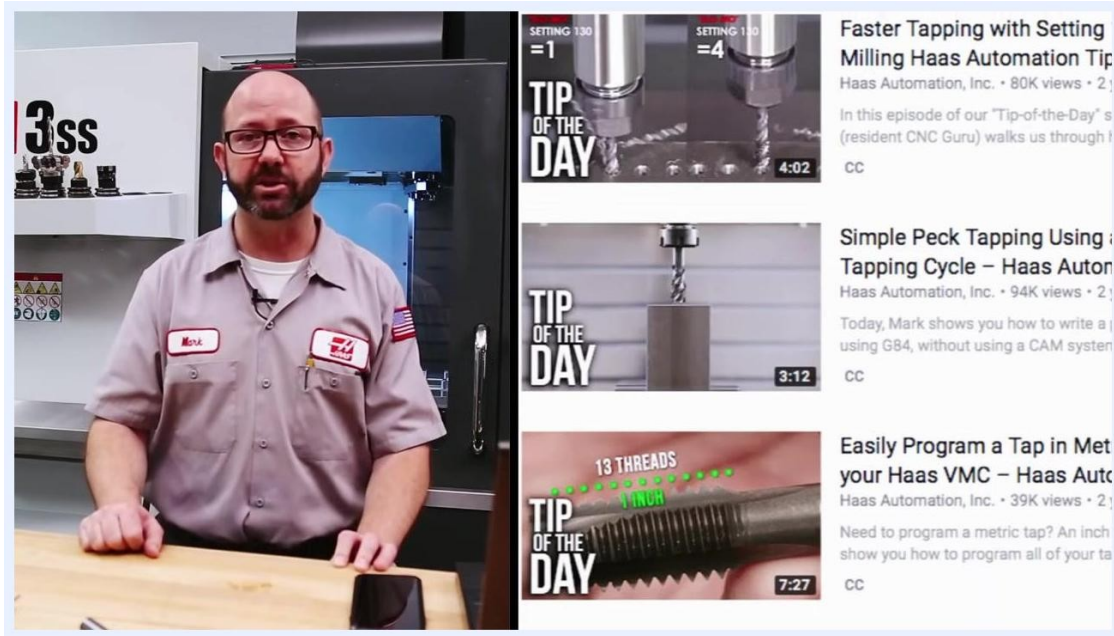


How to Calculate Speeds and Feeds for CNC Machining

This procedure explains how to calculate speeds and feeds for any tool and material combination in CNC machining. It walks through the formula used to determine the correct spindle speed (RPM) for your CNC program, along with the calculation used to establish the correct feed rate. All values and examples in this procedure use metric units; a separate reference is available if you require inch-based values.



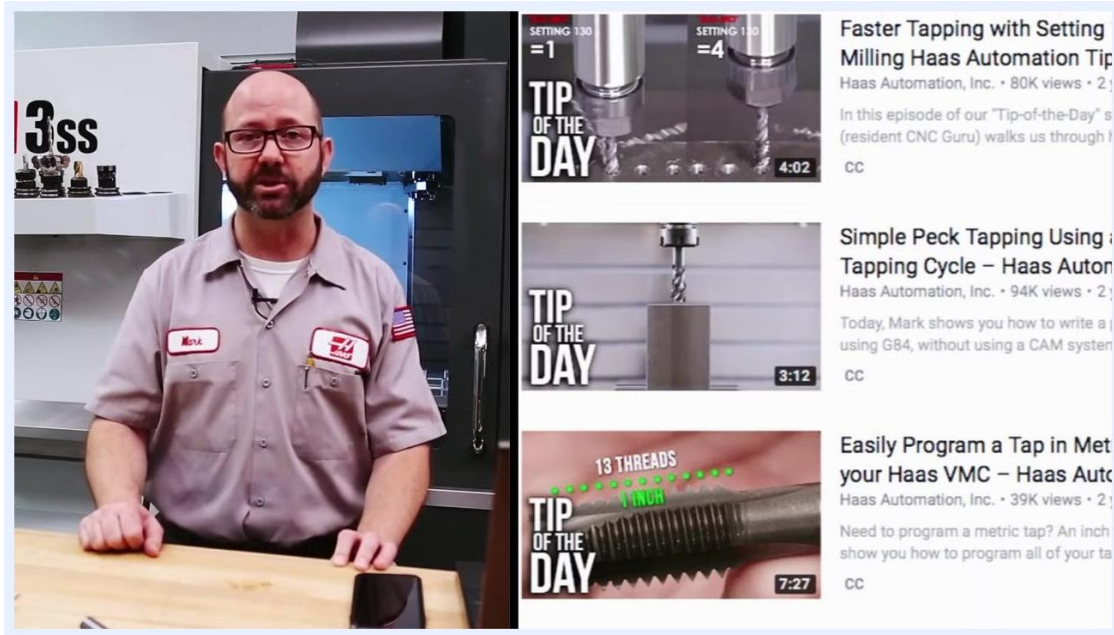
Whiteboard diagram showing a path from a house icon down to two labeled points, SPEED and FEED, drawn in a machine shop with CNC equipment and tool cabinets in the background.



Instructor seated at workbench, smiling, with laptop, model tractor, and machining tools visible, CNC machine in background

Purpose

This procedure provides a repeatable method for calculating reliable starting spindle speed (RPM) and feed rate (F-code) values before running a CNC program, so that tools are neither underused nor damaged by excessive cutting speeds.



Instructor next to a list of "Tip of the Day" videos, including tapping and feed rate calculation topics

Scope

This procedure applies to any CNC milling job where you need to determine the S-code (spindle speed) and F-code (feed rate) for a given tool and workpiece material. All calculations shown use metric units (mm, m/min, RPM).

$$\begin{array}{cc} V_c & D \\ 100 \text{ m/min} & 20 \end{array}$$
$$n = V_c \times 318 \div D$$
$$n = 100 \times 318 \div 20 =$$

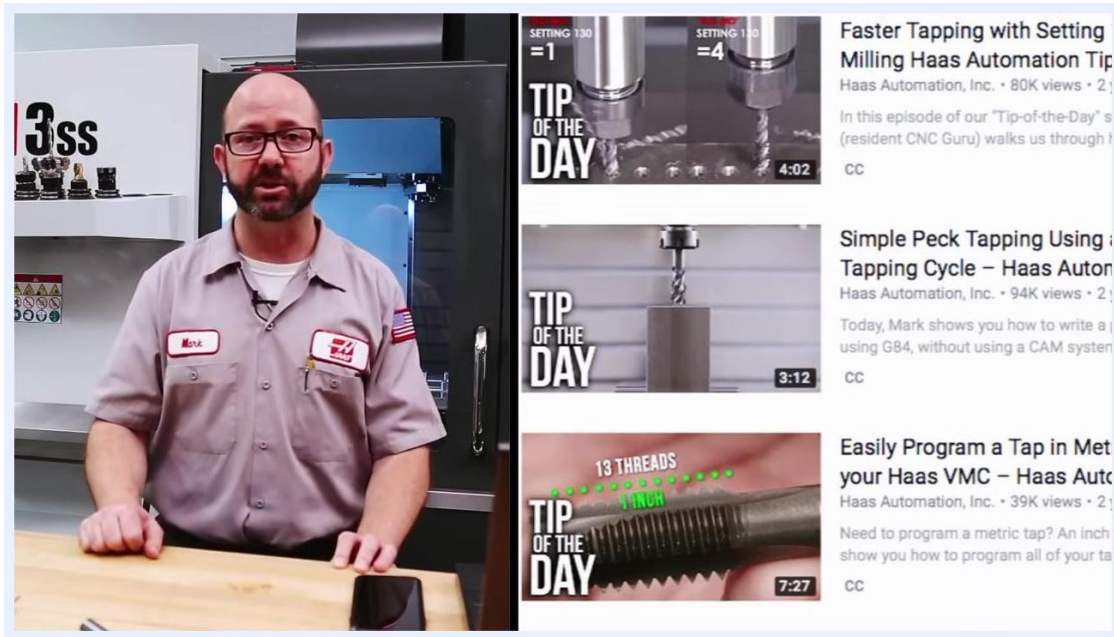
RPM = 1592

A man with a beard and glasses, wearing a light blue short-sleeved button-down shirt with a name tag and a USA patch, is smiling and standing with his hands at his sides. The background is a plain white wall.

Instructor celebrating with an explosion animation in the background, confirming the spindle speed calculation.

