or HP) at each respective RPM
Duty Rating: Power available for a temporary period of time.
Spindle Power (kW)
Kw Cont
Kw Duty
20
15
10
5
0
0
1000
2000
3000
4000
5000
6000
7000
8000
9000
10000
No-Go: Spindle will stall !!!
+100%: Living on borrowed time
Safe Machining: Spindle can safely cut below this line all the time
RPM: Spindle speed, chosen by the user.
Continuous Rating: Spindle can infinitely perform at or below this line

PRESERVING SPINDLE MOTOR LIFETIME

To maintain the health and longevity of a spindle motor, it's crucial to operate it within its continuous duty cycle rating at 100% for most operations. For short bursts of higher performance, the motor can operate at a higher load duty cycle 150-200%, but these should be carefully managed.



General Guidelines for Acceptable Duty Cycles:

- **Continuous vs. Intermittent Operation:** Lower-duty cycles may run continuously and indefinitely, whereas high-duty cycles are better suited for intermittent heavy machining that may require breaks for cooling.
- **Spindle Longevity:** Exceeding the duty cycle can lead to overheating, reduced spindle life, or motor failure.
- **Monitoring:** Always monitor spindle load meters and avoid sustained operation near 100% load for extended periods.

GUIDELINES FOR A HEALTHY SPINDLE

Continuous Operation (S1):

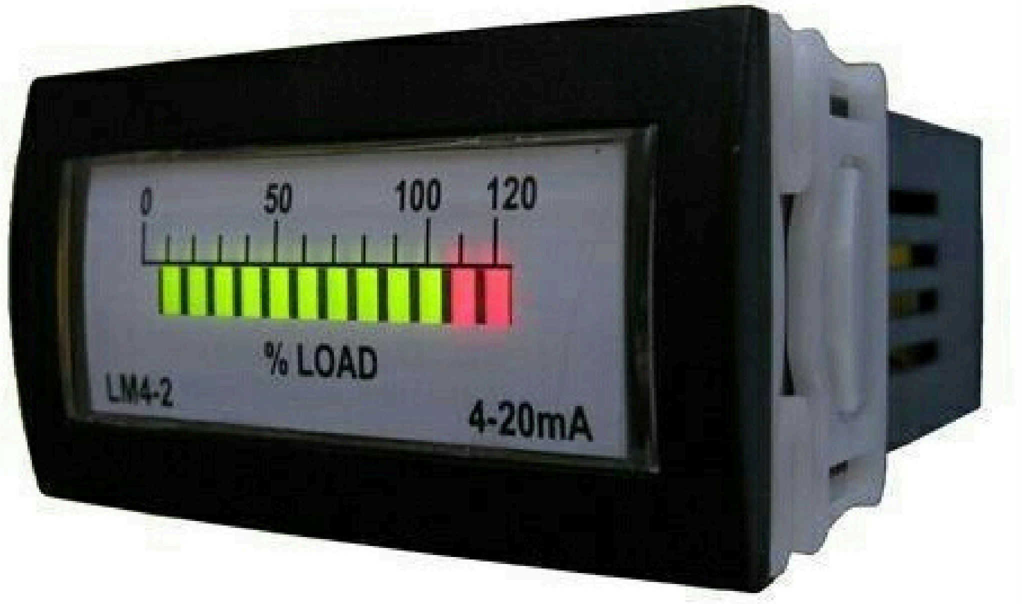
- Use 100% rated load as the baseline for long-term, continuous operations.
- This ensures the spindle can run indefinitely without overheating or excessive wear.
- Short-Burst Duty Cycles (S3-S6):
- Higher loads, such as 150-200% of the rated load, are acceptable for short durations (e.g., 1 minute at 150% load or 30 seconds at 200% load).
- Cooldown periods must follow these bursts to prevent overheating.

Example: If running at 150% load:

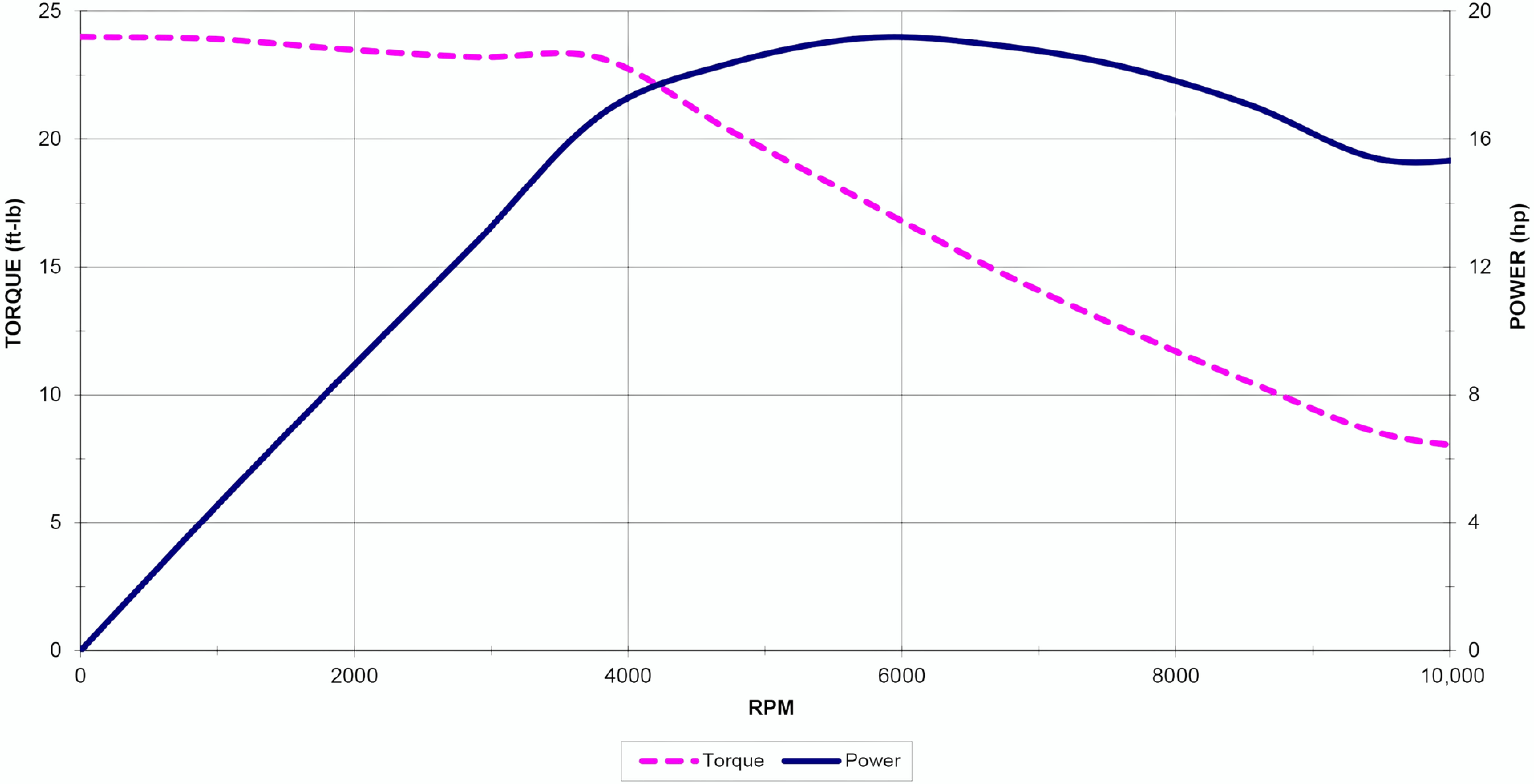
- Operate for 1 minute (10% of a 10-minute cycle).
- Allow 9 minutes of operation at or below 100% load for cooling.

Maximum Safe Usage:

- Avoid running the spindle at 200% load for extended periods, even in short bursts, as this can accelerate wear and thermal stress.
- Stick to 80-90% of the spindle's rated power and torque for continuous heavy-duty operations.



Super Mini Mill Series
10,000-rpm, Belt-Drive Spindle
40 Taper – 15 hp
Standard: Super Mini Mill, Super Mini Mill 2
Optional: None



Values shown are 200% spindle Load

Note: This torque also applies to the 10K grease-packed spindle option.

1/29/2016

EXAMPLE

Let’s optimize the torque and power for the Haas Super Mini Mill

For the Haas Super Mini Mill, the torque and power curve shows:

- **Peak Torque:** ~12 ft-lbs at 1,000 RPM at 100% spindle load
- **Peak Power:** ~9 HP at 6,000 RPM at 100% spindle load.

If you're performing high-speed machining (HSM) where consistent power and moderate torque are critical, you can balance the spindle's duty cycle to optimize performance while avoiding overheating.

Scenario: High-Speed Machining in Aluminum

- **RPM Range:** Operate around 3-4,000 RPM to take advantage of the spindle's moderate power output (~8 HP) and maximum torque output (~12 ft-lbs) at 100% duty cycle.
- **Torque Usage:** At 6,000 RPM, the torque drops to ~9 ft-lbs, which is still sufficient for the lighter cutting forces required in HSM, therefore a lower torque requirement. Much higher RPM’s can be achieved in this manner.

Duty Cycle Management:

- The spindle may operate continuously at full load (100% power) for long periods.
- For higher duty cycles (>100%), run the spindle at boosted power for the short durations, followed by microbursts of cooldown period or a lower-load operation to allow heat dissipation.

HOW TORQUE AND POWER CHARTS HELP CAM

- Program Efficient Toolpaths: Charts provide data to fine-tune speeds, feeds, and cutting parameters for each operation.
- Plan for Duty Cycles: Use the charts to balance high loads with cooldown periods, maximizing spindle efficiency while avoiding damage.
- Select the Right Tools: Match spindle strengths to tool sizes and materials, optimizing cutting performance and reducing costs.
- Increase MRR: High-power zones in the chart enable faster MRR (material removal rate) when paired with adaptive machining strategies.

Ultimately, CAM is all about catering to your machine. Take care of your machine!

MACHINE RIGIDITY AND SPINDLE POWER

Even with a high-performance spindle, the machine's overall rigidity determines how much you can utilize its capabilities. Lightweight or poorly constructed machines cannot fully exploit powerful spindles without risking excessive wear or inaccurate machining.

- Rigid castings, strong linear guides, and stable ball screws extend machine life and ensure accurate cuts.
- Machines with insufficient rigidity will wear out faster and limit your ability to take advantage of the spindle's potential.

A perfect example of this is in how each machine holds your tools in the spindle. Not all machines are alike in this way.

PRACTICE PROBLEM

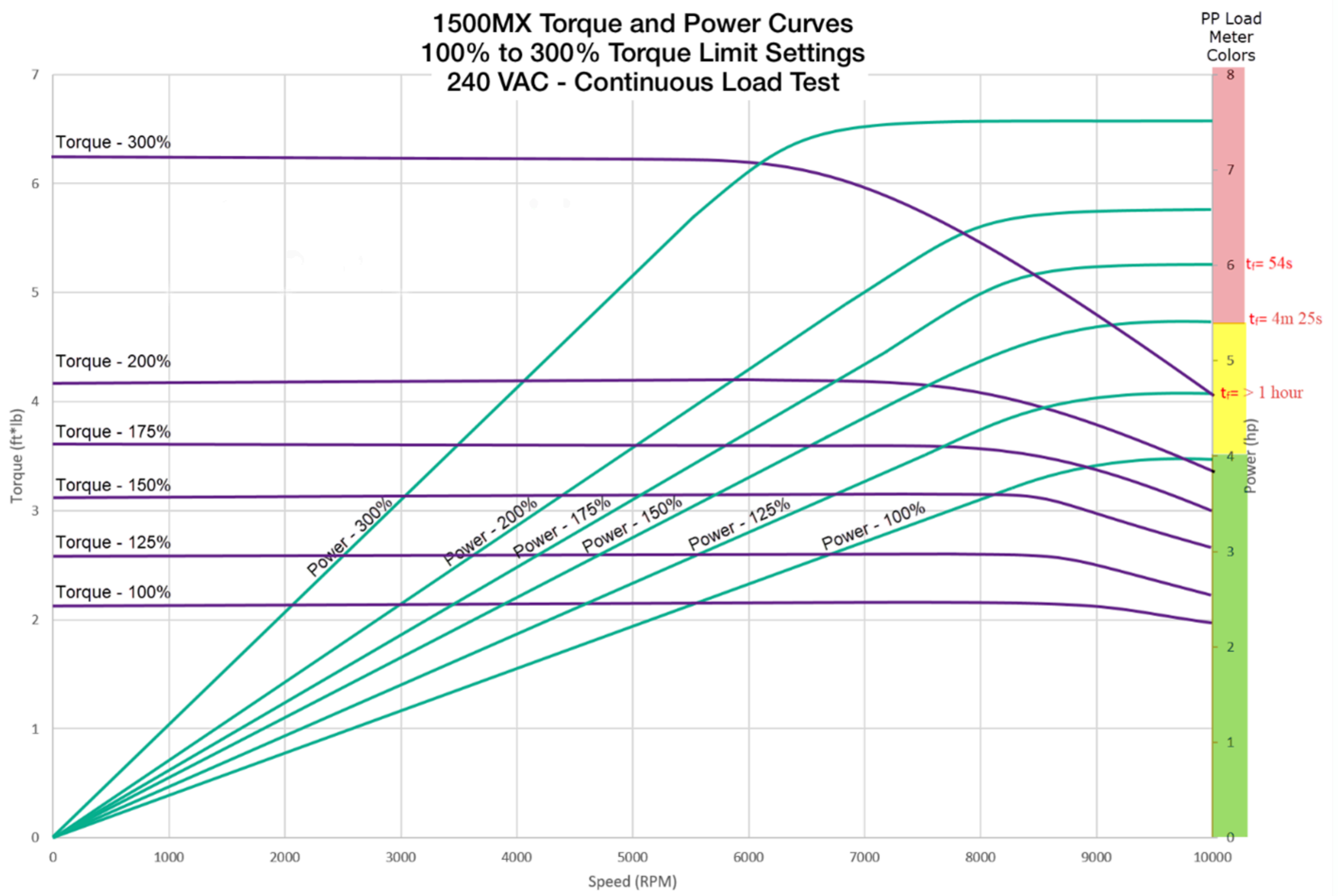
You're tasked with machining a part from 6061 aluminum on the Tormach 1500MX. Your toolpath involves the following operations:

Scenario 1:

- RPM: 8,000
- Estimated spindle load: 125%.

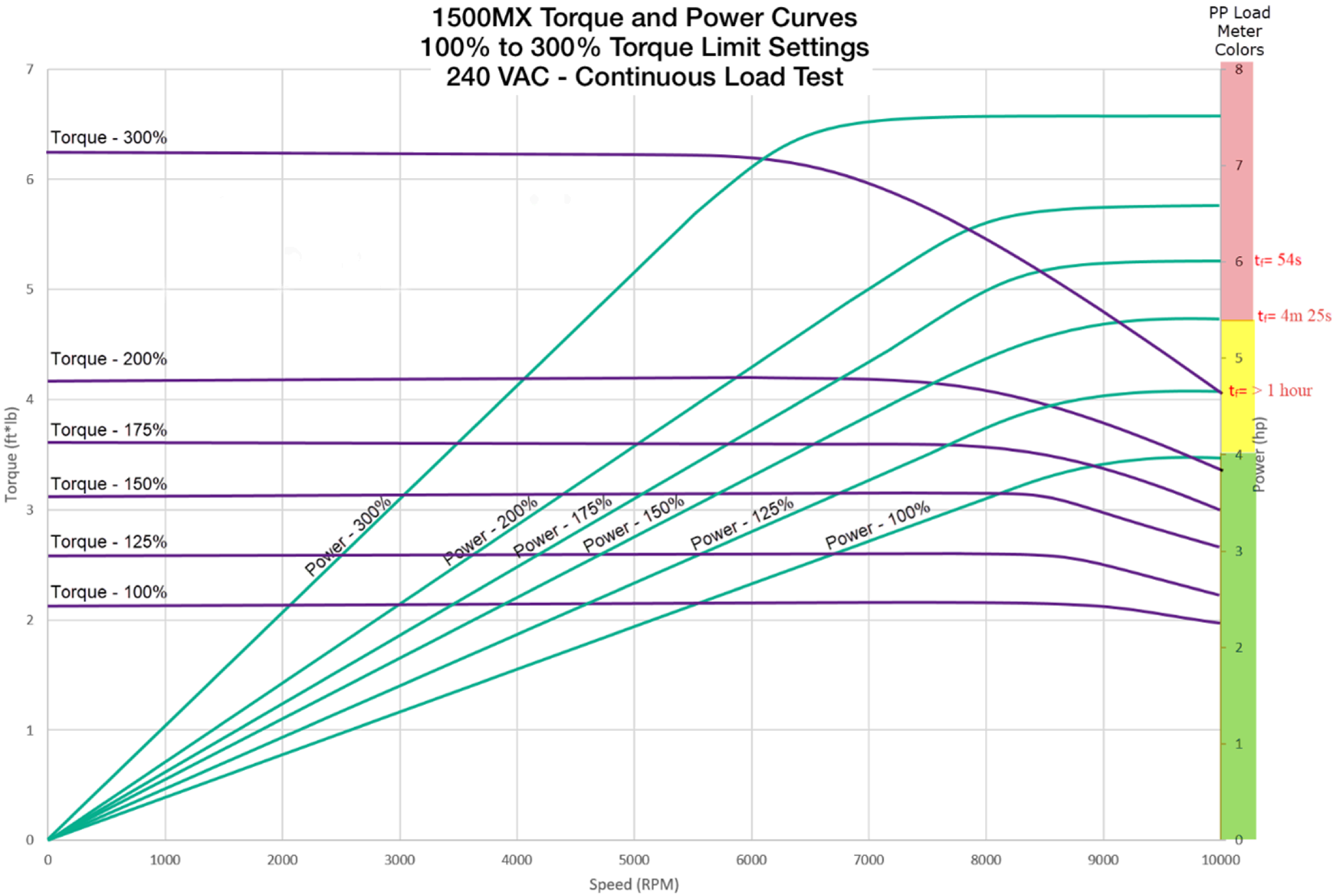
Scenario 2:

- RPM: 10,000
- Estimated spindle load: 50%.