Required reference materials

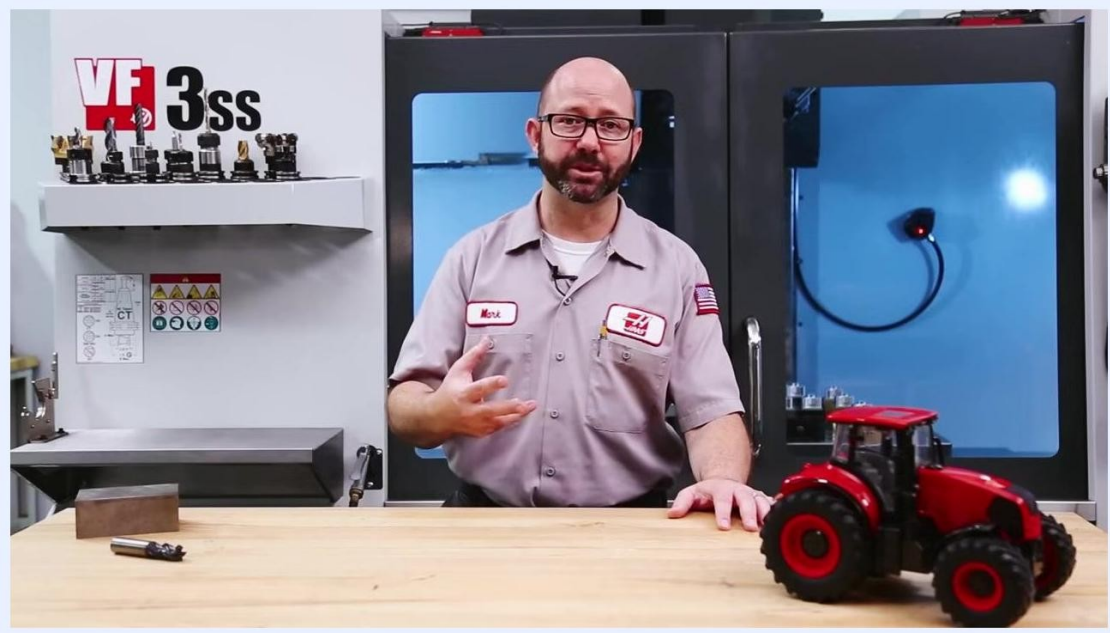
Before you begin, gather the following:



Tip of the Day branding with Haas logo, host's name, caliper, and markers on a whiteboard background

- The job setup sheet, since all calculations and planning begin here.
- The tool manufacturer's catalog or manual, ideally in digital (PDF) form.
- The exact tool being used, including its diameter and number of teeth/flutes, and whether it has a coating (coatings can double allowable cutting speeds).

- A calculator, or a digital speeds-and-feeds app, or access to your CNC control's built-in milling calculator.



Instructor gesturing while explaining the importance of knowing the tool and its coating in a machine shop setting.

Understanding surface speed

1. Review the overall path from setup to speed and feed values. All calculations begin with your setup sheet and lead first to determining spindle speed, then feed rate.

2. Consider the surface speed analogy. If a tractor moves at 1 kilometer per hour (about 17 meters per minute), both the small front tire and the large rear tire travel at the same surface speed where the rubber meets the road, even though the tires are different sizes.



Model tractor with large rear tires and small front tires on a wooden table, illustrating the surface speed analogy in a workshop setting.

3. Confirm the result of the analogy. Both tires travel at 17 surface meters per minute. This shows that surface speed is independent of the diameter of the rotating object, which applies directly to tool surface speed in machining.
4. Remember the key takeaway: surface speed (meters per minute) stays the same at the contact point, regardless of the diameter of the tire or tool.

Calculating circumference and identifying maximum cutting speed

5. Recognize that RPM differs even though surface speed is the same. The smaller tire (or tool) must rotate more times per minute than the larger one to cover the same ground. In the tractor example, the small front tire turns at 54 RPM while the large rear tire turns at 42 RPM to cover the same distance.



Animated scene with a tractor, grass, a chicken, and a cow holding a sign that reads "17 SURFACE METERS PER MINUTE."

6. Calculate circumference using $\text{diameter} \times \pi$. Pi (π) is the constant ratio between a circle's diameter and its circumference, approximately 3.14.
7. Work through the example calculations:
- For a 100 mm diameter tire: Circumference = $100 \text{ mm} \times 3.14 = 314 \text{ mm}$.
 - For a 1 meter diameter tire: Circumference = $1 \text{ m} \times 3.14 = 3.14 \text{ m}$.

8. Note the safety and tooling limit implied by this concept. Just as a tire can overheat and fail if spun too fast, a cutting tool has a maximum surface speed for each material. Exceeding it causes overheating and rapid tool wear. Always check the recommended maximum cutting speed for your specific tool and material combination.



Split-view close-up of a tractor's front and rear tires. The larger rear tire is labeled 42 RPM, and the smaller front tire is labeled 54 RPM.

9. Summarize the two rules to carry forward:
 - Use Circumference = Diameter $\times$ π (3.14) to find how much material is covered per revolution.

- Never exceed the maximum recommended cutting speed for your tool and material, to avoid premature tool failure.



Workbench view with a model tractor and a large tire, overlaid with the calculation $1 \text{ m} \times 3.14 = 3.14 \text{ m CIRCUMFERENCE}$.

Converting cutting speed to spindle speed (RPM)

10. Distinguish between the two key speeds in machining. Cutting speed (V_c) is the surface speed where the tool meets the part, measured in meters per minute (m/min). Spindle speed (n) is the number of revolutions per minute the spindle turns – the value your CNC control needs, entered as the S-code.



Machine shop workbench with a model tractor and on-screen text showing $\pi = 3.14$ circled in red, reinforcing the constant value used in the calculations.

11. Confirm the units for each variable:

- Cutting speed (V_c): meters per minute (m/min)
- Spindle speed (n): revolutions per minute (RPM)
- Tool diameter (D): millimeters (mm)
- Pi (π): 3.14

12. Locate where RPM and feed values are entered in your G-code. Spindle speed is set with the S-code (for example, S1500 for 1500 RPM), and feed rate is set with the F-code (for example, F1000 for 1000 mm/min).

Calculate both values before running the program.

13. Apply the spindle speed formula:

$$n = (V_c \times 1000) / (D \times \pi)$$

- n = spindle speed (RPM, S-code)
- V_c = cutting speed (m/min)
- D = tool diameter (mm)
- $\pi = 3.14$

This formula converts the surface speed from the tool catalog and the tool diameter into the RPM value your CNC machine needs.

14. Reference the standard symbols used by tool suppliers:

Symbol	Meaning
n	rev/min (RPM, S-Code)
V_c	Cutting Speed (m/min)
D	Tool Diameter (mm)
π	Pi (3.14, diameter-to-circumference ratio)

15. Pay particular attention to V_c and D , since these are the two values you will need to obtain from the catalog or measure directly before you can calculate spindle speed.

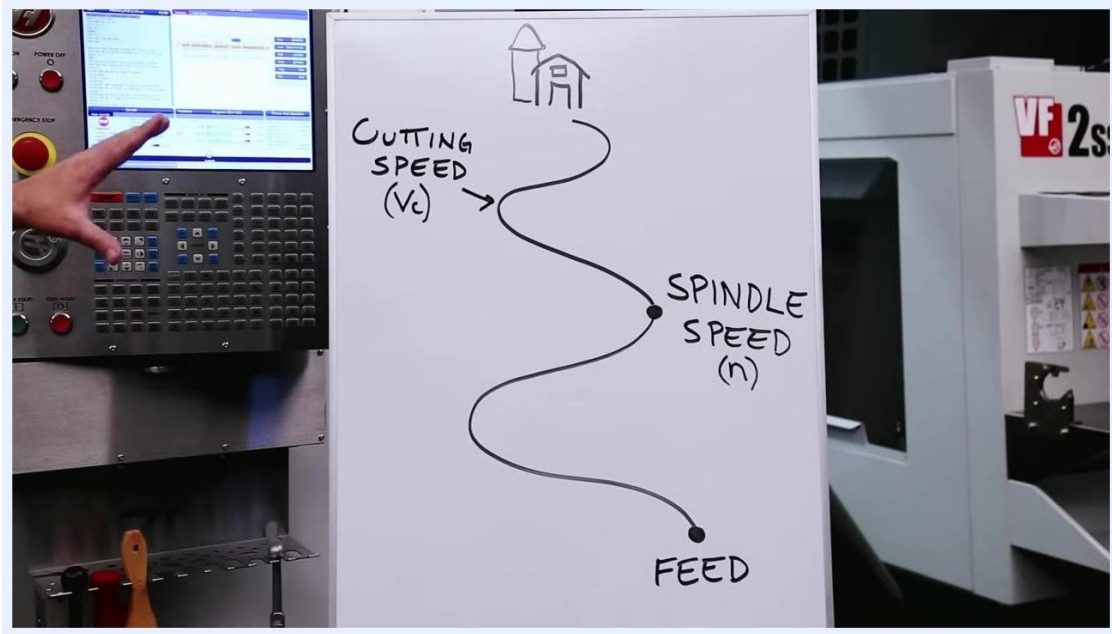
16. Simplify the formula for faster calculation. Dividing 1000 by π gives approximately 318, allowing the formula to be simplified as follows:

$$n = (V_c \times 1000) / (D \times \pi)$$

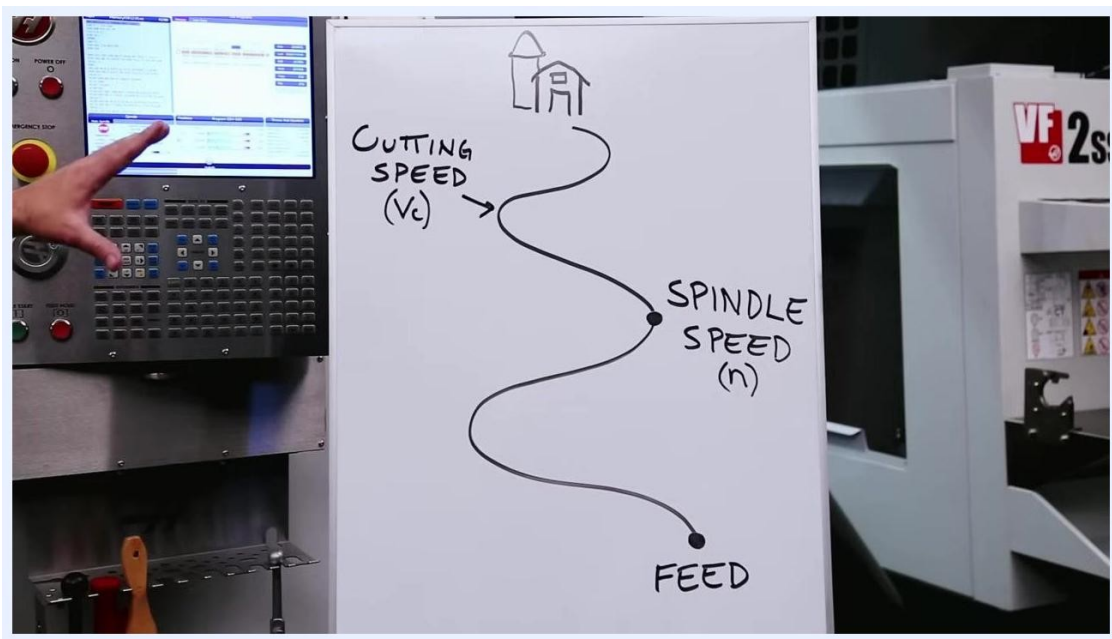
$$n = (V_c / D) \times (1000 / \pi)$$

$$n = (V_c / D) \times 318$$

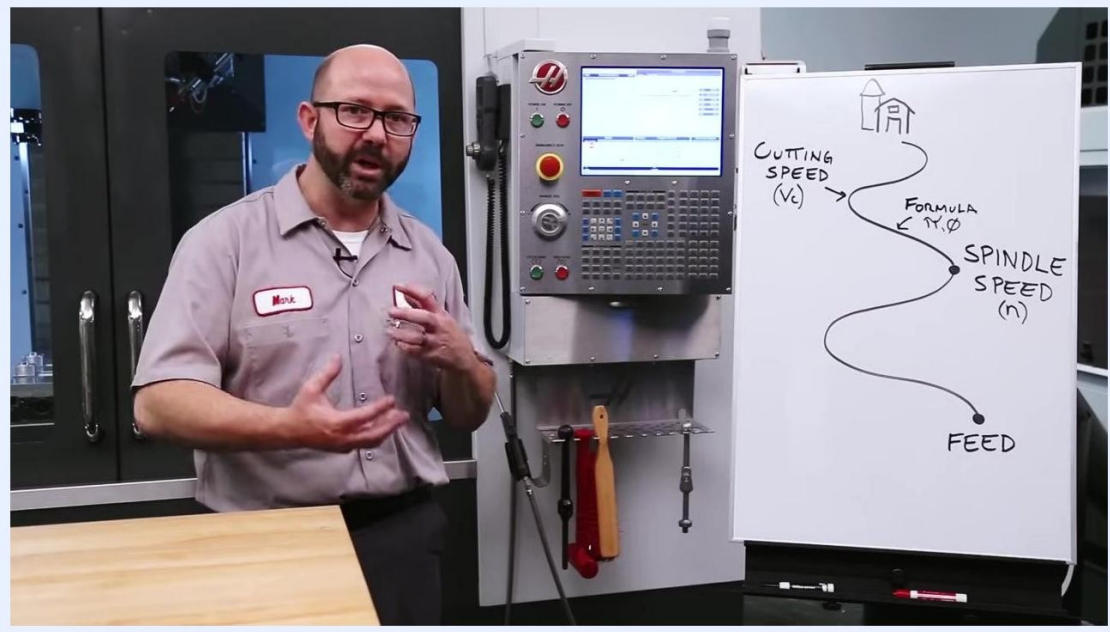
$$n = V_c \times 318 \div D$$



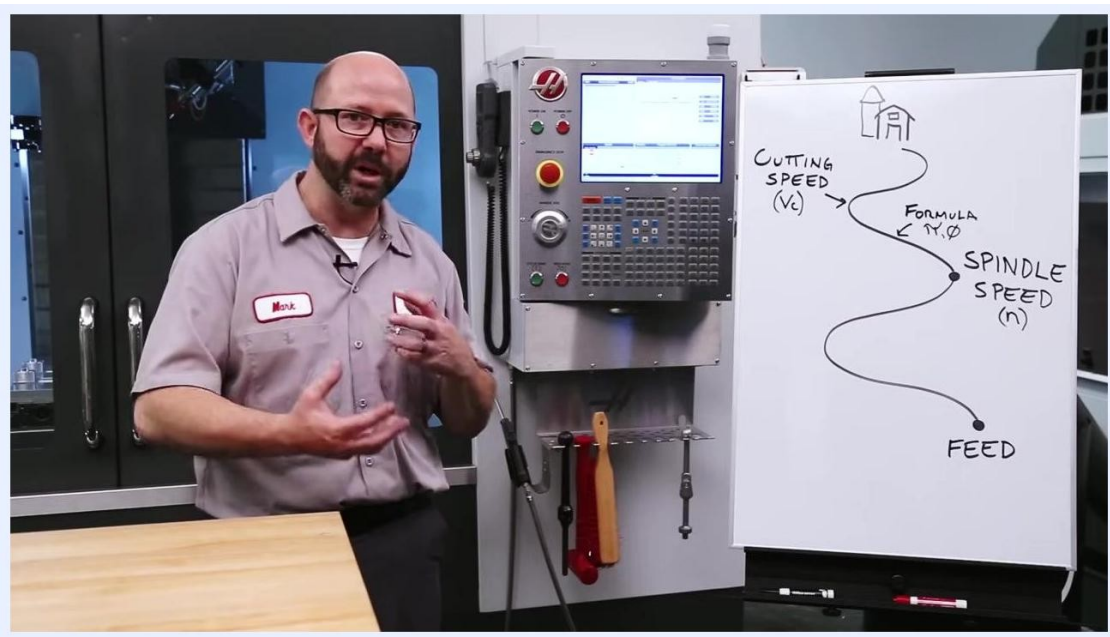
Whiteboard diagram showing the relationship between Cutting Speed (V_c), Spindle Speed (n), and Feed, with a CNC control panel and part of a VF 2ss machine visible.



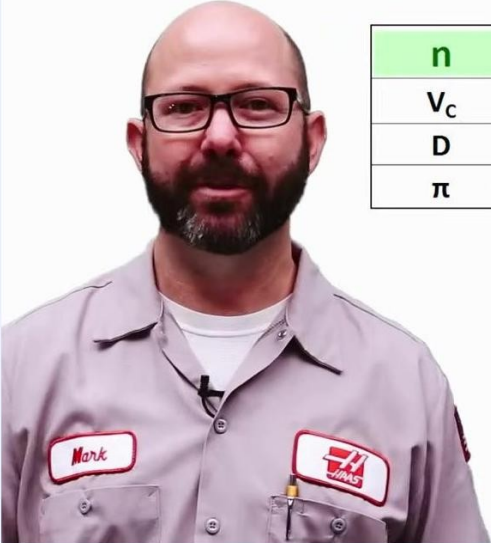
Close-up of a whiteboard showing "Cutting Speed (V_c) m/min" and "Spindle Speed (RPM) (n)" with a connecting path.



G-code snippet with the S-code and F-code locations circled in red, indicating where RPM and feed values should be entered.



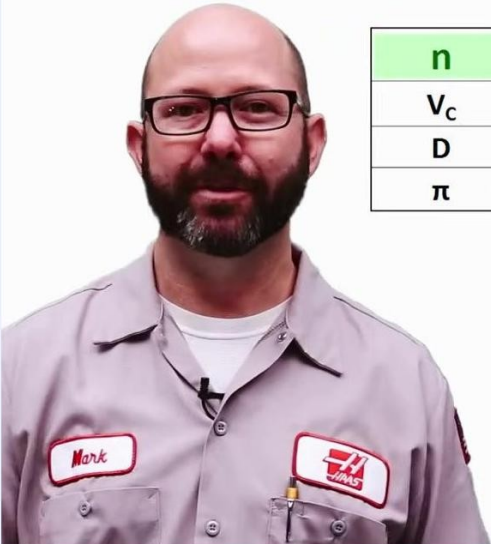
Whiteboard formula path showing the conversion from cutting speed to spindle speed, with the word "Formula" and the symbols π and $\varnothing$ marked at the conversion point.



n	rev/min (RPM, S-Code)
V_c	Cutting Speed (m/min)
D	Tool Diameter (mm)
π	Pi (3.14, our Ø to Circ. ratio)

$$n = \frac{V_c \times 1000}{D \times \pi}$$

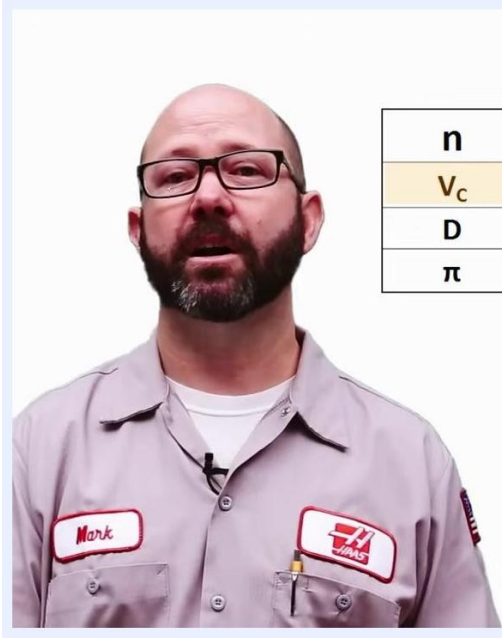
On-screen formula $n = (V_c \times 1000) / (D \times \pi)$ with a table defining each symbol.



n	rev/min (RPM, S-Code)
V_c	Cutting Speed (m/min)
D	Tool Diameter (mm)
π	Pi (3.14, our Ø to Circ. ratio)

$$n = \frac{V_c \times 1000}{D \times \pi}$$

Table of symbol definitions with the spindle speed formula displayed below; the row for n is highlighted in green.



n	rev/min (RPM, S-Code)
V_c	Cutting Speed (m/min)
D	Tool Diameter (mm)
π	Pi (3.14, our Ø to Circ. ratio)

$$n = \frac{V_c \times 1000}{D \times \pi}$$

Table defining n, V_c, D, and π next to the spindle speed formula $n = (V_c \times 1000) / (D \times \pi)$, with the V_c row highlighted.