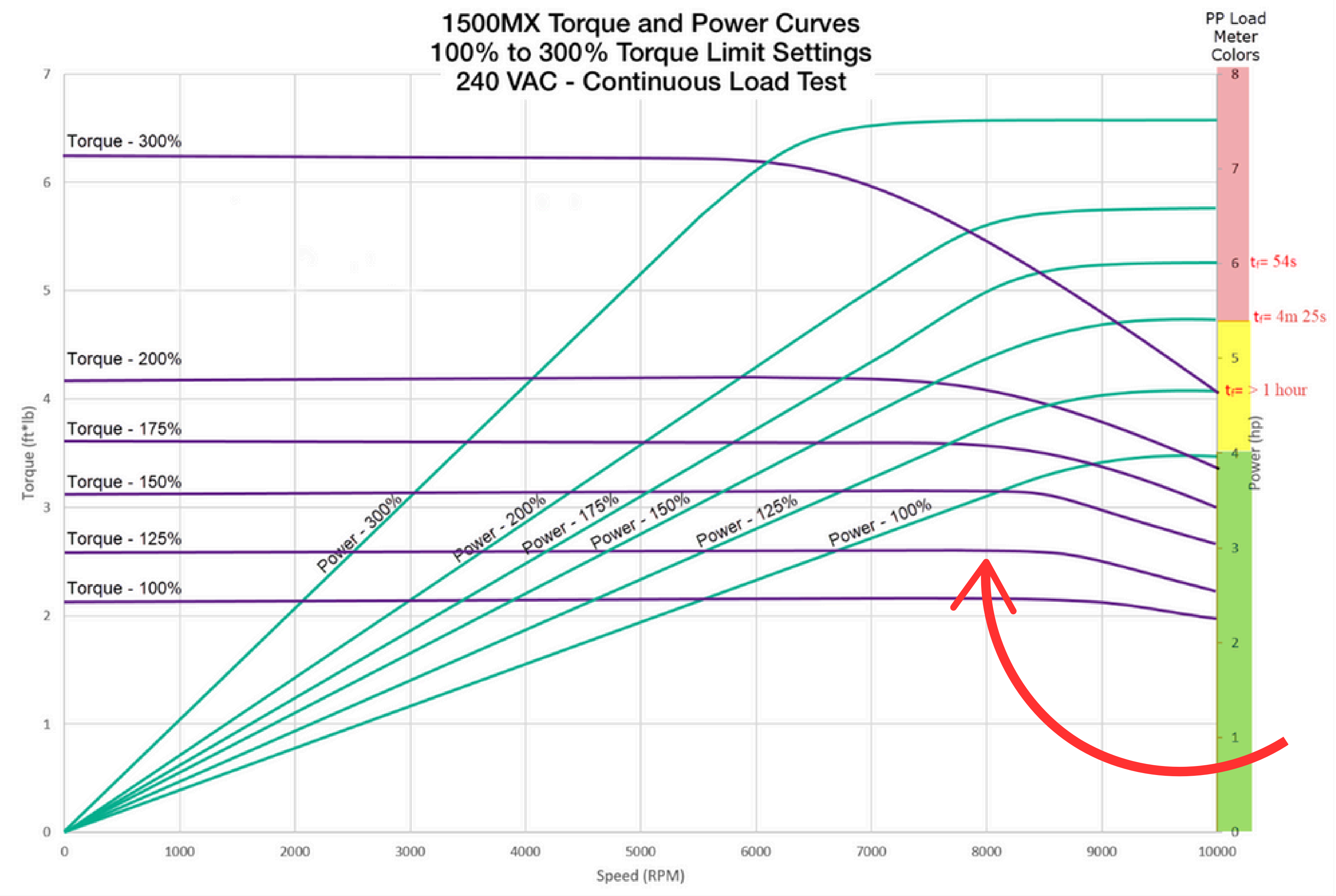
Using the provided 1500MX torque and power curve:

- **Identify the torque available at 8,000 RPM. Is it sufficient for high spindle load operations at 125% load?**
- **Compare the power output at 8,000 RPM and 10,000 RPM. Which operation benefits more from the available power?**
- **Based on the chart, what is the continuous duty cycle range (100% load) for the 1500MX spindle, and how does it impact your toolpath planning?**


1500MX Torque and Power Curves
100% to 300% Torque Limit Settings
240 VAC - Continuous Load Test
PP Load Meter Colors
Torque - 300%
Torque - 200%
Torque - 175%
Torque - 150%
Torque - 125%
Torque - 100%
Power - 300%
Power - 200%
Power - 175%
Power - 150%
Power - 125%
Power - 100%
tf= 54s
tf= 4m 25s
tf= > 1 hour
Torque (ft*lb)
Power (hp)
Speed (RPM)
0
1000
2000
3000
4000
5000
6000
7000
8000
9000
10000

PRACTICE PROBLEM

Identify the torque available at 8,000 RPM. Is it sufficient for high spindle load operations at 125% load?



From the chart:

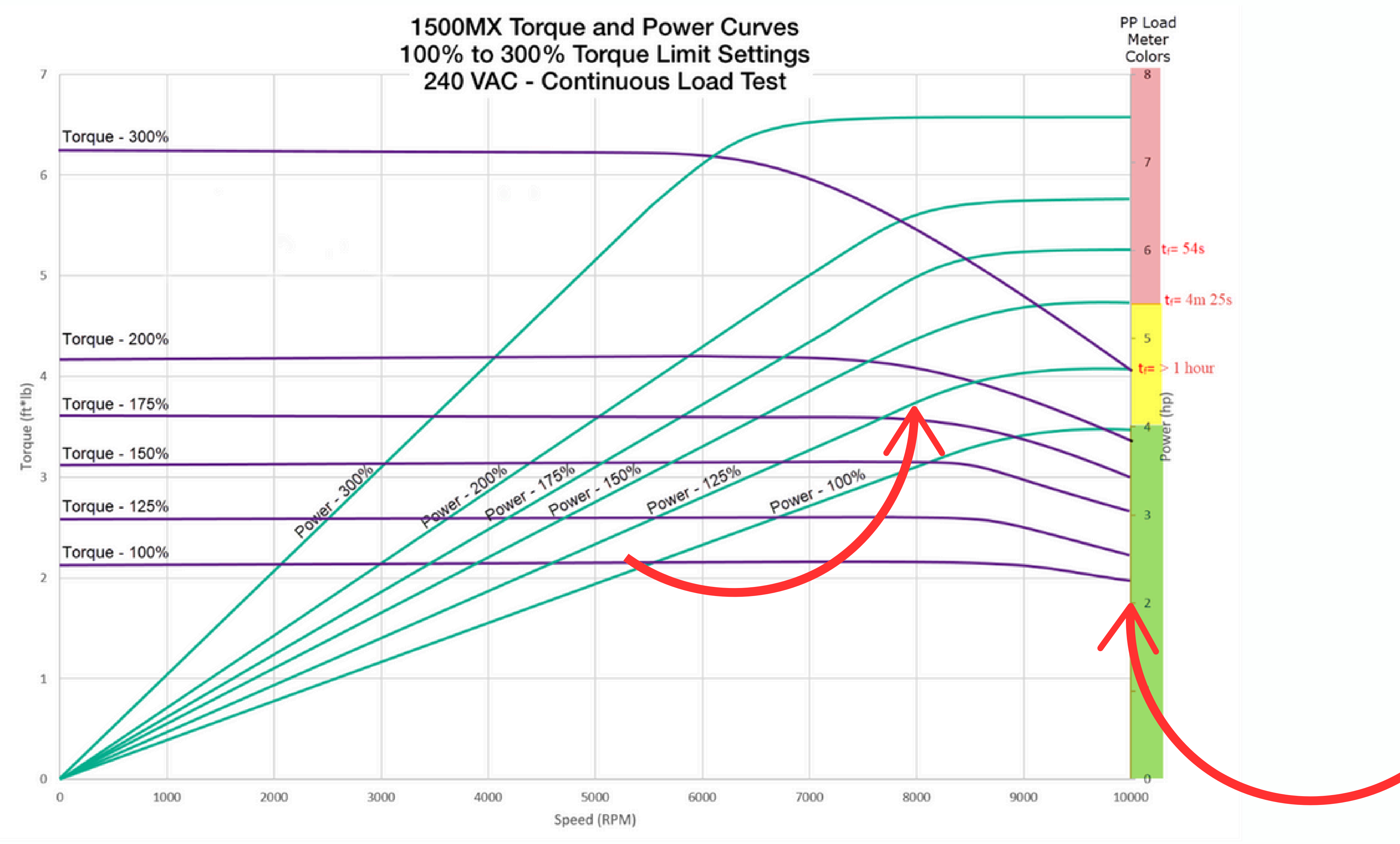
- At 8,000 RPM, the 125% torque line shows approximately 2.5 ft-lbs of torque.
- The available torque is sufficient for moderate material removal at 125% load, but since torque drops significantly at higher RPMs, you must ensure your cutting forces (e.g., Depth of Cut and Width of Cut) align with this torque.

Conclusion:

- While 125% load is supported, the reduced torque at higher RPMs indicates that high-load cutting operations may require a smaller diameter tool or reduced engagement to avoid spindle strain.

PRACTICE PROBLEM

Compare the power output at 8,000 RPM and 10,000 RPM. Which operation benefits more from the available power?



From the chart:

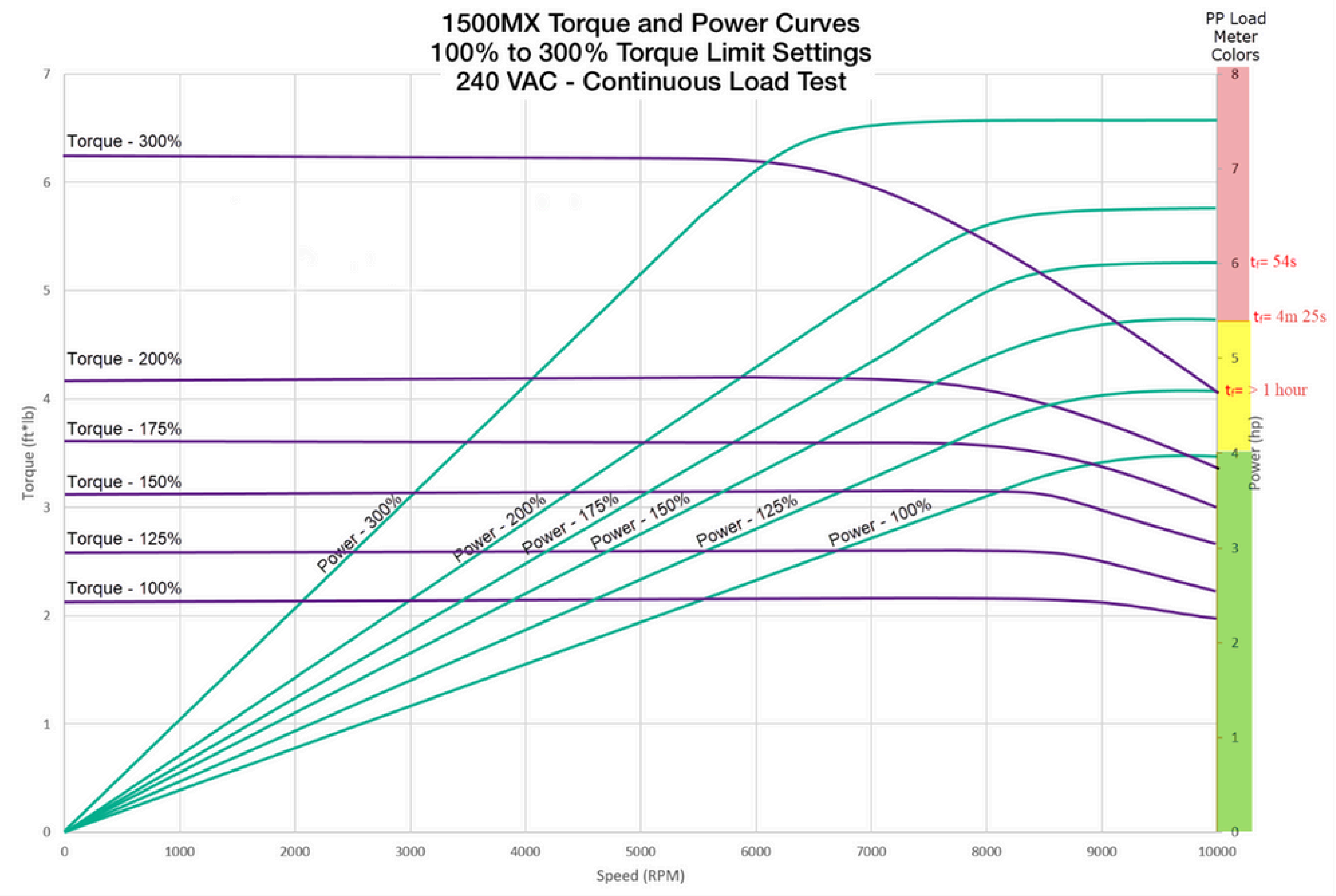
- At 8,000 RPM, the power output at 125% load is approximately 4.5 HP.
- At 10,000 RPM, the power output drops to around 2 HP at 50% spindle load. Notice there is no line for the 50% Power Curve, so we'll take the 100% Power Curve and divide by 2 in this case to get an **approximate** power requirement.

Conclusion:

- 10,000 RPM operates with less power output (2 HP) and there is a significant torque dropoff
- You may benefit more from 8,000 RPM but may use it sparingly at 125% spindle load

PRACTICE PROBLEM

Based on the chart, what is the continuous duty cycle range (100% load) for the 1500MX spindle, and how does it impact your toolpath planning?



From the chart:

- At 100% load, the spindle can operate continuously (green region).
- For higher loads (e.g., 125%, 150%, 200%), the duty cycle progressively decreases:
  - 125% load: Approximately 4 minutes, 25 seconds.
  - 150% load: Approximately 1 minute.
  - 200% load: Approximately 54 seconds.

Conclusion:

- Toolpath planning should prioritize staying within the 100% load range (continuous duty cycle) to avoid overheating.
- If higher spindle loads are necessary (e.g., 150%), limit the operation to short bursts (e.g. 1 minute max) followed by a cooldown period or lighter machining.

MATERIAL THEORY

Starting with Stock

STOCK? YEA📈💰

Stock material refers to the raw material from which a workpiece is machined. Proper understanding and management of stock material are critical for machining efficiency, tool life, and final product quality.

Understanding stock material ensures proper machining strategies.

- stock material is the starting raw material for machining
- correct stock choice impacts machining efficiency and quality
- understanding material properties prevents tooling issues
- design for manufacturing

WHY IT'S IMPORTANT TO KNOW YOUR MATERIAL

The properties of a material—such as hardness, thermal conductivity, and machinability—determine the best tools, cutting parameters, and strategies to use. Knowledge of the material ensures efficient machining, minimizes tool wear, and enhances safety.

- **Machinability:** Impacts cutting speeds, feeds, and tool selection
- **Hardness:** Determines the type of tooling and machining strategy
- **Thermal Properties:** Influence heat management and coolant needs
- **Surface Finish:** Different materials require specific tools for precision
- **Tool Wear:** Abrasive or hard materials need tougher tools

BRINELL HARDNESS

Material hardness is a measure of its resistance to deformation and cutting. Understanding hardness helps determine the best tools and machining parameters for a given material, ensuring both efficiency and tool longevity.

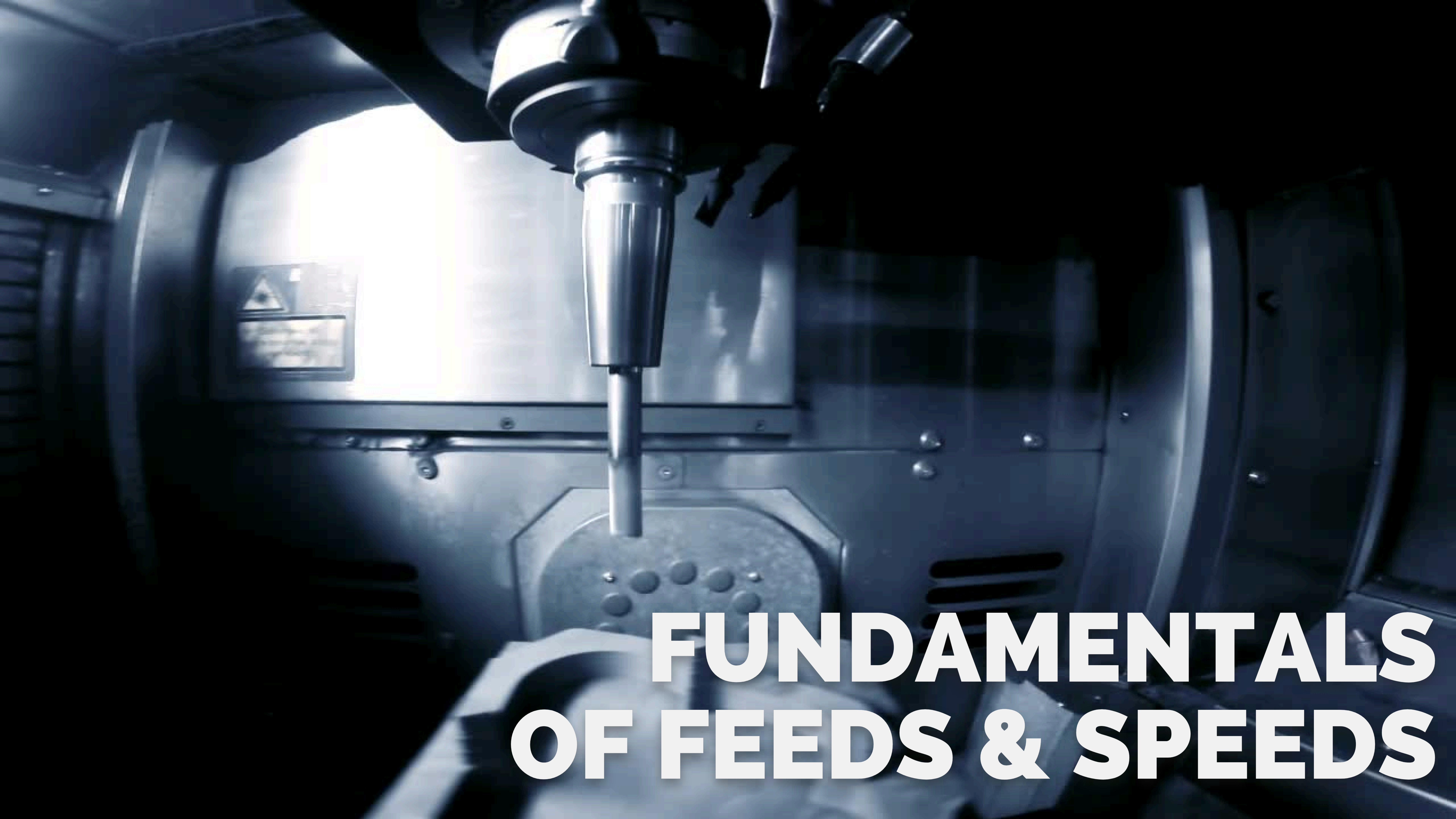
Brinell Hardness Number (BHN) is one of the most commonly used hardness scales for metals, providing a measure of a material's resistance to indentation. It’s especially useful when selecting stock materials for machining because it helps determine the material's machinability and the horsepower required to remove material effectively.

Brinell Hardness is calculated by pressing a hard steel or carbide ball into the material's surface under a fixed load. The size of the resulting indentation determines the hardness value.

SOFT STEEL, GREY AND MALLEABLE CAST IRON AND MOST NON-FERROUS METALS

Rockwell®							Superficial			Knoop	Brinell		Tensile Strength	Micro ficial
												10 mm Vickers 136°		
B	F	G	A	E	H	K	15-T	30-T	45-T	HK	HB	HB/HV	KSI	WMN
100 kg 1/16" ball	60 kg 1/16" ball	150 kg 1/16" ball	60 kg Brale	100 kg 1/8" ball	60 kg 1/8" ball	150 kg 1/8" ball	15 kg 1/16" ball	30 kg 1/16" ball	45 kg 1/16" ball	500 gm and over	500 kg 10 mm ball	3000 kg 10 kg	1000lbs/ sq in	1000 gm
100	▲	82.5	61.5	▲	▲	▲	93.1	83.1	72.9	251	201	240	116	730
99	•	81.0	60.9	•	•	•	92.8	82.5	71.9	246	195	234	114	725
98	•	79.0	60.2	•	•	•	92.5	81.8	70.9	241	189	228	109	719
97	•	77.5	59.5	•	•	•	92.1	81.1	69.9	236	184	222	104	713
96	•	76.0	58.9	•	•	•	91.8	80.4	68.9	231	179	216	102	707
95	•	74.0	58.3	•	•	•	91.5	79.8	67.9	226	175	210	100	701
94	•	72.5	57.6	•	•	•	91.2	79.1	66.9	221	171	205	98	696
93	•	71.0	57.0	•	•	NA	90.8	78.4	65.9	216	167	200	94	690
92	•	69.0	56.4	•	•	100	90.5	77.8	64.8	211	163	195	92	684
91	•	67.5	55.8	•	•	99.5	90.2	77.1	63.8	206	160	190	90	679
90	•	66.0	55.2	•	•	98.5	89.9	76.4	62.8	201	157	185	89	674
89	•	64.0	54.6	•	•	98.0	89.5	75.8	61.8	196	154	180	88	668
88	•	62.5	54.0	•	•	97.0	89.2	75.1	60.8	192	151	176	86	662
87	•	61.0	53.4	•	•	96.5	88.9	74.4	59.8	188	148	172	84	656
86	•	59.0	52.8	•	•	95.5	88.6	73.8	58.8	184	145	169	83	651
85	•	57.5	52.3	•	•	94.5	88.2	73.1	57.8	180	142	165	82	646
84	•	56.0	51.7	•	•	94.0	87.9	72.4	56.8	176	140	162	81	640
83	•	54.0	51.1	•	•	93.0	87.6	71.8	55.8	173	137	159	80	634
82	•	52.5	50.6	•	•	92.0	87.3	71.1	54.8	170	135	156	77	629
81	•	51.0	50.0	•	•	91.0	86.9	70.4	53.8	167	133	153	73	624
80	•	49.0	49.5	•	•	90.5	86.6	69.7	52.8	164	130	150	72	618
79	•	47.5	48.9	•	•	89.5	86.3	69.1	51.8	161	128	147	70	612
78	•	46.0	48.4	•	•	88.5	86.0	68.4	50.8	158	126	144	69	607
77	•	44.0	47.9	•	•	88.0	85.6	67.7	49.8	155	124	141	68	602
76	NA	42.5	47.3	•	•	87.0	85.3	67.1	48.8	152	122	139	67	596
75	99.6	41.0	46.8	•	•	86.0	85.0	66.4	47.8	150	120	137	66	592
74	99.1	39.0	46.3	•	•	85.0	84.7	65.7	46.8	147	118	135	65	587
73	98.5	37.5	45.8	•	•	84.5	84.3	65.1	45.8	145	116	132	64	581
72	98.0	36.0	45.3	NA	•	83.5	84.0	64.4	44.8	143	114	130	63	576
71	97.4	34.5	44.8	100	•	82.5	83.7	63.7	43.8	141	112	127	62	571
70	96.8	32.5	44.3	99.5	•	81.5	83.4	63.1	42.8	139	110	125	61	566
69	96.2	31.0	43.8	99.0	•	81.0	83.0	62.4	41.8	137	109	123	60	561
68	95.6	29.5	43.3	98.0	•	80.0	82.7	61.7	40.8	135	107	121	59	556
67	95.1	28.0	42.8	97.5	•	79.0	82.4	61.0	39.8	133	106	119	58	551
66	94.5	26.5	42.3	97.0	•	78.0	82.1	60.4	38.7	131	104	117	57	546
65	93.9	25.0	41.8	96.0	•	77.5	81.8	59.7	37.7	129	102	116	56	542
64	93.4	23.5	41.4	95.5	•	76.5	81.4	59.0	36.7	127	101	114	NA	537
63	92.8	22.0	40.9	95.0	•	75.5	81.1	58.4	35.7	125	99	112	•	532
62	92.2	20.5	40.4	94.5	•	74.5	80.8	57.7	34.7	124	98	110	•	527
61	91.7	19.0	40.0	93.5	•	74.0	80.5	57.0	33.7	122	96	108	•	522
60	91.1	17.5	39.5	93.0	•	73.0	80.1	56.4	32.7	120	95	107	•	517
59	90.5	16.0	39.0	92.5	•	72.0	79.8	55.7	31.7	118	94	106	•	512
58	90.0	14.5	38.6	92.0	•	71.0	79.5	55.0	30.7	117	92	104	•	507
57	89.4	13.0	38.1	91.0	•	70.5	79.2	54.4	29.7	115	91	103	•	502
56	88.8	11.5	37.7	90.5	•	69.5	78.8	53.7	28.7	114	90	101	•	497
55	88.2	10.0	37.2	90.0	•	68.5	78.5	53.0	27.7	112	89	100	•	492
54	87.7	8.5	36.8	89.5	•	68.0	78.2	52.4	26.7	111	87	NA	•	487
53	87.1	7.0	36.3	89.0	•	67.0	77.9	51.7	25.7	110	86	•	•	482
52	86.5	5.5	35.9	88.0	•	66.0	77.5	51.0	24.7	109	85	•	•	477
51	86.0	4.0	35.5	87.5	•	65.0	77.2	50.3	23.7	108	84	•	•	472
50	85.4	2.5	35.0	87.0	•	64.5	76.9	49.7	22.7	107	83	•	•	468
49	84.8	NA	34.6	86.5	•	63.5	76.6	49.0	21.7	106	82	•	•	463
48	84.3	•	34.1	85.5	•	62.5	76.2	48.3	20.7	105	81	•	•	458

FUNDAMENTALS OF FEEDS & SPEEDS



WHY START FROM BHN-TO-HP INSTEAD OF SFM?

What Is SFM?

Surface Feet per Minute (SFM) is a measure of cutting speed. It quantifies how fast the material moves past the cutting edge of a tool, influencing heat generation, tool wear, and surface finish.

$$SFM = \frac{\pi \times D \times N}{12}$$

Where:

- D: Tool diameter (inches).
- N: Spindle speed (RPM).
- Dividing by 12 converts inches to feet.

Why It Matters:

- SFM is critical for selecting spindle speeds to avoid tool overheating or excessive wear.
- Different materials have recommended SFM ranges, based on their machinability and hardness.

So it makes sense to start with SFM when programming your machine, instead of having to calculate the BHN-to-HP, and then the HP to Torque, and then Torque to RPM? After all, there are so many SFM values posted everywhere on the internet readily available to plug into CAM...

...Right?

MATERIAL HARDNESS IS THE FOUNDATION

BHN (Brinell Hardness Number) directly quantifies a material's resistance to cutting and deformation, providing a universal baseline for machinability. SFM is a derived value based on cutting speeds and is more specific to a given tool and operation.

Starting with BHN allows you to estimate the inherent energy demands of machining a material, which applies universally across tools, machines, and operations.

BHN is directly related to the **Unit Power (UP)** needed to remove material, helping to predict the machine’s capabilities regardless of the tool or speed. It simplifies understanding of how different materials challenge your machine based on hardness alone, allowing for better stock selection.

SFM Focuses on the Tool, Not the Machine

It is more specific to tool performance and geometry than to the machine's raw power or torque capabilities. While SFM is essential for setting spindle speeds, it doesn't directly address whether your machine has the horsepower or torque to maintain those speeds under load.

SFM is critical once you've established that your machine can handle the material (via BHN-to-HP).

And then finally, use SFM to fine-tune spindle speeds and feed rates for a given tool, ensuring optimal cutting conditions.

MATERIAL REMOVAL RATE

The first step in determining whether a machine can handle a given configuration for machining a particular material is to calculate the Material Removal Rate (MRR), which is the volume of material removed per minute:

$$MRR = DOC \times WOC \times V_f$$

- DOC: Depth of cut (in inches or mm).
- WOC: Width of cut (in inches or mm).
- Vf: Feed rate (inches or mm per minute).

We need a starting point...

Material removal has no single correct answer because it is a parametric problem. Adjusting any variable—such as speeds, feeds, depth of cut, or width of cut—affects the outcome, resulting in an infinite number of possible solutions.

HORSEPOWER CALCULATION

The spindle horsepower required to remove material at the cutter - indicated by power at the cutter, Pc - is calculated as:

$$P_c = UP \times MRR$$

- Unit Power (UP): Power required to cut 1 cubic inch of the material per minute, which depends on the BHN of the material you wish to cut.

The spindle horsepower, or power at the motor, Pm, is related by the following equation using the machine's spindle efficiency, E, to transmit that power to the cutter.

$$P_m = \frac{UP \times MRR}{E}$$

Approximate Unit Power values:

- Aluminum (BHN 50–150): 0.25–0.33 HP/in^3/min.
- Mild Steel (BHN 80-360): 0.63–1.14 HP/in^3/min.
- Tool Steel (BHN 250–400): 0.98–1.30 HP/in^3/min.
- Hardened Steel (BHN >400): >1.30 HP/in^3/min.

Machinery's Handbook 26th Edition, p. 1046-1053

ADJUSTING FOR DUTY CYCLE AND CORR. FACTORS

When operating under intermittent load (short-burst duty cycles like S3–S6), you can calculate the **predicted adjusted power** by factoring in the duty cycle:

$$P_m = \frac{UP \times MRR}{E} \times DC$$

- Duty cycle percentages are derived from the machine's specifications:
- S1: 100% continuous operation.
- S3: Short bursts, typically 10–30% of a time period (e.g., 15% of 10 minutes = 1.5 minutes operation).

Two additional factors may further refine the horsepower calculation:

- K Factor: Accounts for tooling effects (e.g., sharpness, geometry, coatings). Typical K values range from 1.0–1.5, with sharper tools closer to 1.0 and duller tools closer to 1.5.
- C Factor: The Feed Factor, often denoted as C, is an empirically derived constant used in machining power equations to relate material removal rates (MRR) to the power required for a cutting process.

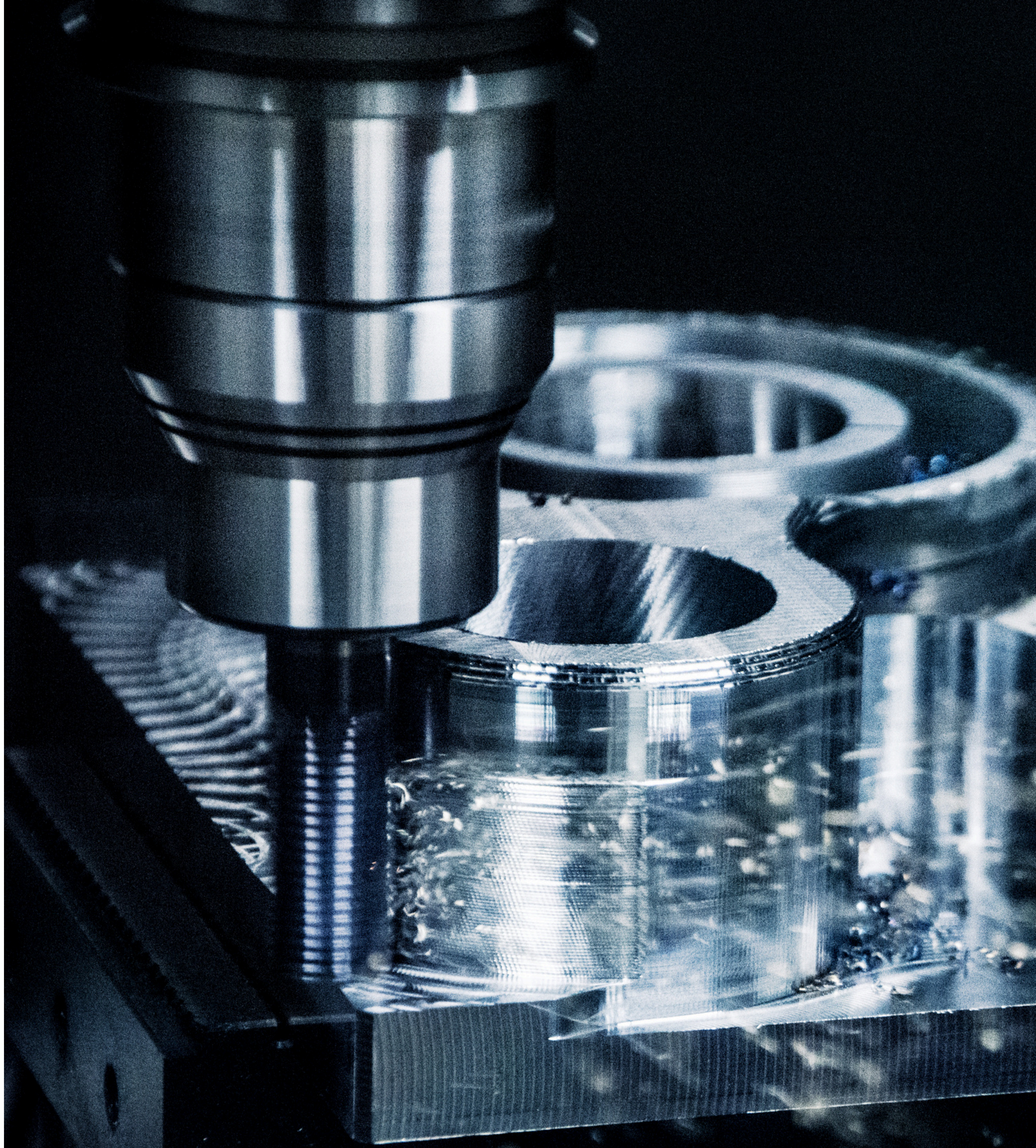
These factors may be deemed negligible, and thus you wouldn't need to adjust Pm further from the previous equation.

We'll look at these factors more in-depth later...

EXAMPLE PROBLEM

Given:

- Material: Mild Steel (BHN 350, Unit Power = 1.14 $HP/in^3/min$)
- Depth of Cut (DOC): 0.250 inch
- Width of Cut (WOC): 0.030 inch
- Feed Rate (Vf): 100 inches per minute
- Duty Cycle (DC): 80%
- Spindle Efficiency (E): 95%



STEP 1: CALCULATE MRR

$$MRR = DOC \times WOC \times V_f$$

$$MRR = 0.250 \times 0.030 \times 100 = 0.750 in^3/min$$

STEP 2: CALCULATE HORSEPOWER

$$P_c = UP \times MRR$$

$$P_c = 1.14 \times 0.75$$

$$P_c = 0.855HP$$

STEP 3: ADJUST FOR DUTY CYCLE

$$P_m = \frac{P_c}{E} \times DC$$

$$P_m = \frac{0.855HP}{0.95} \times 0.8 = 0.72HP$$

Key Takeaways:

- BHN directly correlates to Unit Power, making it a critical factor for material selection and horsepower calculations.
- Adjust cutting conditions and tooling to optimize machining performance for high-BHN materials.

HORESPOWER REQUIREMENT

$$P_m = 0.72HP$$

So we require 0.72 HP to mill into mild steel with a BHN of 150 or a Unit Power of 1.14 HP/in^3/min.

We need to now calculate S/F’s

Let’s use an **SFM** of 100 to calculate the required **RPM** and feed rate. Remember we stated at the very beginning a feed rate, **Vf**, of 100 inches per minute, **IPM**. Let’s use our formula and work backwards to find the RPM, **N**, for a 1/2" endmill:

$$SFM = \frac{\pi \times D \times N}{12}$$

We find that we need an RPM of **763**.

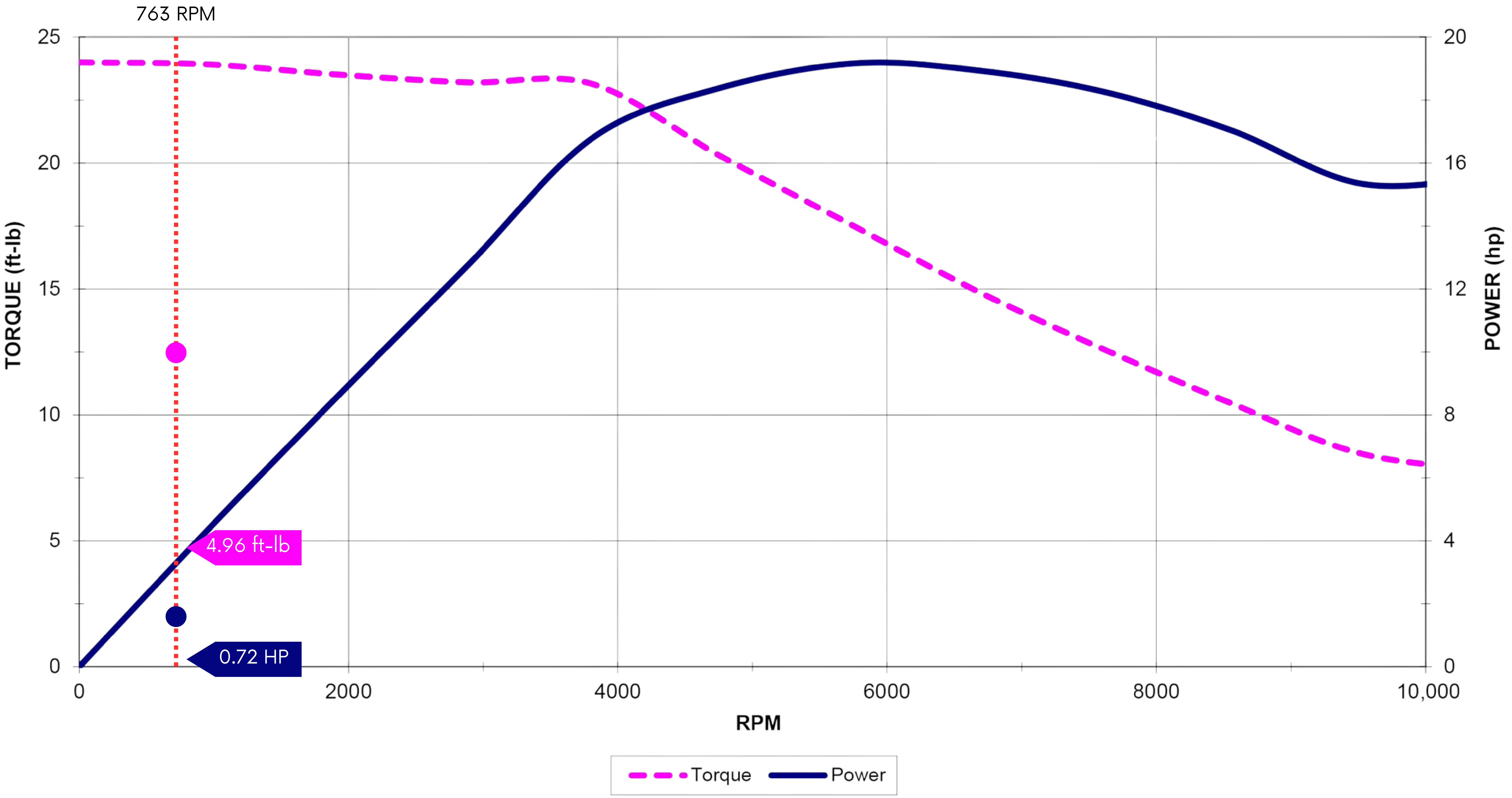
So we have our required horsepower, now what's our required torque in ft-lbs?

$$T = \frac{P_m \times 5252}{N}$$

We find that we need a Torque of **4.96 ft-lbs.**

So let’s look at our charts to see if we can output that amount of torque and power at that RPM value.