Selecting the correct cutting speed from the tool catalog

17. Confirm the workpiece material before opening the catalog. In this example, the workpiece is labeled STEEL 4140, and the cutting tool is marked SD99-200.

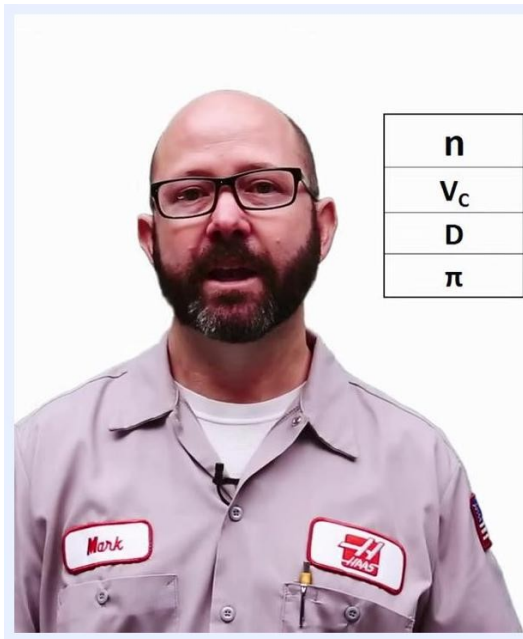


Caution: Modern cutting tools may have coatings that can double their allowable cutting speeds. Know the exact tool you are using so that you don't rely on outdated or overly conservative speed values.

18. Access the tool manufacturer's catalog, downloading the manual or catalog in PDF form when possible.
19. Use the color-coded material chart to quickly identify your material group:
 - P (Blue): Non-Alloy Steel and Cast Steel, Low-Alloy Steel and Cast Steel, High-Alloy Steel and Cast Steel, Stainless Steel and Cast Steel

- M (Yellow): Stainless Steel
- K (Red): Grey Cast Iron, Ductile Cast Iron, Nodular Cast Iron
- N (Green): Non-Ferrous Metals, Aluminum and Aluminum Alloys
- S (Brown): Superalloys, Titanium and Titanium Alloys

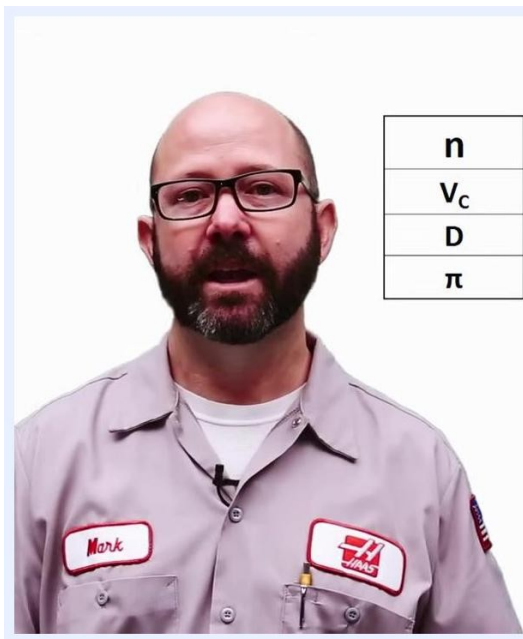
- H (Gray): Hard Cast Iron, Hardened Steel



n	rev/min (RPM, S-Code)
V_c	Cutting Speed (m/min)
D	Tool Diameter (mm)
π	Pi (3.14, our Ø to Circ. ratio)

$$n = \frac{V_c \times 1000}{D \times \pi}$$

Table of variables and the spindle speed formula displayed on screen in the workshop, showing the progression to the simplified version using 318.



n	rev/min (RPM, S-Code)
V_c	Cutting Speed (m/min)
D	Tool Diameter (mm)
π	Pi (3.14, our Ø to Circ. ratio)

$$n = \frac{V_c \times 1000}{D \times \pi}$$

Four versions of the spindle speed formula, showing the progression from the full formula to the simplified version using 318.

$$n = \frac{V_c \times 1000}{D \times \pi}$$
$$n = \frac{V_c}{D} \times \frac{1000}{\pi}$$
$$n = \frac{V_c}{D} \times 318$$
$$n = V_c \times 318 \div D$$

Close-up of a metal block labeled "STEEL 4140" and a cutting tool marked "SD99-200" on a wooden workbench.

P	Non-Alloy Steel and Cast Steel, Low-Alloy Steel and Cast Steel High-Alloy Steel and Cast Steel, Stainless Steel and Cast Steel
M	Stainless Steel
K	Grey Cast Iron Ductile Cast Iron Modular Cast Iron
N	Non-Ferrous Metals Aluminum and Aluminum Alloys
S	Superalloys Titanium and Titanium Alloys
H	Hard Cast Iron Hardened Steel

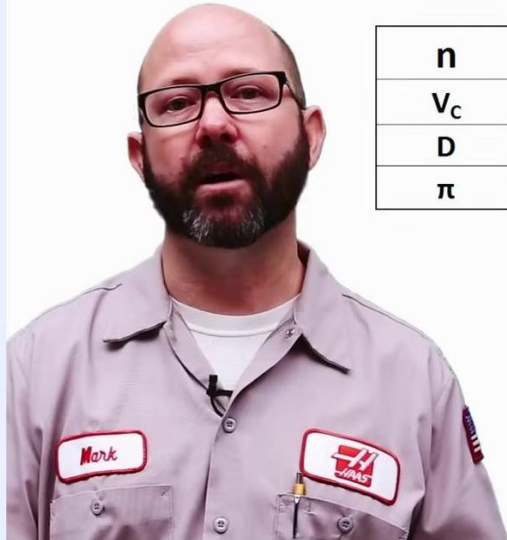
A laptop being placed on the workbench to access a digital tool catalog.

P	Non-Alloy Steel and Cast Steel, Low-Alloy Steel and Cast Steel High-Alloy Steel and Cast Steel, Stainless Steel and Cast Steel
M	Stainless Steel
K	Grey Cast Iron Ductile Cast Iron Modular Cast Iron
N	Non-Ferrous Metals Aluminum and Aluminum Alloys
S	Superalloys Titanium and Titanium Alloys
H	Hard Cast Iron Hardened Steel

Color-coded chart listing material groups (P, M, K, N, S, H) and their corresponding materials.

Matching your material to the ISO group and catalog code

20. Identify your material group on the chart. The ISO groups are: N (Green) Non-Ferrous Metals/Aluminum Alloys, S (Brown) Superalloys/Titanium Alloys, K (Red) Cast Irons, M (Yellow) Stainless Steel, P (Blue) Steels, and H (Gray) Hardened Materials.



n	rev/min (RPM, S-Code)
V_c	Cutting Speed (m/min)
D	Tool Diameter (mm)
π	Pi (3.14, our Ø to Circ. ratio)

$$n = \frac{V_c \times 1000}{D \times \pi}$$

Machine shop workbench with a CNC machine labeled VF 3ss, a metal block, a cutting tool, and a model tractor.

21. Confirm your specific workpiece material. In this example, the material is 4140 steel, a common alloy steel used in machining.
22. Be aware that material names vary by country. 4140 is a U.S. designation for this material; other regions may use a different name for the same alloy.
23. Match your material to the ISO group regardless of local naming. Worldwide, 4140 steel falls under the P (blue) ISO material group for steels. Reference the full chart for all groups and descriptions.
24. Look up your material in the manufacturer's catalog. For Kennametal, 4140 steel is listed as P4 (P4 Alloy Steels and Tool Steels). The catalog cross-references international standards (AISI, DIN, JIS, etc.), tensile strength, and hardness.
25. Note that the code following the ISO letter varies by manufacturer. For the same material, Sandvik uses P2.1, and Iscar/Widia use P6 or P7. Use the

material cross-reference list to match your material to the correct code in each catalog.



Caution: Choosing the wrong material group or code can lead to improper speeds and feeds, causing rapid tool wear or failure.

26. Never apply cutting speeds for one material group to another. For example, running a drill or end mill in titanium at speeds meant for mild steel will damage the tool.

27. Contact your tooling representative if you are unsure which material group or catalog code applies to your job.



Highlighted chart section showing "N" for Non-Ferrous Metals and Aluminum Alloys, with "S" for Superalloys below.



Machine shop workbench with a block of 4140 steel and cutting tool, the material used for this example.

P	Non-Alloy Steel and Cast Steel, Low-Alloy Steel and Cast Steel High-Alloy Steel and Cast Steel, Stainless Steel and Cast Steel
M	Stainless Steel
K	Grey Cast Iron Ductile Cast Iron Nodular Cast Iron
N	Non-Ferrous Metals Aluminum and Aluminum Alloys
S	Superalloys Titanium and Titanium Alloys
H	Hard Cast Iron Hardened Steel

A hand resting on the steel block while noting that material names such as 4140 steel can vary by country.