Super Mini Mill Series
10,000-rpm, Belt-Drive Spindle
40 Taper – 15 hp
Standard: Super Mini Mill, Super Mini Mill 2
Optional: None



Values shown are 200% spindle Load

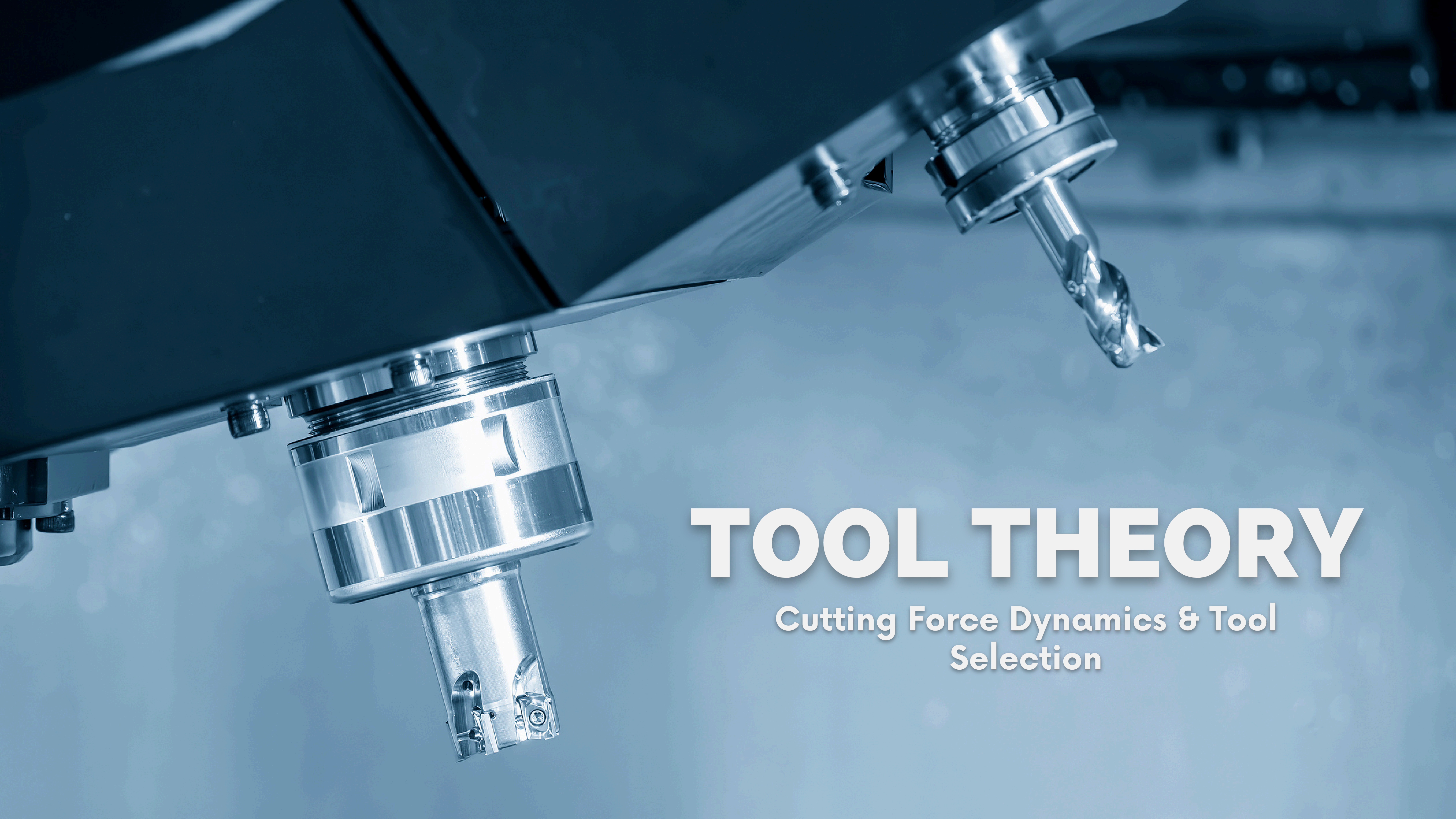
Note: This torque also applies to the 10K grease-packed spindle option.

1/29/2016

WE'RE NOT ACCOUNTING FOR SOMETHING...

We are not considering the cutting forces at the tool cutter, we must also consider the flute engagement with the stock material explicitly. We must consider phenomena such as chip thinning, tool deflection, and frictional forces.

We need to correct for this...


TOOL THEORY
Cutting Force Dynamics & Tool Selection

CUTTING FORCE DYNAMICS ARE HARD!

"The actual force on the machine tool spindle during machining includes the cutting force caused by material removal... However, these factors lead to a nonlinear modeling of the spindle system, and it is **very difficult** to measure or identify each parameter precisely. A simplified model is proposed by introducing the shear correction coefficient βQc and the edge correction coefficient βQe(Q= t, r) are introduced to represent the influence of machine tool on the cutting force. Therefore, the new cutting force model can be expressed as:

$$\begin{bmatrix} F_{x,j,m}(t,z) \\ F_{y,j,m}(t,z) \end{bmatrix} = \int_0^{a_p} g(\phi) T_j \left\{ \begin{bmatrix} \beta_{tc} K_{tc} \\ \beta_{rc} K_{rc} \end{bmatrix} h(\phi) + \begin{bmatrix} \beta_{te} K_{te} \\ \beta_{re} K_{re} \end{bmatrix} \right\} dz$$

where βtc, βte, βrc, βre are the machine tool correction coefficients, which are utilized to reflect the characteristic of the machine tool system, and can be determined by milling tests."

CUTTING FORCE DYNAMICS

Understanding cutting force dynamics is critical for optimizing machining processes. Forces acting at the cutter directly impact power requirements, tool life, and part quality. By examining the underlying mechanics, we can address discrepancies in calculated horsepower, manage tool deflection, and optimize coolant strategies, tool geometries, and machining parameters.

Why HP Values Differ

Manual horsepower (HP) calculations at the motor underestimate or oversetimate real-world power requirements because:

- They rely on simplified Material Removal Rate (MRR) calculations.
- They omit dynamic factors like chip thinning and tool geometry.
- Torque and cutting force variations at specific RPMs aren't considered.

Forces at the Cutter

Cutting forces arise from material resistance and tool engagement at three distinct axes:

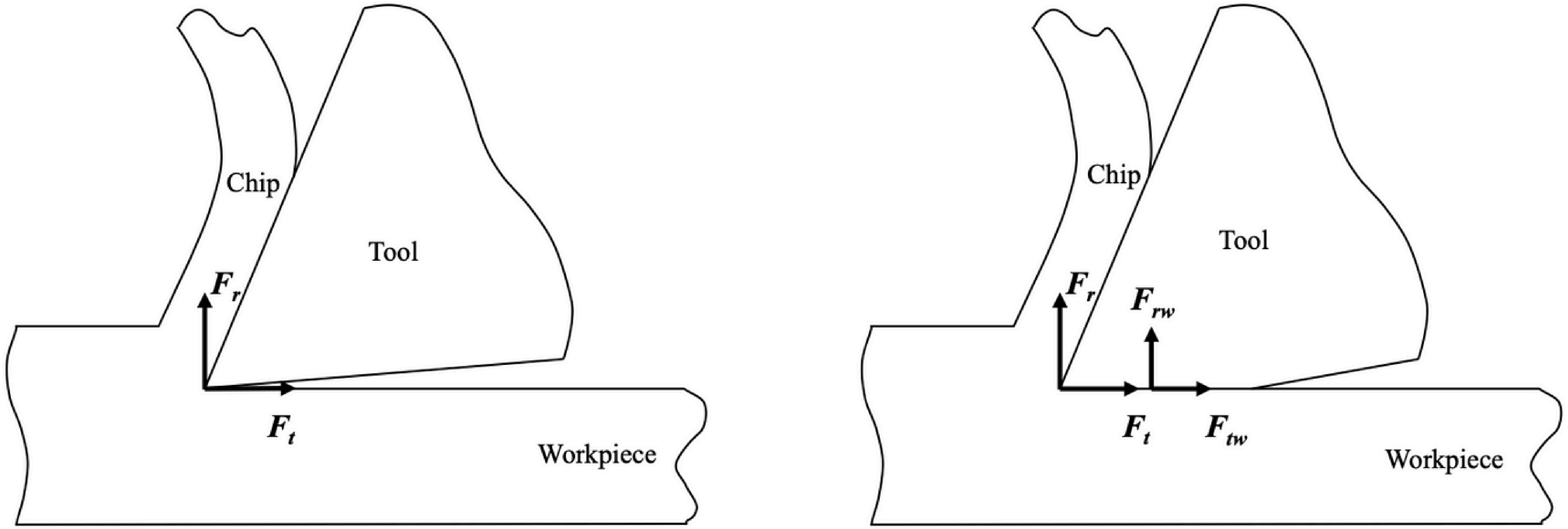
- **Tangential Force (Ft):** Primary force responsible for material removal.
- **Radial Force (Fr):** Acts perpendicular to the cutter, contributing to tool deflection.
- **Axial Force (Fa):** Acts parallel to the cutter axis, influencing tool stability.

Using Kennametal's calculator:

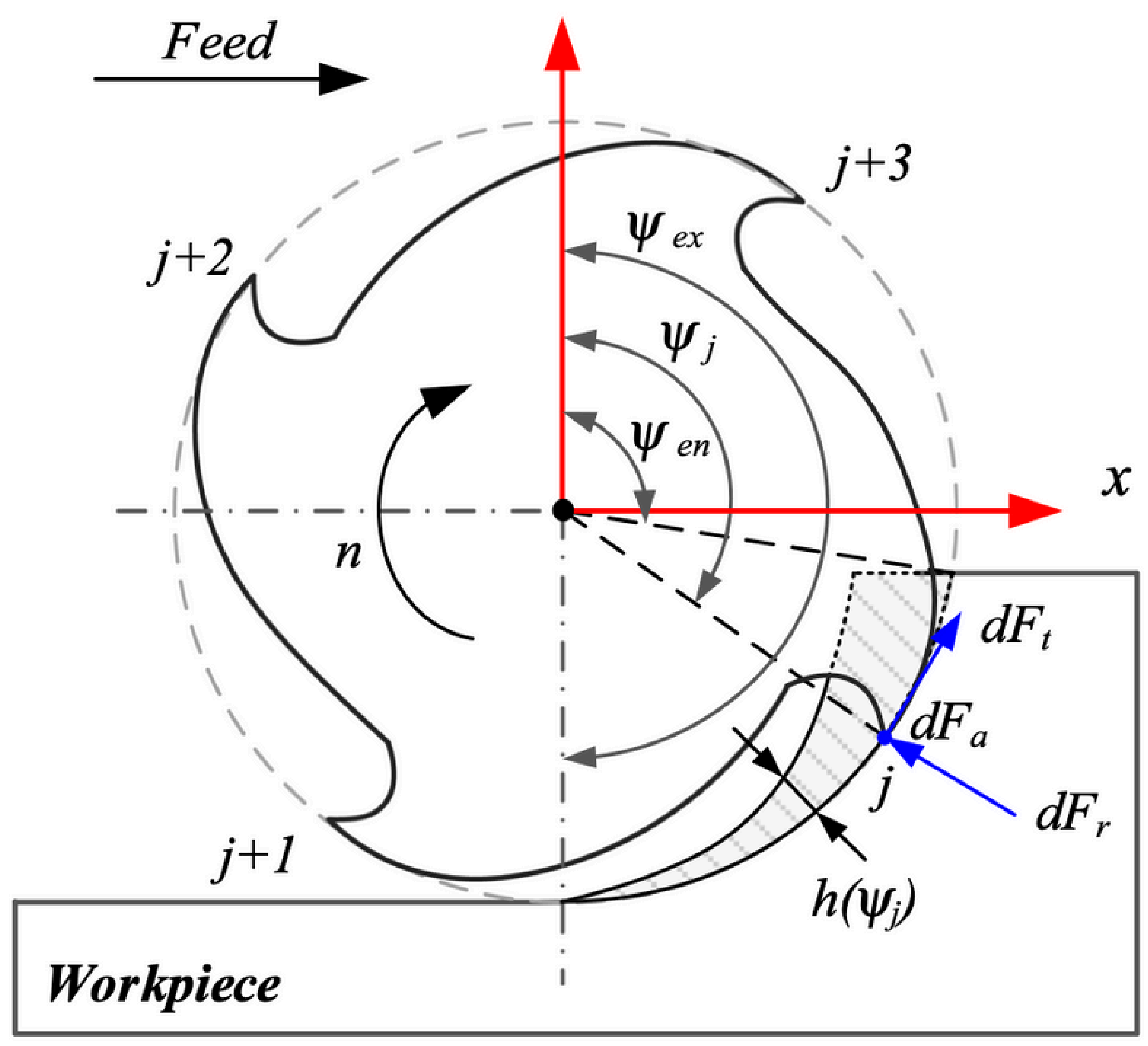
- Tangential Force (Ft): Accounts for chip thickness and cutting speed.
- **Power at the Cutter (Pc):** Reflects torque and radial engagement, providing more realistic values.

2D CUTTING FORCE DYNAMICS

Fig. 3 The influence of tool wear on cutting force



2D CUTTING FORCE DYNAMICS



2D CUTTING FORCE DYNAMICS

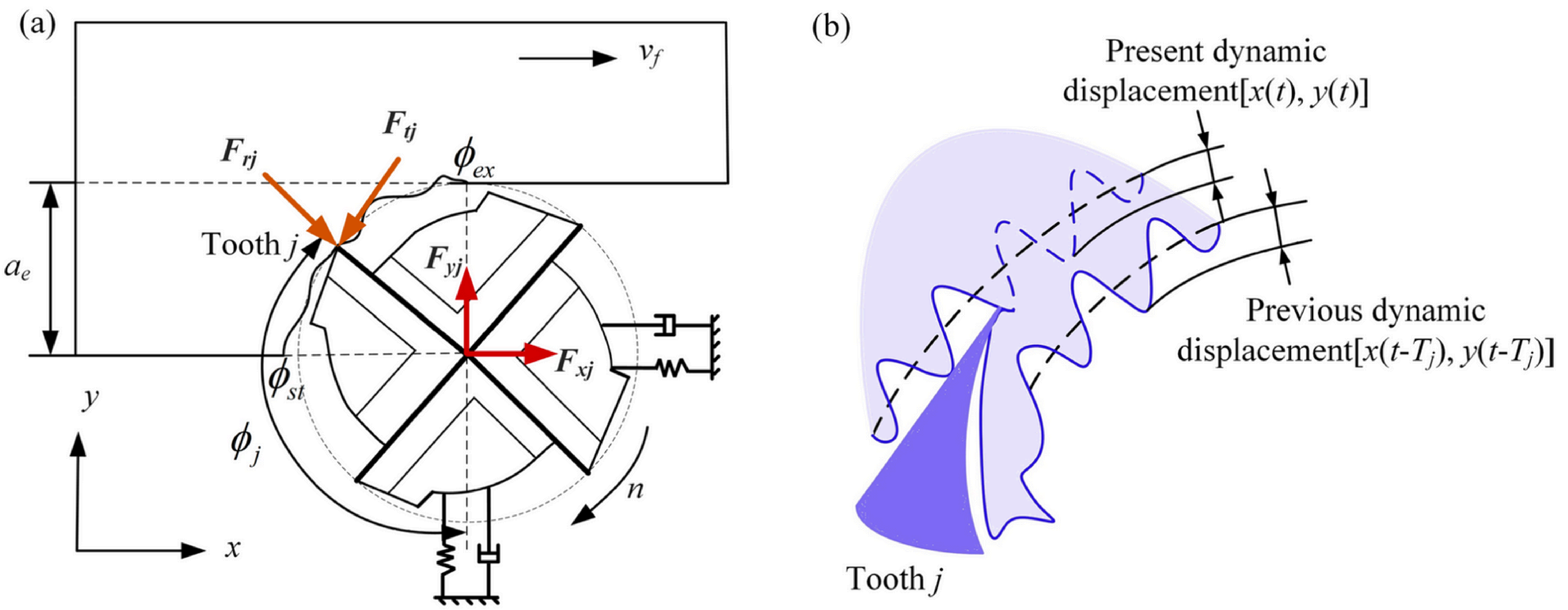
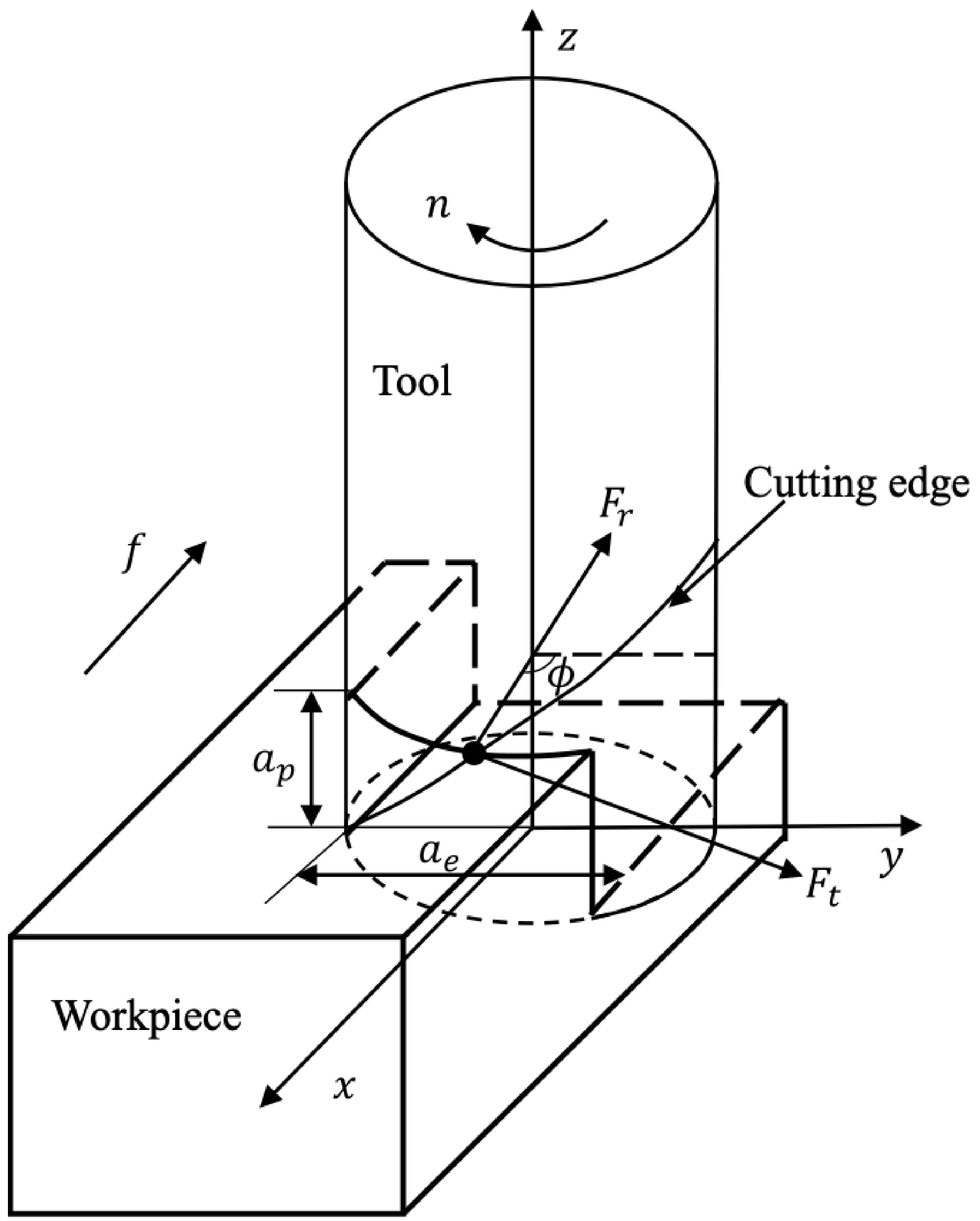
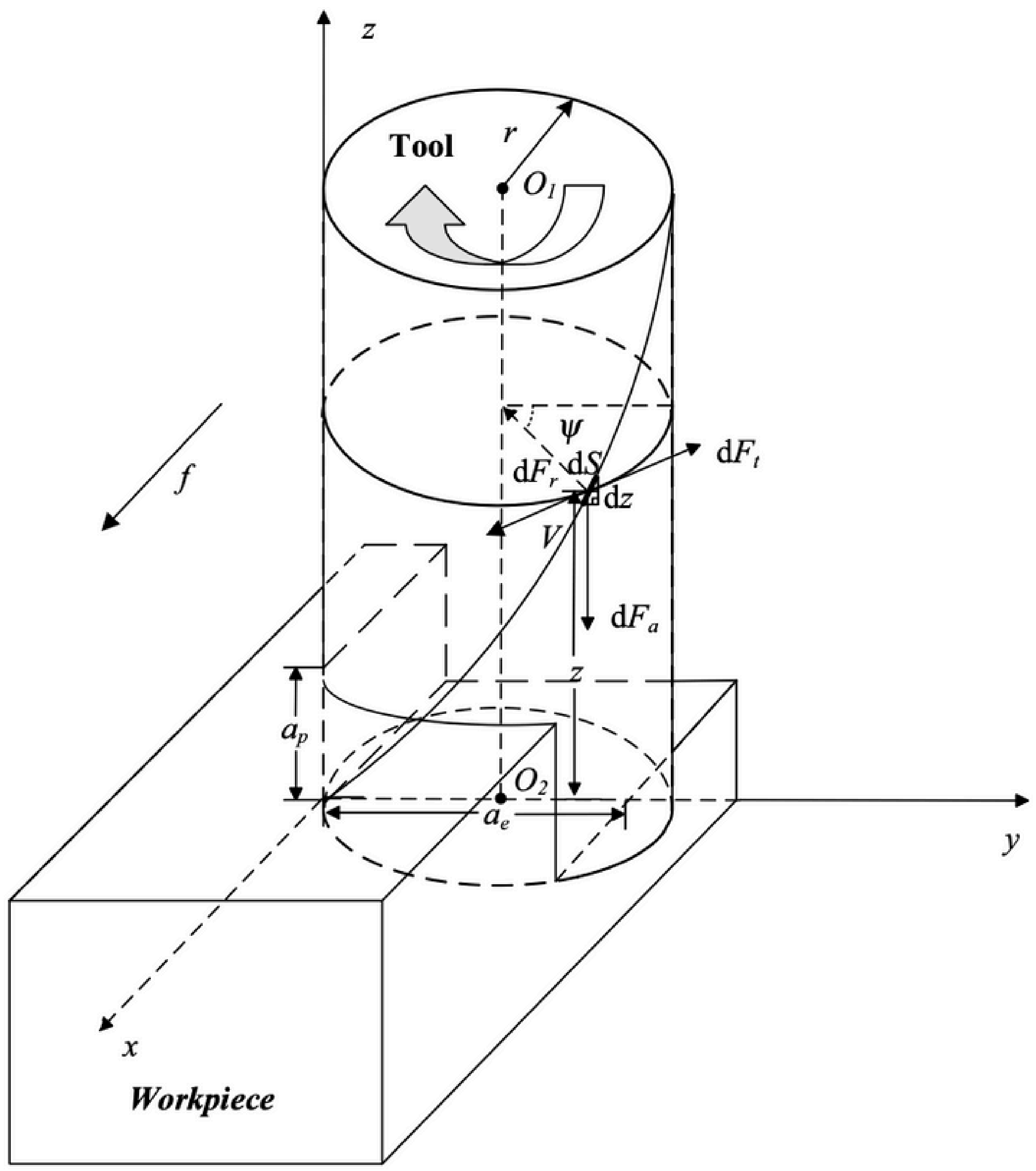


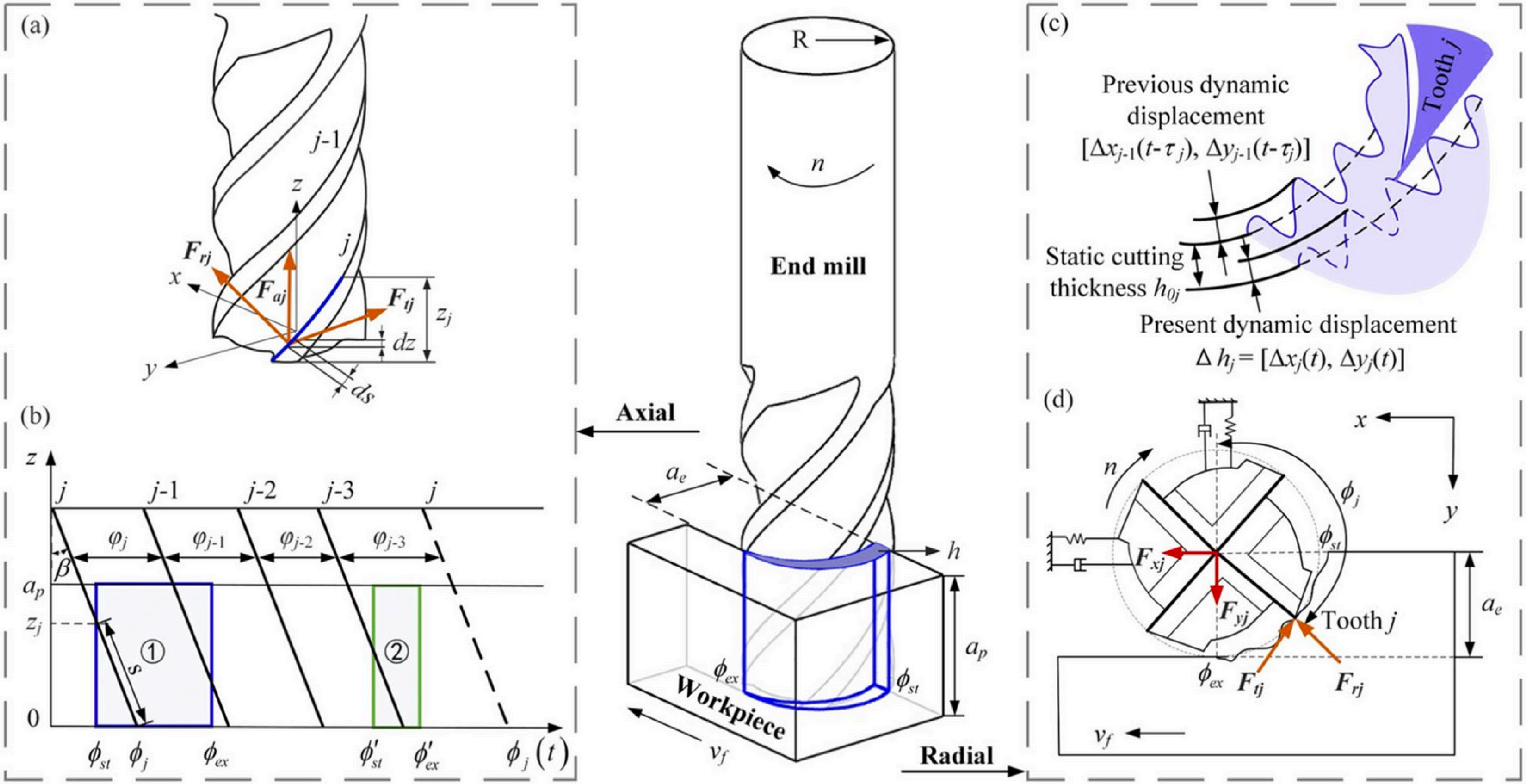
Fig. 1 Dynamic model of end mill with 2-DOF system. **a** Cutting force. **b** Dynamic displacement of regenerative chatter

CUTTING FORCE DYNAMICS

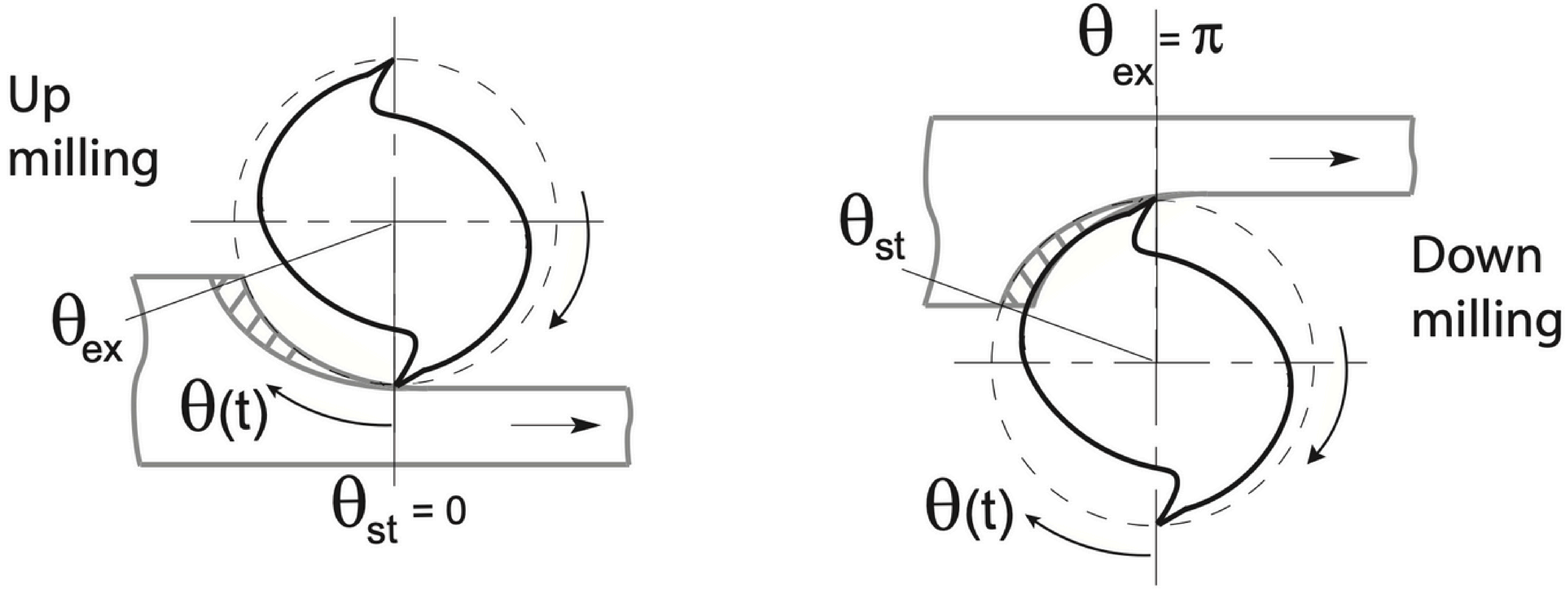




CUTTING FORCE DYNAMICS



CUTTING FORCE DYNAMICS



TOOL DEFLECTION AND CHIP DYNAMICS

Forces at the Cutter

Tool deflection occurs when cutting forces exceed the tool's rigidity, leading to bending and inaccuracy.

Factors Influencing Deflection:

- Cutting forces (Ft, Fr, Fa).
- Tool stickout: Longer tools increase deflection risk.
- Tool material: Carbide tools resist deflection better than HSS.

Impact of Deflection:

- Reduced dimensional accuracy and poor surface finish.
- Increased risk of tool wear and breakage.

Chip Thinning

Chip thinning occurs when radial engagement (WOC) is smaller than the tool's edge radius, reducing the chip's actual thickness.

- Reduced cutting forces and heat generation.
- Increased tool life and higher feed rates.

Compensation:

- Adjust feed rate to maintain optimal chip load.

CORRECTION FACTORS

Forces at the Cutter

Two additional factors may further refine the horsepower calculation:

- K Factor: Accounts for tooling effects (e.g., sharpness, geometry, coatings). Typical K values range from 1.0–1.5, with sharper tools closer to 1.0 and duller tools closer to 1.5.
- C Factor: The Feed Factor, often denoted as C, is an empirically derived constant used in machining power equations to relate material removal rates (MRR) to the power required for a cutting process.

$$P_m = \frac{UP \times MRR}{E} \times DC \times K \times C$$

Above is the complete theoretical equation for the power requirement at the spindle motor of the machine, including the duty cycle, DC parameter, the tool wear factor, K, and the feed factor, C.

The Feed Factor C is a critical component in the empirical equations used to estimate the power needed for material removal in machining processes. It integrates multiple factors—ranging from material and tool characteristics to cutting conditions and machine dynamics—into a single constant that can help predict and optimize the machining power requirements. And as such, is highly impractical for practical use.

JUST USE THE CALCULATORS🙄

KENNAMETAL®

End Milling Force, Torque, and Power Calculator

Member IMC Group

ISCAR

Machine Power Calculator

M DOCTOR

Machining Power Calculator and Formulas

CUSTOMPART.NET

Milling Horsepower Calculator

UNDERSTANDING FLUTE NUMBERS

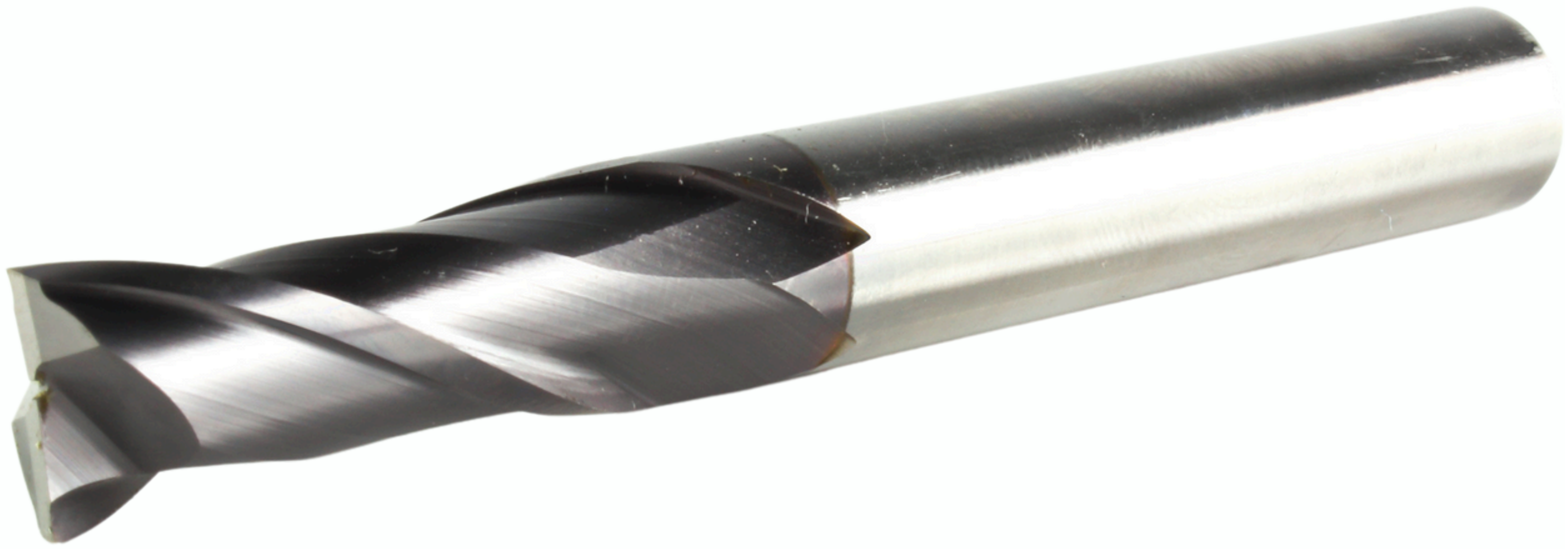
Flute Number

The number of flutes on a cutting tool, such as an end mill, refers to the number of cutting edges or channels that extend along the length of the tool. Flutes serve two primary purposes:

- **Material Removal:** Cutting edges on the flutes shear material from the workpiece.
- **Chip Evacuation:** The spaces between flutes provide a path for chips to exit the cutting area.