Workpiece Materials Overview Listing

Steel

KENNAMETAL

P4

Workpiece Materials Listing • Steel • P4 (continued)

P4 Alloy Steels and Tool Steels

Content: C>.25%

Tensile Strength RM (MPa)*: 850–1400

Hardness (HB): 350–450 (HRC): 35–48

AISI**	DIN	BTS	JIS	UNS	EN	AFNOR	UNI	SIS	SAE	ASTM	Material Number	Manufacturer Reference
–	C620	–	SWRH 62 B	–	C 62 D	FM 62	C 62 D	–	–	–	1.0611	–
–	C660	–	–	–	C 66 D	FM 66	C 66 D	–	–	–	1.0612	–
–	C680	–	SWRH 67 B	–	C 68 D	FM 68	C 68 D	–	–	–	1.0613	–
–	C700	–	–	–	C 70 D	FM 70	C 70 D	–	–	–	1.0615	–
–	C720	–	SWRH 72 B	–	C 72 D	FM 72	C 72 D	–	–	–	1.0617	–
–	GS-C 25	–	SCPH 1	–	GP 240 GH	GP 240 GH	GP 240 GH	–	–	–	1.0619	–
–	GP 240 GR	–	–	–	GP 240 GR	GP 240 GR	GP 240 GR	–	–	–	1.0621	–
–	C800	–	–	–	C 80 D	FM 80	C 80 D	–	–	–	1.0622	–
–	StSch 900 A (R 0900)	–	–	–	R 0900	–	–	–	–	–	1.0623	–
–	StSch 900 B (R 0900 Mn)	–	–	–	R 0900 Mn	–	–	–	–	–	1.0624	–
1345	StSch 90 C/GP 280 GH	–	–	G13450	GP 280 GH	GP 280 GH	GP 280 GH	–	–	–	1.0625	–
–	C820	–	SWRH 82 B	–	C 82 D	FM 82	C 82 D	–	–	–	1.0626	–
–	C880	–	–	–	C 88 D	–	C 88 D	–	–	–	1.0628	–
–	StSch 1200 (R 1200)	–	–	–	R 1200	–	–	–	–	–	1.0631	–
–	C 61/60 Mn 3	–	–	–	C 61	–	C 61	–	–	–	1.0642	–
–	C 85/85 Mn 3	–	–	–	C 85	–	C 85	–	–	–	1.0647	–
–	35 SPb 20	35 SPb 20	–	–	35 SPb 20	35 SPb 20	–	–	–	–	1.0756	–
–	45 SPb 20	–	–	–	46 SPb 20	–	–	–	–	–	1.0757	–
–	38 SMnPb 28	38 SMnPb 28	–	–	38 SMnPb 28	38 SMnPb 28	–	–	–	–	1.0761	–
–	44 SMn 28	44 SMn 28	–	–	44 SMn 28	44 SMn 28	–	–	–	–	1.0762	–
–	44 SMnPb 28	44 SMnPb 28	–	–	44 SMnPb 28	44 SMnPb 28	–	–	–	–	1.0763	–
–	36 SMnPb 14	36 SMnPb 14	–	–	36 SMnPb 14	36 SMnPb 14	–	–	–	–	1.0765	–
9255	51 Si 7	250 A 53	–	G92550	55 NiCrMoV 7	51 Si 7	48 Si 7	2090	–	–	1.0903	–
9255	55 Si 7	250 A 53	SKH 1; SKT 4	G92550	R 1100 Cr	55 Si 7	55 Si 8	2085	–	–	1.0904	–
–	StSch 1100 (R 1100 Cr)	–	–	–	R 1100 Cr	–	–	–	–	–	1.0915	–
9262	60 SiCr 7	250 A 61	SUP 7	G92620	60 SiCr 8	60 SiCr 7	60 SiCr 8	–	–	–	1.0961	–
–	1.05 F 600 TM (S 700 MPa)	–	–	–	S 700 MPa	S 600 MPa	S 700 MPa	–	–	–	1.0966	–

Full color-coded chart listing all ISO material groups and their corresponding materials.

Workpiece Materials Overview Listing

Steel



P4

Workpiece Materials Listing • Steel • P4 (continued)

P4 Alloy Steels and Tool Steels

Content: C>.25%

Tensile Strength RM (MPa)*: 850–1400

Hardness (HB): 350–450 (HRC): 35–48

AISI**	DIN	BTS	JIS	UNS	EN	AFNOR	UNI	SIS	SAE	ASTM	Material Number	Manufacturer Reference
–	C620	–	SWRH 62 B	–	C 62 D	FM 62	C 62 D	–	–	–	1.0611	–
–	C660	–	–	–	C 66 D	FM 66	C 66 D	–	–	–	1.0612	–
–	C680	–	SWRH 67 B	–	C 68 D	FM 68	C 68 D	–	–	–	1.0613	–
–	C700	–	–	–	C 70 D	FM 70	C 70 D	–	–	–	1.0615	–
–	C720	–	SWRH 72 B	–	C 72 D	FM 72	C 72 D	–	–	–	1.0617	–
–	GS-C 25	–	SCPH 1	–	GP 240 GH	GP 240 GH	GP 240 GH	–	–	–	1.0619	–
–	GP 240 GR	–	–	–	GP 240 GR	GP 240 GR	GP 240 GR	–	–	–	1.0621	–
–	C800	–	–	–	C 80 D	FM 80	C 80 D	–	–	–	1.0622	–
–	StSch 900 A (R 0900)	–	–	–	R 0900	–	–	–	–	–	1.0623	–
–	StSch 900 B (R 0900 Mn)	–	–	–	R 0900 Mn	–	–	–	–	–	1.0624	–
1345	StSch 90 C/GP 280 GH	–	–	G13450	GP 280 GH	GP 280 GH	GP 280 GH	–	–	–	1.0625	–
–	C820	–	SWRH 82 B	–	C 82 D	FM 82	C 82 D	–	–	–	1.0626	–
–	C880	–	–	–	C 88 D	–	C 88 D	–	–	–	1.0628	–
–	StSch 1200 (R 1200)	–	–	–	R 1200	–	–	–	–	–	1.0631	–
–	C 61/60 Mn 3	–	–	–	C 61	–	C 61	–	–	–	1.0642	–
–	C 85/85 Mn 3	–	–	–	C 85	–	C 85	–	–	–	1.0647	–
–	35 SPb 20	35 SPb 20	–	–	35 SPb 20	35 SPb 20	–	–	–	–	1.0756	–
–	45 SPb 20	–	–	–	46 SPb 20	–	–	–	–	–	1.0757	–
–	38 SMnPb 28	38 SMnPb 28	–	–	38 SMnPb 28	38 SMnPb 28	–	–	–	–	1.0761	–
–	44 SMn 28	44 SMn 28	–	–	44 SMn 28	44 SMn 28	–	–	–	–	1.0762	–
–	44 SMnPb 28	44 SMnPb 28	–	–	44 SMnPb 28	44 SMnPb 28	–	–	–	–	1.0763	–
–	36 SMnPb 14	36 SMnPb 14	–	–	36 SMnPb 14	36 SMnPb 14	–	–	–	–	1.0765	–
9255	51 Si 7	250 A 53	–	G92550	51 Si 7	48 Si 7	2090	–	–	–	1.0903	–
9255	55 Si 7	250 A 53	SKH 1; SKT 4	G92550	55 NiCrMoV 7	55 Si 7	55 Si 8	2085	–	–	1.0904	–
–	StSch 1100 (R 1100 Cr)	–	–	–	R 1100 Cr	–	–	–	–	–	1.0915	–
9262	60 SiCr 7	250 A 61	SUP 7	G92620	60 SiCr 8	60 SiCr 7	60 SiCr 8	–	–	–	1.0961	–
–	(not for RM TM (S 7000 M3))	–	–	–	S 7000 M3	S 7000 M3	S 7000 M3	–	–	–	1.0966	–

Kennametal catalog page showing P4 workpiece materials listing, including 4140 steel and cross-references.

Workpiece Materials Overview Listing

Steel



P4

Workpiece Materials Listing • Steel • P4 (continued)

P4 Alloy Steels and Tool Steels

Content: C>.25%

Tensile Strength RM (MPa)*: 850–1400

Hardness (HB): 350–450 (HRC): 35–48

AISI**	DIN	BTS	JIS	UNS	EN	AFNOR	UNI	SIS	SAE	ASTM	Material Number	Manufacturer Reference
–	C620	–	SWRH 62 B	–	C 62 D	FM 62	C 62 D	–	–	–	1.0611	–
–	C660	–	–	–	C 66 D	FM 66	C 66 D	–	–	–	1.0612	–
–	C680	–	SWRH 67 B	–	C 68 D	FM 68	C 68 D	–	–	–	1.0613	–
–	C700	–	–	–	C 70 D	FM 70	C 70 D	–	–	–	1.0615	–
–	C720	–	SWRH 72 B	–	C 72 D	FM 72	C 72 D	–	–	–	1.0617	–
–	GS-C 25	–	SCPH 1	–	GP 240 GH	GP 240 GH	GP 240 GH	–	–	–	1.0619	–
–	GP 240 GR	–	–	–	GP 240 GR	GP 240 GR	GP 240 GR	–	–	–	1.0621	–
–	C800	–	–	–	C 80 D	FM 80	C 80 D	–	–	–	1.0622	–
–	StSch 900 A (R 0900)	–	–	–	R 0900	–	–	–	–	–	1.0623	–
–	StSch 900 B (R 0900 Mn)	–	–	–	R 0900 Mn	–	–	–	–	–	1.0624	–
1345	StSch 90 C/GP 280 GH	–	–	G13450	GP 280 GH	GP 280 GH	GP 280 GH	–	–	–	1.0625	–
–	C820	–	SWRH 82 B	–	C 82 D	FM 82	C 82 D	–	–	–	1.0626	–
–	C880	–	–	–	C 88 D	–	C 88 D	–	–	–	1.0628	–
–	StSch 1200 (R 1200)	–	–	–	R 1200	–	–	–	–	–	1.0631	–
–	C 61/60 Mn 3	–	–	–	C 61	–	C 61	–	–	–	1.0642	–
–	C 85/85 Mn 3	–	–	–	C 85	–	C 85	–	–	–	1.0647	–
–	35 SPb 20	35 SPb 20	–	–	35 SPb 20	35 SPb 20	–	–	–	–	1.0756	–
–	45 SPb 20	–	–	–	46 SPb 20	–	–	–	–	–	1.0757	–
–	38 SMnPb 28	38 SMnPb 28	–	–	38 SMnPb 28	38 SMnPb 28	–	–	–	–	1.0761	–
–	44 SMn 28	44 SMn 28	–	–	44 SMn 28	44 SMn 28	–	–	–	–	1.0762	–
–	44 SMnPb 28	44 SMnPb 28	–	–	44 SMnPb 28	44 SMnPb 28	–	–	–	–	1.0763	–
–	36 SMnPb 14	36 SMnPb 14	–	–	36 SMnPb 14	36 SMnPb 14	–	–	–	–	1.0765	–
9255	51 Si 7	250 A 53	–	G92550	51 Si 7	48 Si 7	2090	–	–	–	1.0903	–
9255	55 Si 7	250 A 53	SKH 1; SKT 4	G92550	55 NiCrMoV 7	55 Si 7	55 Si 8	2085	–	–	1.0904	–
–	StSch 1100 (R 1100 Cr)	–	–	–	R 1100 Cr	–	–	–	–	–	1.0915	–
9262	60 SiCr 7	250 A 61	SUP 7	G92620	60 SiCr 8	60 SiCr 7	60 SiCr 8	–	–	–	1.0961	–
–	(not for RM TM (S 7000 M3))	–	–	–	S 7000 M3	S 7000 M3	S 7000 M3	–	–	–	1.0966	–

Material cross-reference list showing ISO group P and various manufacturer codes for steels.