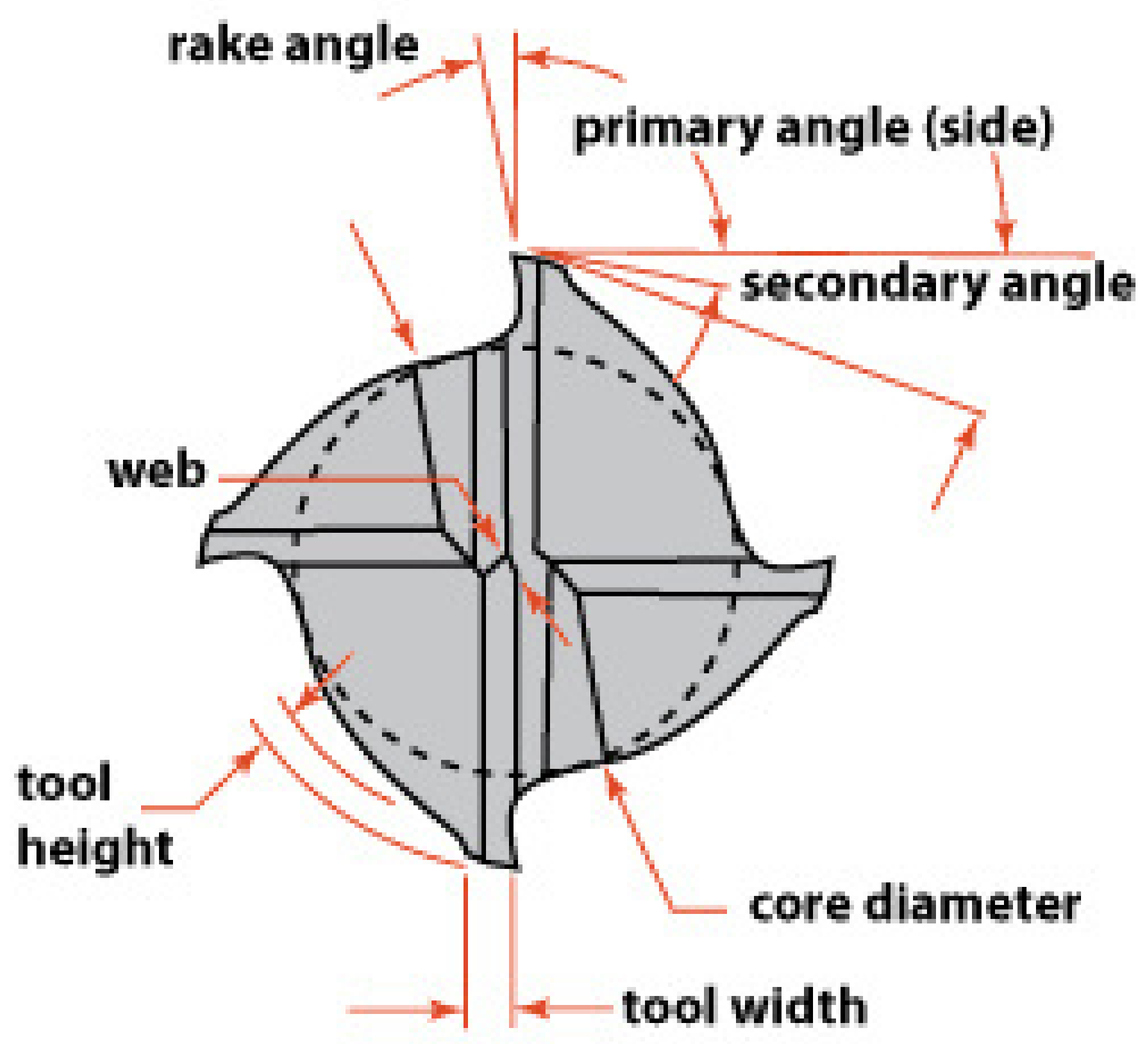
Material Being Machined

- **Soft Materials (e.g., Aluminum, Plastics):** Fewer flutes (typically 2 or 3) provide larger chip clearance, preventing clogging and ensuring efficient chip evacuation.
- **Hard Materials (e.g., Steel, Titanium):** More flutes (4 to 6 or more) allow greater cutting edge contact, improving surface finish and distributing cutting forces, though chip evacuation must be managed.



RAKE ANGLE

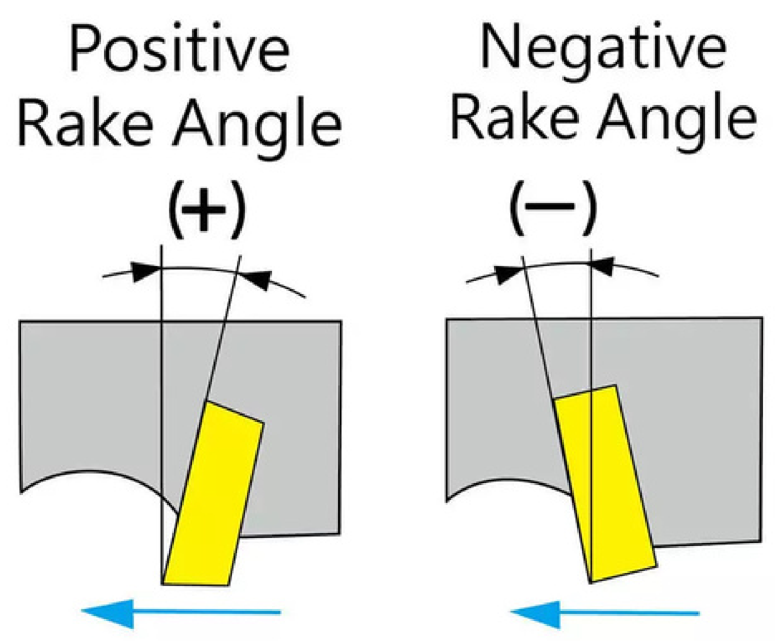
The rake angle of a cutting tool refers to the angle between the cutting edge and the tool's centerline or axis. It determines how the material is sheared during cutting and greatly influences cutting forces, chip flow, and tool performance.



Clearance Angle

The angle formed by the cleared surface and line tangent to the cutting edge. Is normally tilted forward, creating a sharper edge.

- Clearance: Primary (first angle 5 – 9 degree) – Relief adjacent to the cutting edge.
- Clearance: Secondary (2nd angle 14 – 17 degrees) – Relief adjacent to the cutting edge.
- Clearance: Tertiary (3rd) – Additional relief clearance provided adjacent to the secondary angle.



EFFECT OF FLUTE NUMBER AND RANKE ANGLE ON CUTS

- **Slotting:** Use fewer flutes for better chip clearance in deep cuts.
- **Finishing:** Use more flutes for higher surface quality.

Tools with fewer flutes can support higher feed rates because each flute removes more material. Tools with more flutes may require slower feed rates but offer finer finishes.

- Fewer flutes with a positive rake angle are ideal for aluminum and plastics, promoting chip evacuation and reducing cutting forces.
- More flutes with a shallower rake angle are better for hard materials like steel, providing strength and higher cutting edge engagement.
- Balance flute number and rake angle based on machining requirements, surface finish needs, and material properties for optimal performance.

ROUGHING ENDMILLS

Roughing endmills are ideal for quickly removing large amounts of material during the initial machining stages. Their serrated cutting edges break chips into smaller pieces, reducing cutting forces and promoting better chip evacuation.

- Serrated edges for efficient material removal and reduced cutting forces especially for higher flute counts.
- Best for roughing operations with less emphasis on surface finish.

Adaptive Clearing:

- Used for roughing out large amounts of material while maintaining consistent chip load.

2D Pocket Clearing:

- Quickly removes material from flat-bottomed pockets.

2D Contour Roughing:

- Great for roughing external or internal contours where precise finishing isn't required.

FINISHING ENDMILLS

Finishing endmills are used for the final machining pass to achieve tight tolerances and smooth surface finishes. These tools have smooth cutting edges, higher flute counts (4–8 or more), and sharp edges for precise cuts. They are ideal for shallow cuts.

- Smooth edges for clean cuts and fine surface finishes.
- Suitable for tight tolerances and detailed geometries.
- Used in shallow, precise finishing passes.

2D Contour Finishing:

- Precisely machines external and internal contours.

Horizontal (Flat Area Clearing):

- Fine-tunes flat areas for smooth finishes.

COATINGS

Tool coatings enhance the performance, lifespan, and efficiency of cutting tools by improving properties like wear resistance, heat resistance, and friction. Each coating type has distinct characteristics that make it suitable for specific applications and materials. Below is an analysis of common tool coatings, their properties, and how to select the right coating.

Material Being Machined

- Non-Ferrous Metals (Aluminum, Brass, Copper):
  - Coatings like ZrN and DLC prevent adhesion and improve surface finish.
- Hard Metals (Steel, Titanium):
  - Use AlTiN, TiAlN, or nACRo for their heat and wear resistance.
- Abrasive Materials (Composites, Graphite):
  - DLC or CrN coatings are excellent choices.

COATINGS HONORABLE MENTIONS

Zirconium Nitride (ZrN)

- Properties:
  - High corrosion resistance.
  - Provides excellent chip flow due to a smooth surface.
- Best For:
  - Non-ferrous materials (e.g., aluminum, brass, plastics).
  - Applications requiring minimal adhesion and buildup.

Titanium Nitride (TiN)

- Properties:
  - General-purpose coating with good wear resistance.
  - Increases tool hardness and reduces friction.
- Best For:
  - Mild steels, cast iron, and other general-purpose machining.
  - Applications with moderate speeds and feeds.

NanoComposite Coating (nACRo)

- Properties:
  - A composite of aluminum chromium nitride (AlCrN) with nano-structured ceramics.
  - Excellent thermal stability and high wear resistance.
- Best For:
  - High-performance machining in aerospace and automotive applications.
  - Hard materials like hardened steels and nickel alloys

Chromium Nitride (CrN)

- Properties:
  - Excellent adhesion and corrosion resistance.
  - Moderate hardness and wear resistance.
- Best For:
  - Applications involving non-ferrous metals and plastics.

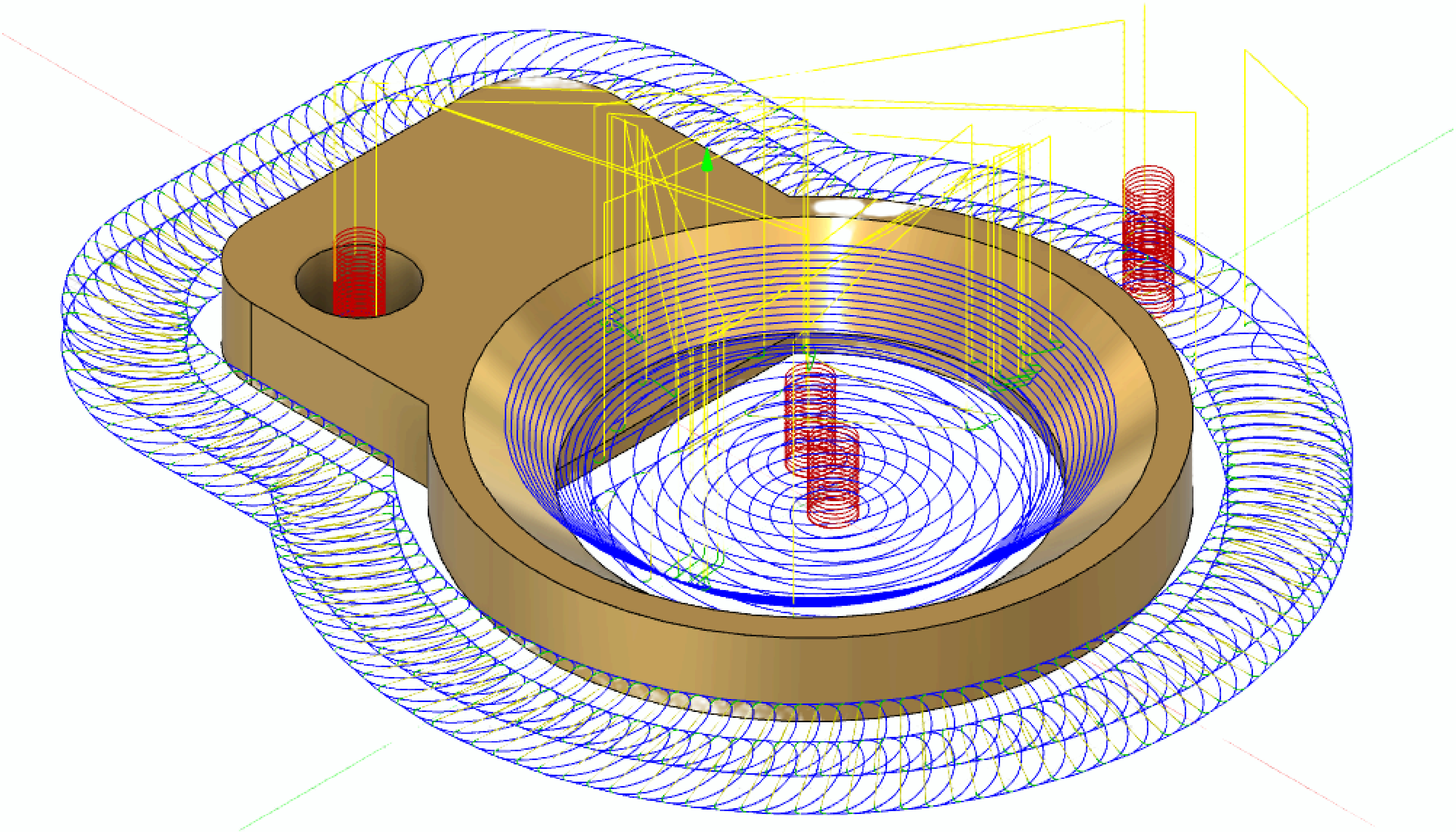
THE PROCESS

WHAT IS CAM?

(Computer-Aided Manufacturing) is the use of software to create toolpaths and instructions for machining parts with CNC machines.

Learning advanced CAM techniques equips us to handle:

- complex geometries
- optimize machining efficiency
- prevent and catch fundamental mistakes in students machining plans



OBJECTIVES

- Gain an understanding of the foundational and advanced principles of CAM.
- Apply the speeds and feeds that we have just calculated into CAM to optimize machining processes.
- Explore innovative workflows to improve accuracy and efficiency.

The manufacturing industry is evolving rapidly with advancements in CAM technology, and students are steadily asking for more help with more advanced geometries.

Modern CAM software integrates AI and adaptive toolpaths, addressing challenges like tool wear, vibration, and part consistency. We won't be covering many of the newer advancements, but we will cover the basics of advanced CAM.

DEFINING OUR TERMS

High-Efficiency Machining (HEM): Advanced machining strategies designed to maximize material removal rates while minimizing tool wear, heat generation, and machine stress. These techniques focus on balancing speed, precision, and tool longevity.

High-Speed Machining (HSM): Techniques focused on high cutting speeds, low depths of cut, and rapid feed rates to improve efficiency and reduce cycle times.

Constant Tool Engagement: A machining approach where the tool's contact with the material remains consistent throughout the operation. This helps avoid spikes in cutting forces, improving tool life and maintaining stability.

DEFINING OUR TERMS

Chip Load: The amount of material removed per cutting edge per revolution, influencing tool wear and machining performance.

Radial Engagement: The percentage of the tool diameter engaged with the material, affecting cutting forces and tool life.

Climb Milling vs. Conventional Milling: Climb milling cuts with the tool moving in the same direction as the feed, producing better surface finishes, while conventional milling cuts against the feed direction.

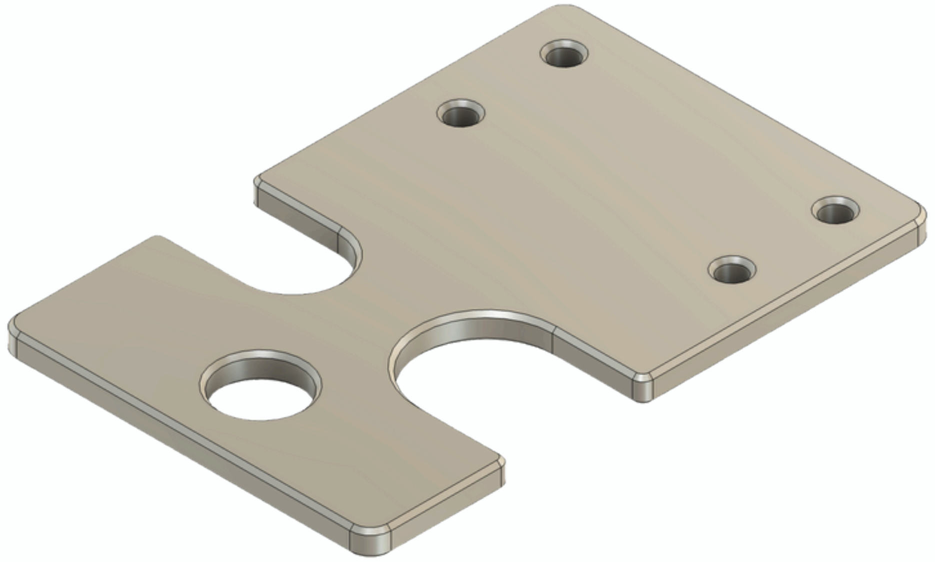
DEFINING OUR TERMS

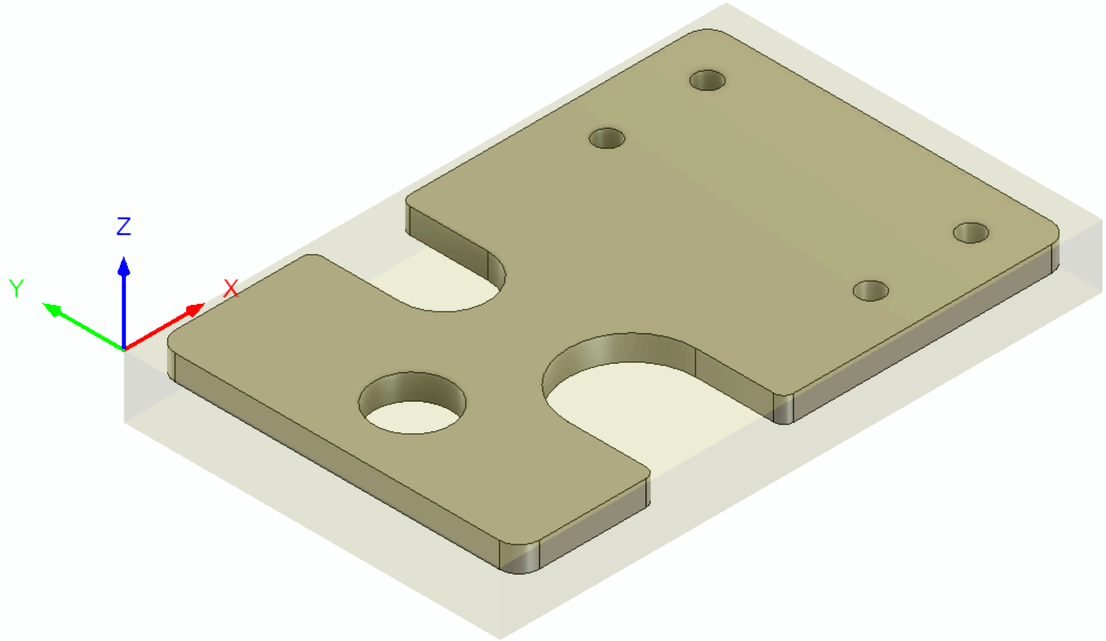
Cutting Speeds: The speed at which a cutting tool moves across the material being machined, typically measured in surface feet per minute (SFM) or meters per minute (m/min). Proper speeds are crucial for efficient machining and tool longevity.

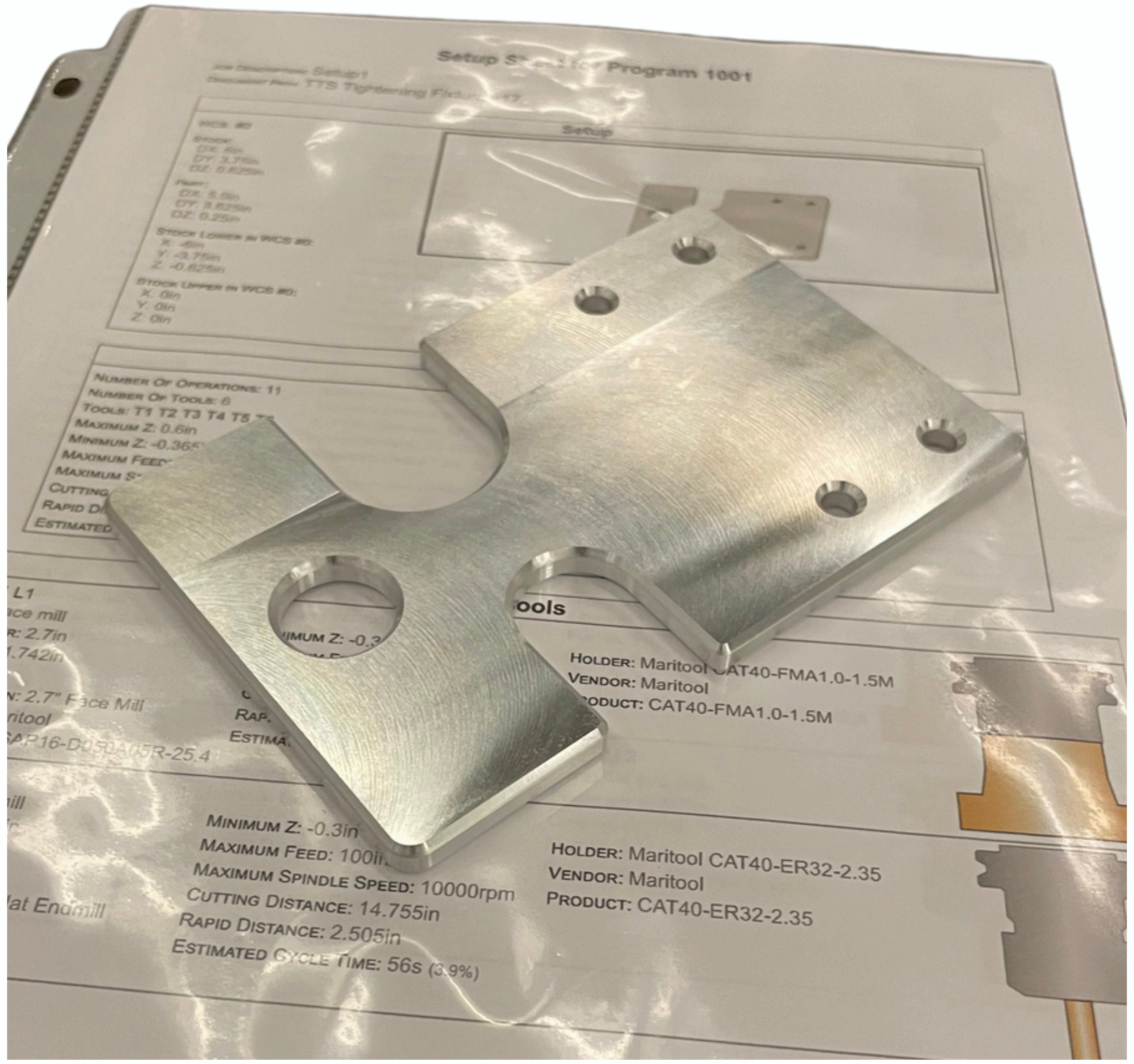
Optimized Stepovers: The controlled lateral distance that a cutting tool moves between passes. Optimizing stepovers ensures efficient material removal while maintaining part accuracy and reducing machining time.

Tool Wear: The gradual degradation of a cutting tool's sharpness or structural integrity due to heat, friction, and mechanical stress during machining operations.

PRACTICE PIECE



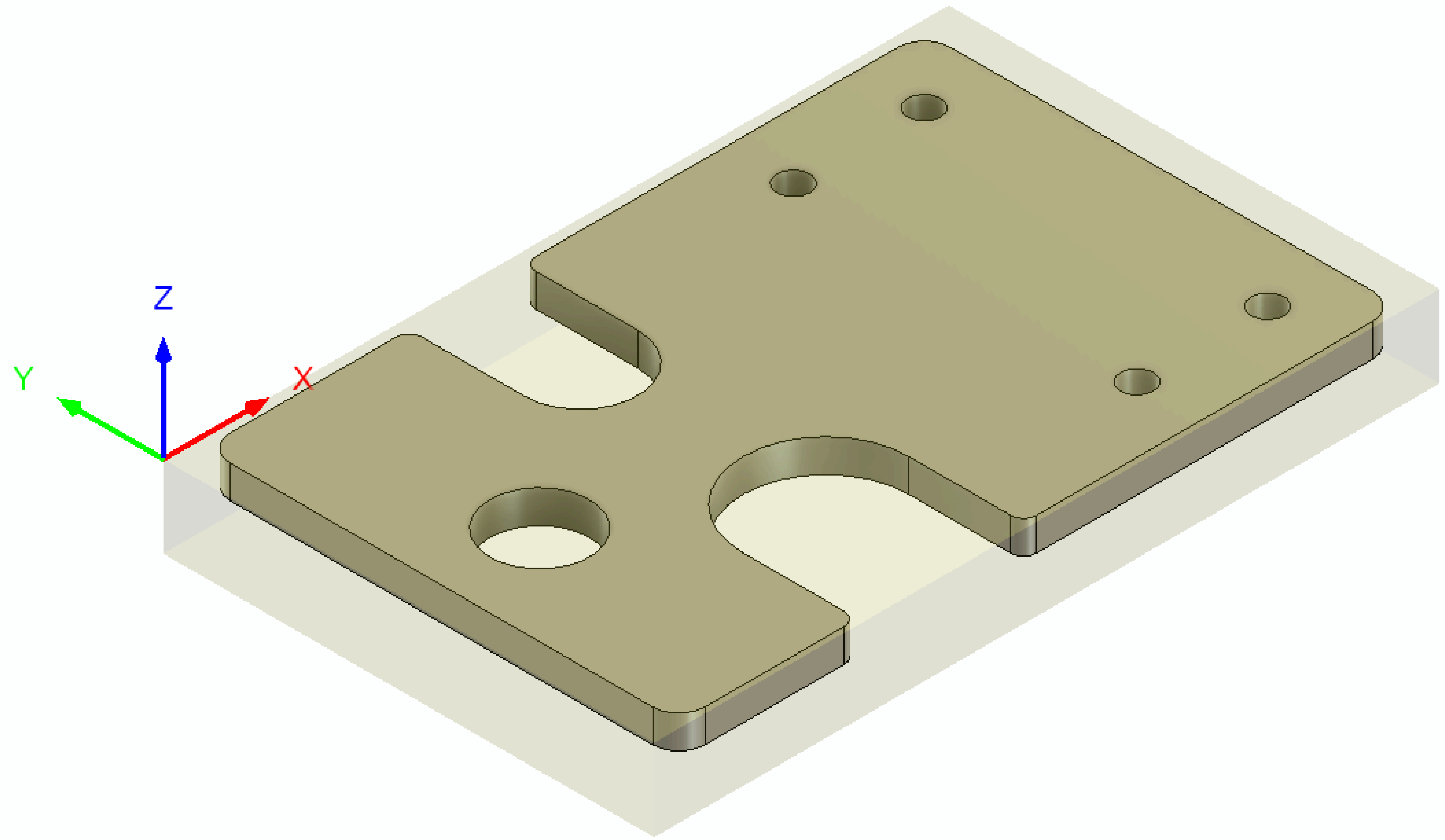




PRACTICE PIECE

Machining Plan

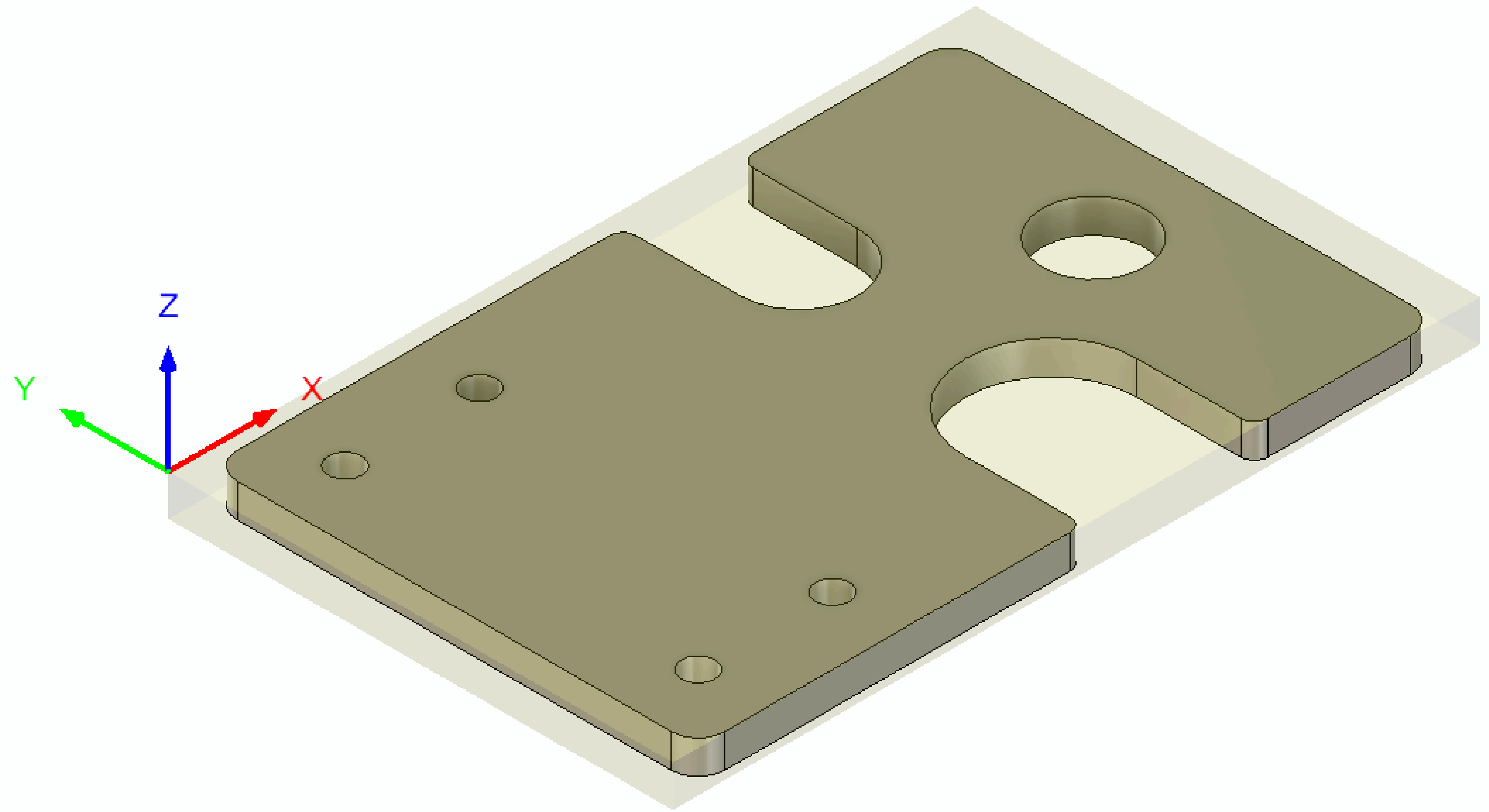
- Setup 1
  - Face
  - 2D Pocket
  - 2D Contour
  - 2D Adaptive
  - 2D Contour
  - Spot Drill
  - Drill
  - 2D Chamfer



PRACTICE PIECE

Machining Plan

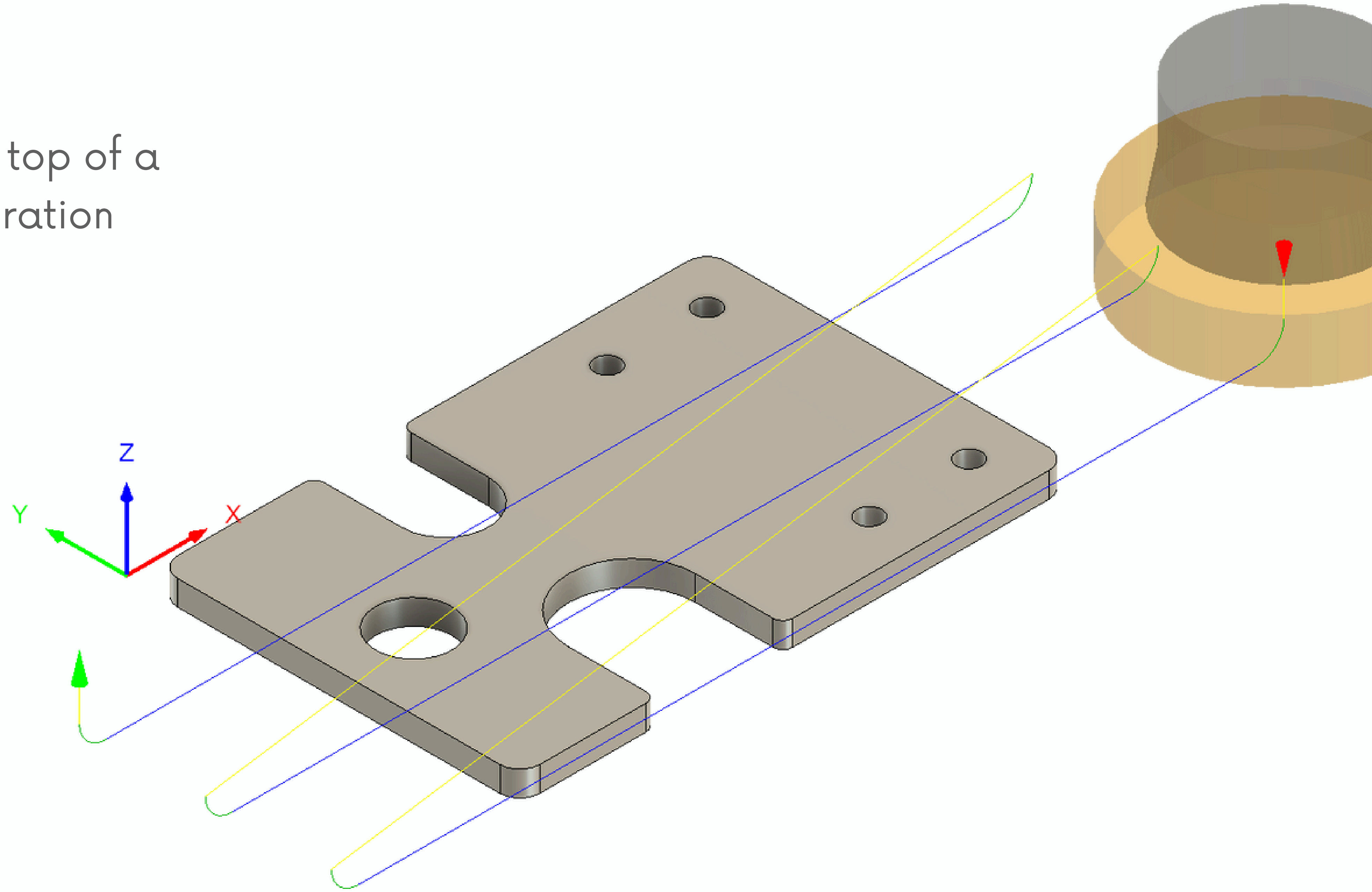
- Setup 2
  - Face
  - 2D Adaptive
  - 2D Chamfer



ADVANCED CAM TOOLPATHS

Face

The Facing Operation is used to machine flat surfaces on the top of a part, often as the first step in the machining process. This operation ensures the stock surface is flat and parallel to the machine's coordinate system, creating a consistent base for subsequent operations.



Efficient Surface Preparation:

- Facing quickly removes material across the entire surface, preparing the workpiece for further machining steps.

Customizable Cutting Patterns:

- The operation offers various patterns, such as zigzag or one-direction passes, allowing flexibility based on material and machine requirements.

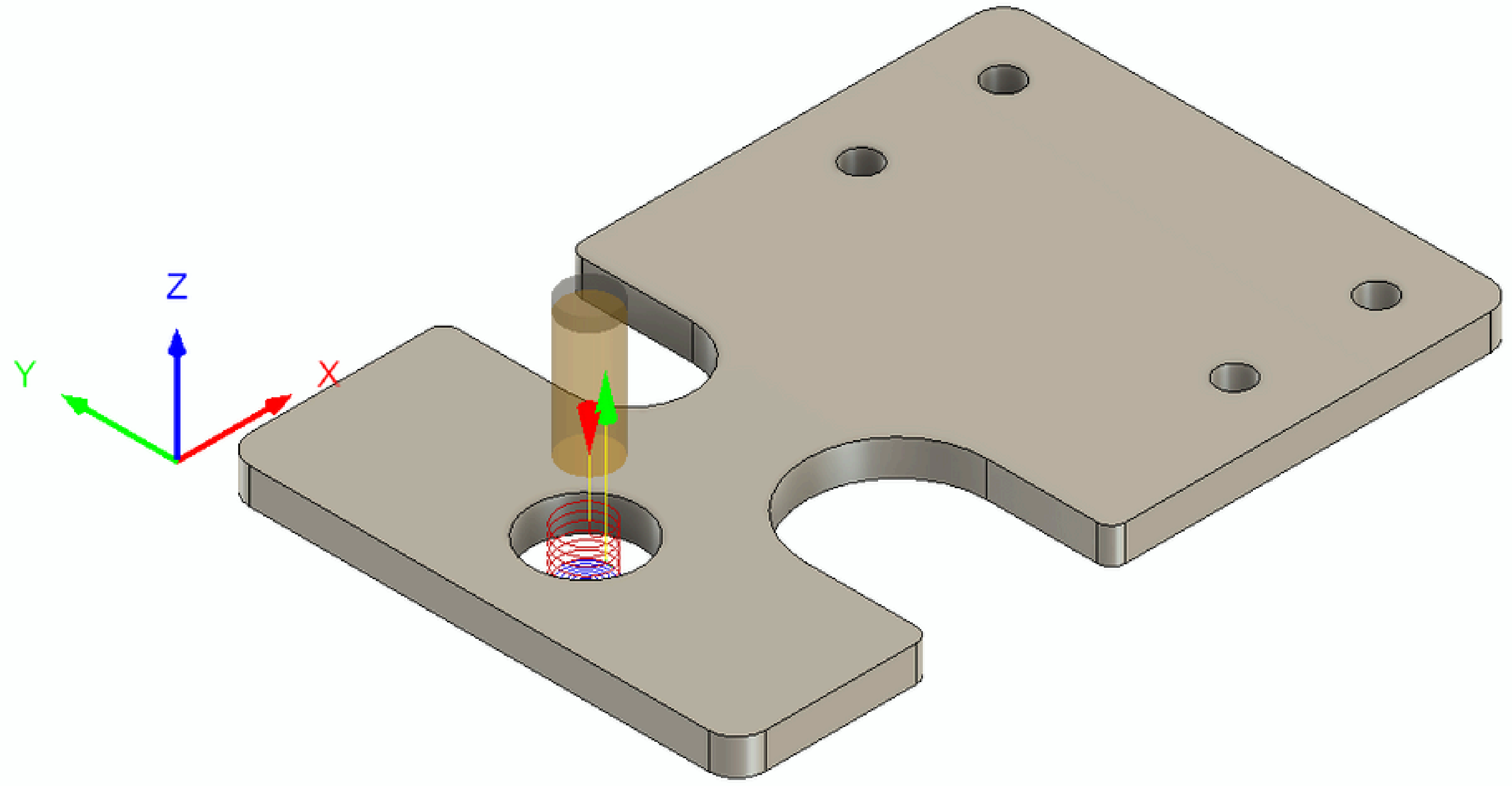
Wide Tool Compatibility:

- Facilitates the use of face mills, fly cutters, or large-diameter endmills for efficient material removal and high-quality surface finishes.