Workpiece Materials Overview Listing

Steel



P4

Workpiece Materials Listing • Steel • P4 (continued)

P4 Alloy Steels and Tool Steels

Content: C>.25%

Tensile Strength RM (MPa)*: 850–1400

Hardness (HB): 350–450 (HRC): 35–48

AISI**	DIN	BTS	JIS	UNS	EN	AFNOR	UNI	SIS	SAE	ASTM	Material Number	Manufacturer Reference
–	C620	–	SWRH 62 B	–	C 62 D	FM 62	C 62 D	–	–	–	1.0611	–
–	C660	–	–	–	C 66 D	FM 66	C 66 D	–	–	–	1.0612	–
–	C680	–	SWRH 67 B	–	C 68 D	FM 68	C 68 D	–	–	–	1.0613	–
–	C700	–	–	–	C 70 D	FM 70	C 70 D	–	–	–	1.0615	–
–	C720	–	SWRH 72 B	–	C 72 D	FM 72	C 72 D	–	–	–	1.0617	–
–	GS-C-25	–	SCPH 1	–	GP 240 GH	GP 240 GH	GP 240 GH	–	–	–	1.0619	–
–	GP 240 GR	–	–	–	GP 240 GR	GP 240 GR	GP 240 GR	–	–	–	1.0621	–
–	C800	–	–	–	C 80 D	FM 80	C 80 D	–	–	–	1.0622	–
–	StSch 900 A (R 0900)	–	–	–	R 0900	–	–	–	–	–	1.0623	–
–	StSch 900 B (R 0900 Mn)	–	–	–	R 0900 Mn	–	–	–	–	–	1.0624	–
1345	StSch 90 C/GP 280 GH	–	–	G13450	GP 280 GH	GP 280 GH	GP 280 GH	–	–	–	1.0625	–
–	C820	–	SWRH 82 B	–	C 82 D	FM 82	C 82 D	–	–	–	1.0626	–
–	C880	–	–	–	C 88 D	–	C 88 D	–	–	–	1.0628	–
–	StSch 1200 (R 1200)	–	–	–	R 1200	–	–	–	–	–	1.0631	–
–	C 61/60 Mn 3	–	–	–	C 61	–	C 61	–	–	–	1.0642	–
–	C 85/85 Mn 3	–	–	–	C 85	–	C 85	–	–	–	1.0647	–
–	35 SPb 20	35 SPb 20	–	–	35 SPb 20	35 SPb 20	–	–	–	–	1.0756	–
–	45 SPb 20	–	–	–	45 SPb 20	–	–	–	–	–	1.0757	–
–	38 SMnPb 28	38 SMnPb 28	–	–	38 SMnPb 28	38 SMnPb 28	–	–	–	–	1.0761	–
–	44 SMn 28	44 SMn 28	–	–	44 SMn 28	44 SMn 28	–	–	–	–	1.0762	–
–	44 SMnPb 28	44 SMnPb 28	–	–	44 SMnPb 28	44 SMnPb 28	–	–	–	–	1.0763	–
–	36 SMnPb 14	36 SMnPb 14	–	–	36 SMnPb 14	36 SMnPb 14	–	–	–	–	1.0765	–
9255	51 Si 7	250 A 53	–	G92550	–	51 Si 7	48 Si 7	2090	–	–	1.0903	–
9255	55 Si 7	250 A 53	SKH 1; SKT 4	G92550	55 NiCrMoV 7	55 Si 7	55 Si 8	2085	–	–	1.0904	–
–	StSch 1100 (R 1100 Cr)	–	–	–	R 1100 Cr	–	–	–	–	–	1.0915	–
9262	60 SiCr 7	250 A 61	SUP 7	G92620	60 SiCr 8	60 SiCr 7	60 SiCr 8	–	–	–	1.0961	–
–	(R 1100 Mn)	–	–	–	S 700 Mn	S 700 Mn	S 700 Mn	–	–	–	1.0966	–

Workbench view emphasizing that choosing the wrong material group can lead to premature tool failure.

Material cross reference list

ISO	MC	CMC	Country					
			Europe	Germany	Great Britain		Sweden	USA
			Standard					
			DIN EN	W.-nr.	BS	EN	SS	AISI/SAE
P	P2.3.Z.AN/H1.2.Z.HA	02.1/02.2	105WCr6	1.2419	-	-	2140	-
	P2.3.Z.AN/H1.2.Z.HA	-	-	-	-	-	-	-
	P2.3.Z.AN/H1.2.Z.HA	02.1/02.2	-	1.2714	-	-	-	L6
	P2.3.Z.AN/H1.3.Z.HA	02.1/02.2	100Cr6	1.2067	BL3	-	-	L3
	P2.4.Z.AN	02.1	16MnCr5	1.7139	-	-	2127	-
	P2.5.Z.HT	02.1	16Mo5	1.5423	1503-245-420	-	-	4520
	P2.5.Z.HT	02.1	40NiCrMo8-4	1.6562	311-Type 7	-	-	8740
	P2.5.Z.HT	02.1	42Cr4	1.7045	-	-	2245	5140
	P2.5.Z.HT	02.1	31NiCrMo14	1.5755	830 M 31	-	2534	-
	P2.5.Z.HT	02.2	36NiCr6	1.5710	640A35	111A	-	3135
	P2.6.C.UT	02.1	22Mo4	1.5419	605A32	-	2108	8620
	P2.6.C.UT	02.1/02.2	25CrMo4	1.7218	1717CDS110	-	2225	4130

On-screen comparison of "P: Low-alloy Steel" and "S: Titanium Ti6AlV4" material groups.



Workbench view accompanying the recommendation to contact your tooling representative if you are unsure of the correct material group.

Calculating spindle speed (RPM) for your job

28. Locate the recommended cutting speed for your material group and code in the cutting speed chart. Values are listed in meters per minute (Vc). Example values from an ISO 513 chart:

- P1: 150 m/min
- P2: 130 m/min
- P3: 115 m/min
- P4: 105 m/min
- P5: 100 m/min

- P6: 50 m/min



Workbench with a block of steel, cutting tool, laptop, and model tractor, set up in front of the CNC machine.

29. Select the appropriate cutting speed for your setup, taking into account tool length, setup rigidity, and how aggressively you plan to cut. A more rigid setup allows for higher cutting speeds; a less rigid setup calls for a lower cutting speed to avoid tool chatter or breakage.
30. Record your chosen cutting speed. For this example, 100 m/min is selected for the P5 group, 4140 steel job.
31. Apply the simplified spindle speed formula:

$$n = V_c \times 318 \div D$$

- n = spindle speed (RPM)
- V_c = cutting speed (m/min)
- D = tool diameter (mm)

32. Substitute your values into the formula. For this example, using a 20 mm end mill:

- $V_c = 100 \text{ m/min}$
- $D = 20 \text{ mm}$
- $n = 100 \times 318 \div 20$

33. Calculate the result. $n = 100 \times 318 \div 20 = 1592 \text{ RPM}$.

34. Confirm the outcome: you now know how fast to spin your tool — your spindle speed is 1592 RPM.



Cutting speed chart showing ISO 513 material groups (P, M) and recommended cutting speeds in m/min; P5 is 100 m/min.



Workbench setup illustrating how tool length, setup rigidity, and cutting aggressiveness affect the choice of cutting speed.

V_c
100 m/min

D
20

$n = V_c \times 318 \div D$

$n = 100 \times 318 \div 20 =$

RPM = 1592



On-screen spindle speed formula: $n = V_c \times 318 \div D$.