ADVANCED CAM TOOLPATHS

2D Pocket

The 2D Pocket Operation is used to remove material from flat-bottomed pockets or cavities within a part. This operation is ideal for creating internal features with vertical walls and flat floors.



Spiral Ramping:

- Spiral ramping is used to smoothly plunge the tool into the material, reducing tool stress and heat buildup compared to direct plunges.
- This technique allows for continuous cutting engagement during entry, improving tool life and creating a clean starting point for pocketing operations.

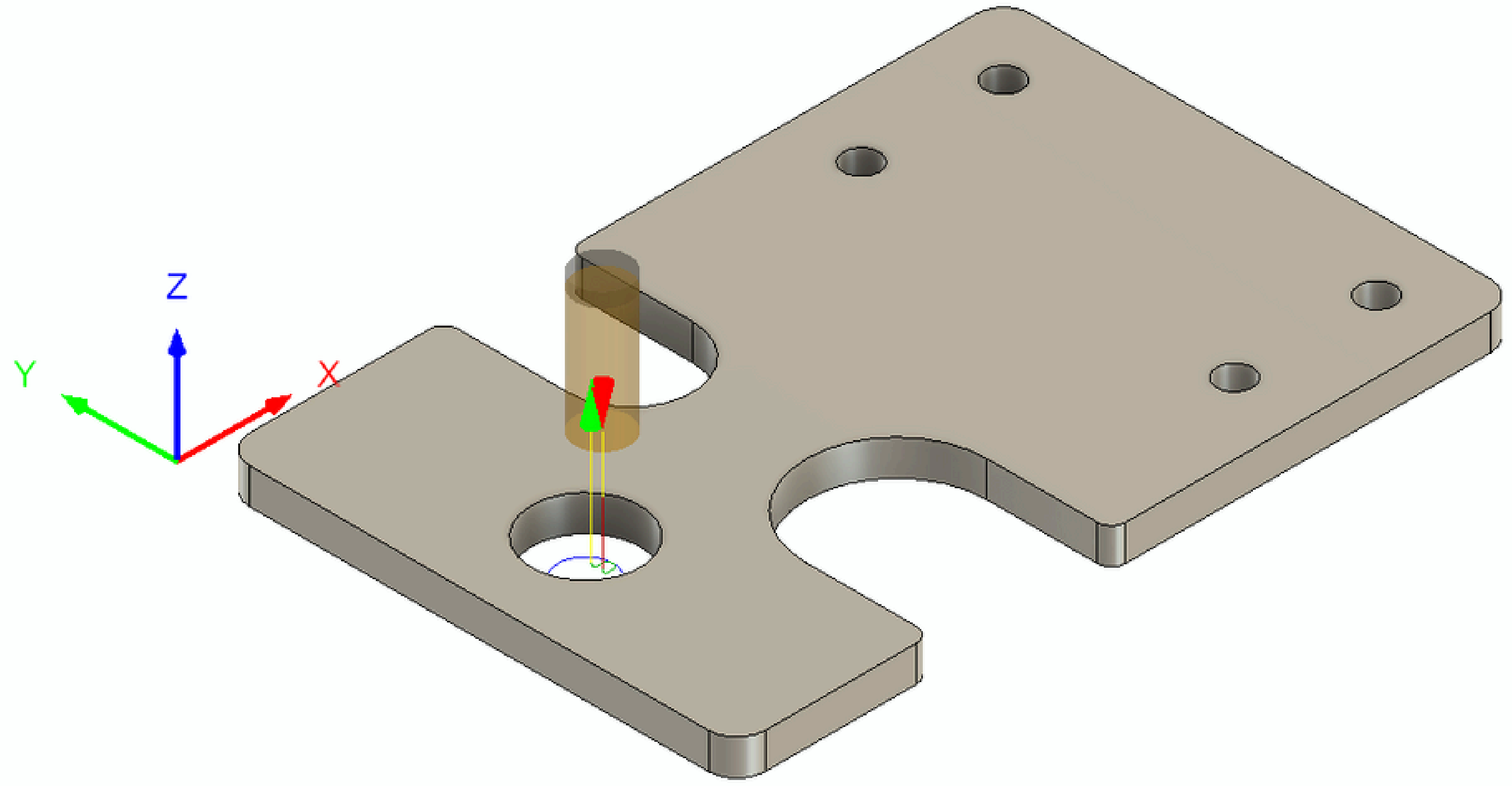
Trochoidal Material Removal:

- The tool avoids full-width cuts and only engages the material with a manageable portion of its cutting edge, ensuring smoother and more efficient cutting.
- Trochoidal toolpaths break the material into smaller segments using continuous, circular cutting motions.

ADVANCED CAM TOOLPATHS

2D Contour

The 2D Contour Operation is used to machine precise edges and profiles along the perimeter of a part or feature. It is ideal for defining part boundaries, external contours, and clean-up cuts.



Consistent Chip Load:

- The toolpath is designed to maintain a consistent chip thickness by dynamically adjusting the tool engagement angle. This reduces cutting forces and prevents tool overloading.

Optimal Tool Engagement:

- The tool avoids full-width cuts and only engages the material with a manageable portion of its cutting edge, ensuring smoother and more efficient cutting.

High Material Removal Rates:

- It prioritizes fast, efficient material removal while leaving the user the choice to leave small stock for finishing operations.

ADVANCED CAM TOOLPATHS

2D Adatpive

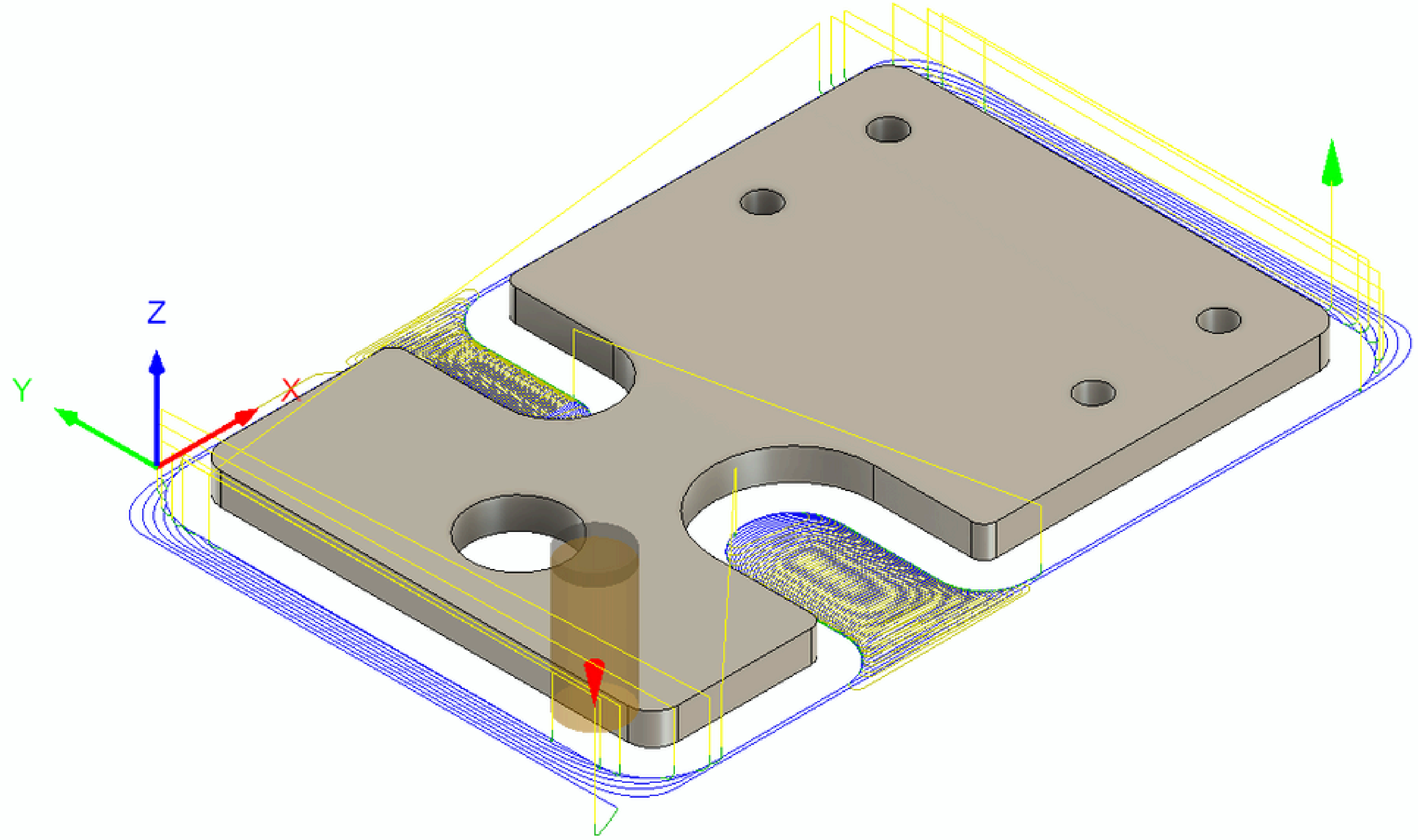
The 2D Adaptive Clearing operation in Fusion360 is a roughing strategy designed to efficiently remove material in 2D machining. It uses advanced algorithms to maintain consistent cutting conditions, minimize tool wear, and optimize material removal rates.

Accurate Edge Profiling:

- 2D Contouring follows the selected geometry closely to create sharp, clean edges and accurate profiles.

Flexible Depth Control:

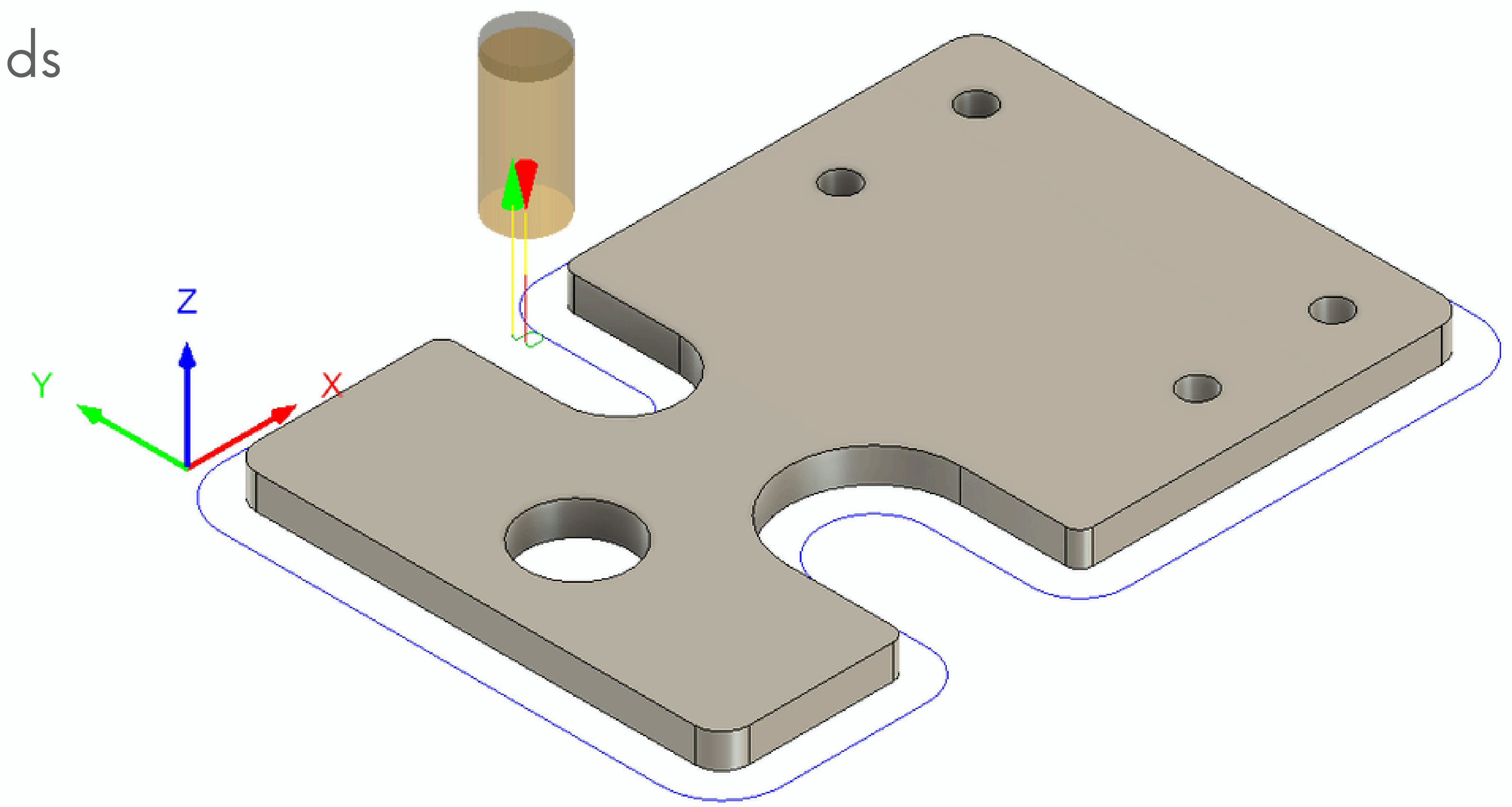
- Allows for multiple depth passes or a single pass, making it suitable for deep cuts or shallow finishing.



ADVANCED CAM TOOLPATHS

2D Contour

The final contouring in Setup 1 is performed similarly to the finishing operation on the large pocket hole. Both 2D Contour operations include a slight offset from the stock floor, ensuring the endmill avoids simultaneous contact with two surfaces, which could lead to unnecessary tool deflection and constrained movement.



Speeds and Feeds for Finishing:

- Use higher spindle speeds and lower feed rates to achieve smooth surface finishes and tight tolerances.

Tool Engagement:

- Keep the radial width of cut (WOC) small, typically 1–3% of the tool diameter, to reduce tool deflection and achieve better precision.

Stock to Leave:

- Ensure a small stock allowance (0.003–0.010 inches) is left during roughing for precise finishing.

Always climb mill for finishing!

ADVANCED CAM TOOLPATHS

2D Chamfer

The Chamfering Operation in Fusion360 is used to create beveled edges along part features, improving aesthetics, removing sharp edges, or preparing parts for assembly. It is commonly applied to internal or external edges.

Precise Edge Beveling:

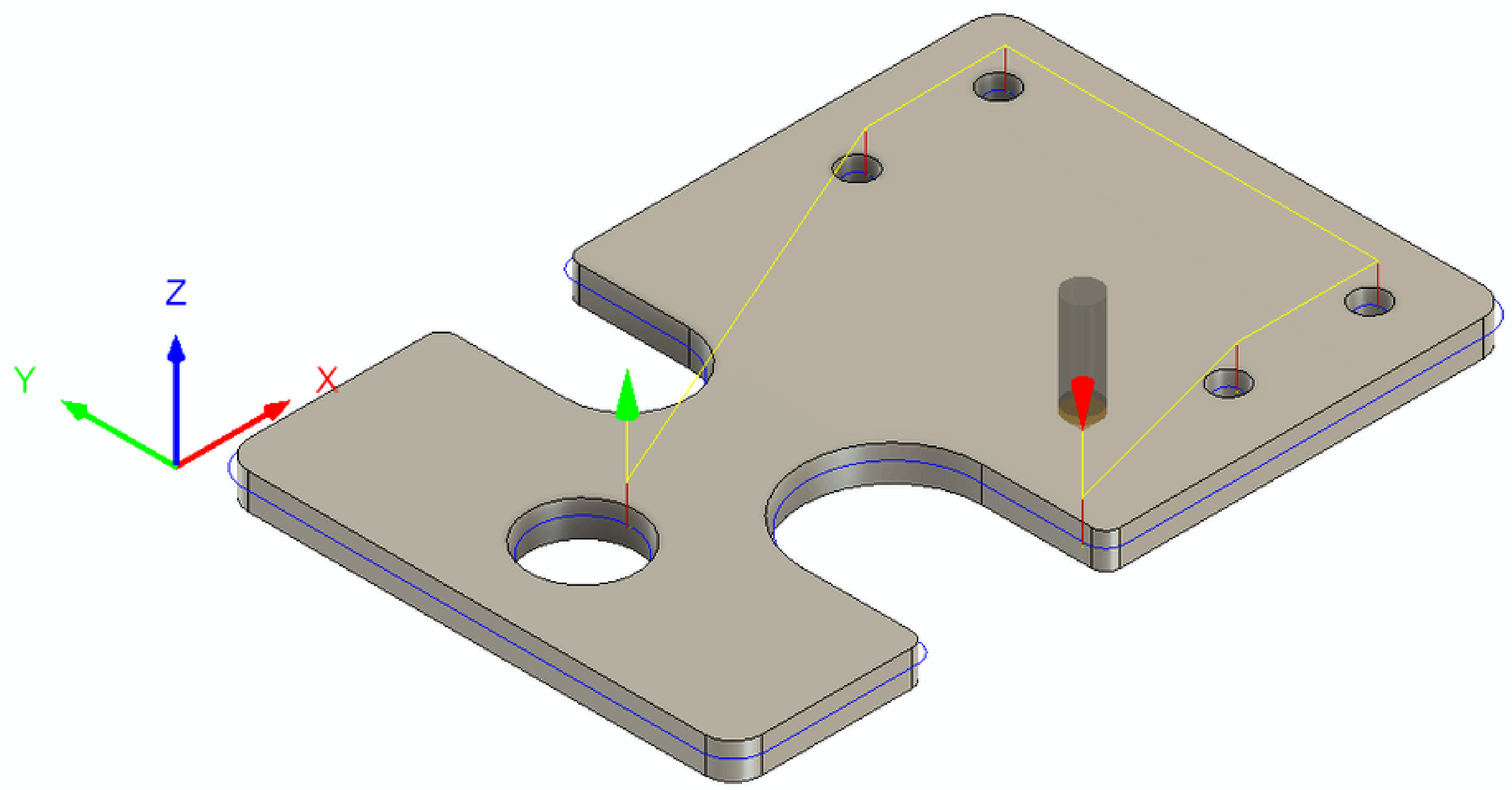
- Chamfering follows selected edges to add uniform bevels with controlled width and angle.

Customization Options:

- Allows users to adjust chamfer width, depth, and angle for specific design requirements.

Tool-Specific Operations:

- Optimized for chamfer mills or other multi-purpose tools to achieve clean, consistent edge finishes.




THANK YOU
AUTODESK®
FUSION 360™