V_c
100 m/min

D
20

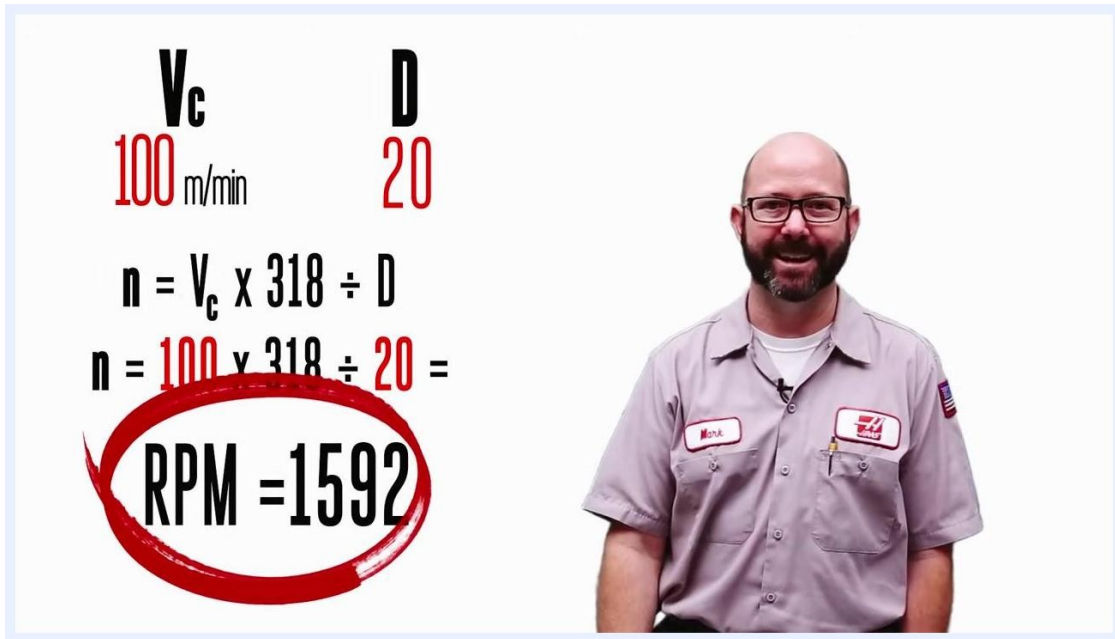
$n = V_c \times 318 \div D$

$n = 100 \times 318 \div 20 =$

RPM = 1592



Cutting speed and tool diameter values ($V_c = 100$ m/min, $D = 20$ mm), formula, and calculated RPM = 1592 circled in red.



Block of steel and 20 mm end mill shown together, representing the workpiece and tool used in the calculation.



Workbench with the block of steel and end mill, ready for the next calculation step now that spindle speed is known.

Calculating feed rate for end mills

35. Apply the feed rate formula for end mills:

$$V_f = f_z \times Z_n \times n$$

- V_f = Table Feed (mm/min, F-Code)
- f_z = Feed per Tooth (mm)
- Z_n = Number of Teeth
- n = Revolutions per minute (RPM, S-Code)

36. Read the formula in plain terms: table feed (mm/min) equals feed per tooth times the number of teeth times RPM.
37. Confirm what "n" represents in the formula – revolutions per minute (RPM, S-Code), the spindle speed you already calculated.
38. Count the number of teeth on your tool. Different end mills have different tooth counts – for example, one tool may have six teeth and another four. This number (Z_n) is required for the formula.
39. Understand why feed per tooth matters: the goal is for each tooth to take a specific, controlled "bite" of material with every pass.
40. Reference the manufacturer's cutting feed chart for recommended starting values of feed per tooth (f_z) and number of teeth (Z_n) by tool diameter. Example values:
- SD99-030: $D = 3.0$ mm, $Z_n = 4$, Roughing $f_z = 0.020$ mm, Finishing $f_z = 0.013$ mm

- SD99-120: D = 12.0 mm, Zn = 4, Roughing fz = 0.085 mm, Finishing fz = 0.055 mm



$$V_F = f_z \times Z_n \times n$$

V_F	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
Z	Number of Teeth
n	rev/min (RPM, S-Code)

$$\text{mm/min} = \text{Feed per Tooth} \times \text{Number of Teeth} \times \text{RPM}$$

Feed rate formula and variable definitions displayed on screen in the workshop.

41. Substitute your values into the formula. Using $f_z = 0.08$ mm, $Z_n = 4$, and $n = 1592$ RPM:

$$V_F = 0.08 \text{ mm} \times 4 \times 1592$$

42. Calculate the final feed rate. $V_F = 0.08 \times 4 \times 1592 = 509.44$, giving a feed rate of F509.44. This represents how far the centerline of the tool moves along its programmed path, in mm per minute.

43. Note that manufacturer recommendations vary. Some manuals provide a single feed rate for all cutting conditions, but most provide separate values depending on the operation – always consult the specific manufacturer data for your tool.

$$V_f = f_z \times Z \times n$$

V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
Z	Number of Teeth
n	rev/min (RPM, S-Code)

$$\text{mm/min} = \text{Feed per Tooth} \times \text{Number of Teeth} \times \text{RPM}$$

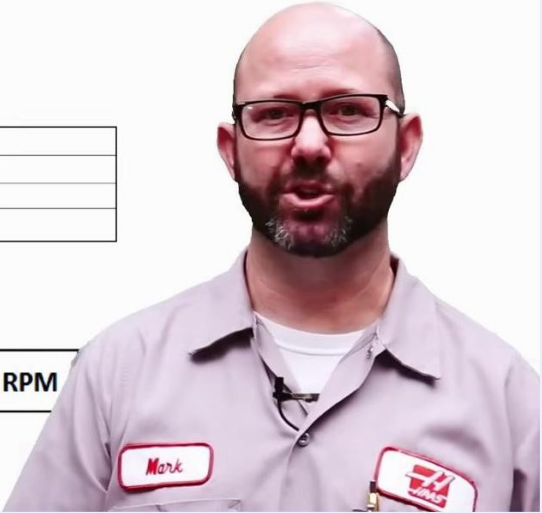


Feed rate formula with the feed-per-tooth variable and its table row both highlighted.

$$V_f = f_z \times Z \times n$$

V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
Z	Number of Teeth
n	rev/min (RPM, S-Code)

$$\text{mm/min} = \text{Feed per Tooth} \times \text{Number of Teeth} \times \text{RPM}$$



Feed rate formula with the n (RPM) variable highlighted.

$$V_f = f_z \times Z_n \times n$$

V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
Z	Number of Teeth
n	rev/min (RPM, S-Code)

$mm/min = \text{Feed per Tooth} \times \text{Number of Teeth} \times \text{RPM}$



Close-up of end mills and cutting tools on the workbench, showing tools with different numbers of teeth.

$$V_f = f_z \times Z_n \times n$$

V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
Z	Number of Teeth
n	rev/min (RPM, S-Code)

$mm/min = \text{Feed per Tooth} \times \text{Number of Teeth} \times \text{RPM}$



Workbench view with the CNC machine labeled VF 3ss, tool holders, and a laptop in the background.

$$V_f = f_z \times Z_n \times n$$

$$\text{mm/min} = \text{FEED PER TOOTH} \times \text{NUMBER OF TEETH} \times \text{RPM}$$

$$V_f = .08 \text{ mm} \times 4 \times 1592 =$$

Manufacturer's chart showing tool part numbers, diameters, number of teeth, and recommended feed per tooth values for roughing and finishing.

$$V_f = f_z \times Z_n \times n$$

$$\text{mm/min} = \text{FEED PER TOOTH} \times \text{NUMBER OF TEETH} \times \text{RPM}$$

$$V_f = .08 \text{ mm} \times 4 \times 1592 =$$

Feed rate formula with example values: $V_f = 0.08 \text{ mm} \times 4 \times 1592$.

$$V_f = f_z \times Z_n \times n$$

$$\text{mm/min} = \text{FEED PER TOOTH} \times \text{NUMBER OF TEETH} \times \text{RPM}$$

$$V_f = .08 \text{ mm} \times 4 \times 1592 = \text{F509.44}$$

Feed rate formula with calculated result: $V_f = 0.08 \text{ mm} \times 4 \times 1592 = \text{F509.44}$, result circled in red.

$$V_f = f_z \times Z_n \times n$$

$$\text{mm/min} = \text{FEED PER TOOTH} \times \text{NUMBER OF TEETH} \times \text{RPM}$$

$$V_f = .08 \text{ mm} \times 4 \times 1592 =$$

CNC machine and control panel, a reminder to always consult manufacturer data for specific cutting conditions.

Adjusting feed rate for slotting, side milling, and feed per revolution

44. Recognize that manufacturers typically provide at least two feed rate recommendations for end mills: one for slotting operations and one for side milling operations.
45. Reference the manufacturer's application data chart for specific values. A typical chart (for example, Kennametal's "General Purpose Solid Carbide End Mills – GOMill GP") provides:
 - Tool type and specifications (e.g., 4 flute, short, regular)
 - Material group codes
 - Cutting speed ranges (SFM)
 - Feed per tooth (IPT) by tool diameter
 - Separate columns for side milling (A) and slotting (B)
 - A note that, for slotting (B), IPT should be reduced by 20% compared to side milling (A)

General Purpose Solid Carbide End Mills • GOMill™ GP

Application Data

GOMill GP • 4SE/CH..IS-IR • 4 Flute • Short • Regular

																							
		Side Milling (A) and Slotting (B)		KC633M		Recommended feed per tooth (IPT = inch/th) for side milling (A). For slotting (B), reduce IPT by 20%.																	
		A		B		Cutting Speed – vc SFM		D1 – Diameter															
Material Group		ap	ae	ap		min	max		1/64	1/32	1/16	5/64	3/32	1/8	3/16	1/4	5/16	3/8	1/2	5/8	3/4	1	
P	0	Ap1 max	0.1 x D	0.5 x D	490	–	660	inch	.0156	.0313	.0625	.0781	.0938	.1250	.1875	.2500	.3125	.3750	.5000	.6250	.7500	1.0000	
	1	Ap1 max	0.1 x D	0.5 x D	490	–	660	IPT	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049	
	2	Ap1 max	0.1 x D	0.5 x D	460	–	620	IPT	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049	
	3	Ap1 max	0.1 x D	0.5 x D	390	–	520	IPT	.0001	.0002	.0004	.0004	.0005	.0007	.0011	.0015	.0020	.0023	.0029	.0034	.0039	.0045	
	4	Ap1 max	0.1 x D	0.5 x D	300	–	490	IPT	.0001	.0002	.0003	.0004	.0005	.0007	.0010	.0014	.0018	.0020	.0026	.0030	.0034	.0039	
M	1	Ap1 max	0.1 x D	0.5 x D	300	–	380	IPT	.0001	.0002	.0004	.0004	.0005	.0007	.0011	.0015	.0020	.0023	.0029	.0034	.0039	.0045	
	2	Ap1 max	0.1 x D	0.5 x D	200	–	260	IPT	.0001	.0002	.0003	.0004	.0004	.0006	.0009	.0012	.0016	.0018	.0023	.0027	.0031	.0036	

CNC shop with a machine, control panel, workbench, and laptop, set up to compare slotting and side milling feed recommendations.

46. Reduce feed per tooth when slotting. When slotting, the feed per tooth ("bite") should typically be about 25% less than when side milling.



Caution: Always follow the manufacturer's speed and feed recommendations rather than estimating, to achieve optimal results and tool life.

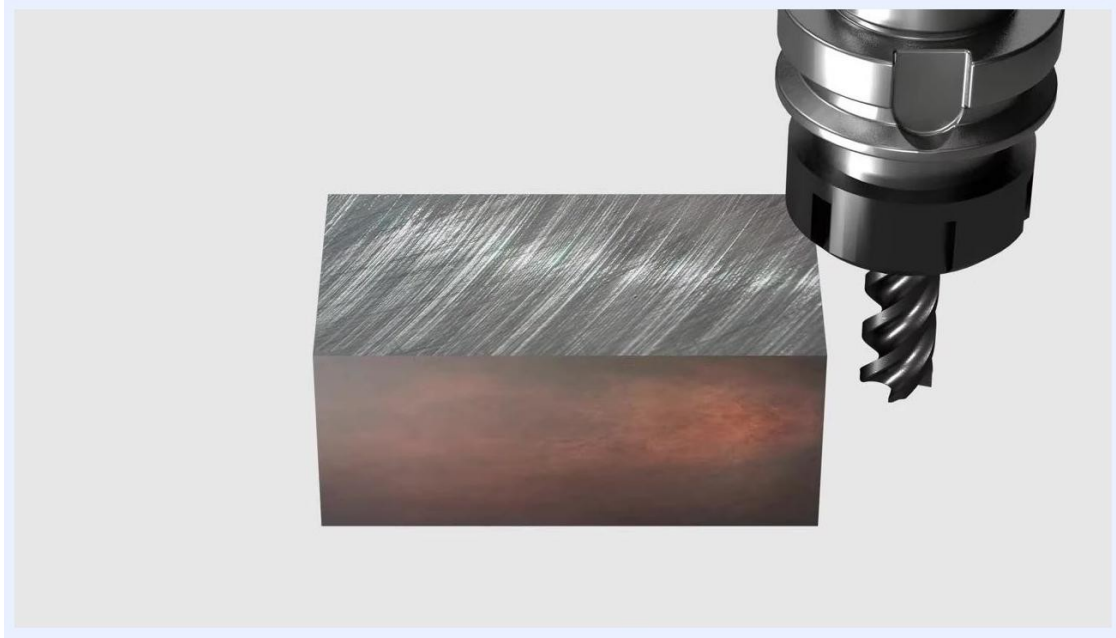
47. Recognize when a chart provides feed per revolution instead of feed per tooth. This is common for tools like drills.
48. Apply the feed-per-revolution formula:

$$V_f = f_n \times n$$

- V_f = Table Feed (mm/min)
- f_n = Feed per Revolution (mm)
- n = RPM

49. Review the complete legend of variables used across all speed and feed calculations:

Symbol	Meaning
V_c	Cutting Speed (m/min)
π	Pi (3.14, diameter-to-circumference ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, S-Code)
V_F	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut



Rendered image showing an end mill approaching a metal block from above, emphasizing the cutting interface where overheating can occur.

50. Keep the three key formulas together for quick reference:

- Spindle Speed: $n = V_c \times 318 \div D$
- Table Feed (End Mill): $V_f = f_z \times Z \times n$

- Table Feed (Drills): $V_f = f_n \times n$

General Purpose Solid Carbide End Mills • GOMill™ GP

Application Data

■ GOMill GP • 4SE/CH..IS-IR • 4 Flute • Short • Regular

																											
		Side Milling (A) and Slotting (B)		KC633M																							
		A		B		Cutting Speed – vc SFM																					

Manufacturer's application data chart for solid carbide end mills, showing feed per tooth, cutting speed, and diameter for side milling and slotting.

General Purpose Solid Carbide End Mills • GOMill™ GP

Application Data

GOMill GP • 4SE/CH..IS-IR • 4 Flute • Short • Regular

																											
		Side Milling (A) and Slotting (B)		KC633M																							
		A		B		Cutting Speed – vc SFM																					
Material Group						min	max	inch			1/64	1/32	1/16	5/64	3/32	1/8	3/16	1/4	5/16	3/8	1/2	5/8	3/4	1			
		ap	ae	ap	ae																						
P	0	Ap1 max	0.1 x D	0.5 x D	490	–	660	IPT	.0001	.0001	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049			
	1	Ap1 max	0.1 x D	0.5 x D	490	–	660	IPT	.0001	.0001	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049			
	2	Ap1 max	0.1 x D	0.5 x D	460	–	620	IPT	.0001	.0001	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049			
	3	Ap1 max	0.1 x D	0.5 x D	390	–	520	IPT	.0001	.0001	.0001	.0002	.0004	.0004	.0005	.0007	.0011	.0015	.0020	.0023	.0029	.0034	.0039	.0045			
	4	Ap1 max	0.1 x D	0.5 x D	300	–	490	IPT	.0001	.0001	.0001	.0002	.0003	.0004	.0005	.0007	.0010	.0014	.0018	.0020	.0026	.0030	.0034	.0039			
M	1	Ap1 max	0.1 x D	0.5 x D	300	–	380	IPT	.0001	.0001	.0001	.0002	.0004	.0004	.0005	.0007	.0011	.0015	.0020	.0023	.0029	.0034	.0039	.0045			
	2	Ap1 max	0.1 x D	0.5 x D	200	–	260	IPT	.0001	.0001	.0001	.0002	.0003	.0004	.0004	.0006	.0009	.0012	.0016	.0018	.0023	.0027	.0031	.0036			

CNC shop workbench illustrating the feed-per-tooth adjustment needed for slotting operations.

General Purpose Solid Carbide End Mills • G0mill™ GP

Application Data



■ G0mill GP • 4SE/CH..IS-IR • 4 Flute • Short • Regular

																							
		Side Milling (A) and Slotting (B)		KC633M		Recommended feed per tooth (IPT = inch/th) for side milling (A). For slotting (B), reduce IPT by 20%.																	
		A		B		Cutting Speed – vc SFM		D1 – Diameter															
Material Group			ap	ae	ap	min		max	inch	1/64	1/32	1/16	5/64	3/32	1/8	3/16	1/4	5/16	3/8	1/2	5/8	3/4	1
P	0	Ap1 max	0.1 x D	0.5 x D	490	–	660	IPT	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049	
	1	Ap1 max	0.1 x D	0.5 x D	490	–	660	IPT	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049	
	2	Ap1 max	0.1 x D	0.5 x D	460	–	620	IPT	.0001	.0002	.0004	.0005	.0007	.0009	.0014	.0018	.0023	.0027	.0034	.0040	.0044	.0049	
	3	Ap1 max	0.1 x D	0.5 x D	390	–	520	IPT	.0001	.0002	.0004	.0004	.0005	.0007	.0011	.0015	.0020	.0023	.0029	.0034	.0039	.0045	
	4	Ap1 max	0.1 x D	0.5 x D	300	–	490	IPT	.0001	.0002	.0003	.0004	.0005	.0007	.0010	.0014	.0018	.0020	.0026	.0030	.0034	.0039	
M	1	Ap1 max	0.1 x D	0.5 x D	300	–	380	IPT	.0001	.0002	.0004	.0004	.0005	.0007	.0011	.0015	.0020	.0023	.0029	.0034	.0039	.0045	
	2	Ap1 max	0.1 x D	0.5 x D	200	–	260	IPT	.0001	.0002	.0003	.0004	.0004	.0006	.0009	.0012	.0016	.0018	.0023	.0027	.0031	.0036	

Workbench with the CNC machine and tool holders, reinforcing the importance of following manufacturer speed and feed recommendations.

$$V_f = \cancel{f_n} \times n$$



$$\text{mm/min} = \text{FEED per REV} \times \text{RPM}$$

Workbench with a large drill and a standard end mill, used to illustrate feed per revolution versus feed per tooth.



V_c	Cutting Speed (m/min)
π	Pi (3.14, our $\varnothing$ to Circ. ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, Revolutions/Minute, S-Code)
V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut

Feed rate formula graphic: $V_f = f_n \times n$ and mm/min = FEED per REV $\times$ RPM, with FEED per REV underlined.

SPINDLE SPEED

$n = V_c \times 318 \div D$

**TABLE FEED
END MILL**

$V_f = f_z \times Z \times n$

**TABLE FEED
DRILLS**

$V_f = f_n \times n$

	Cutting Speed (m/min)
π	Pi (3.14, our $\varnothing$ to Circ. ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, Revolutions/Minute, S-Code)
V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut

Feed rate formula graphic: $V_f = f_n \times n$ and mm/min = FEED per REV $\times$ RPM, with RPM circled in red.

SPINDLE SPEED

$$n = V_c \times 318 \div D$$

**TABLE FEED
END MILL**

$$V_f = f_z \times Z \times n$$

**TABLE FEED
DRILLS**

$$V_f = f_n \times n$$

	Cutting Speed (m/min)
π	Pi (3.14, our $\varnothing$ to Circ. ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, Revolutions/Minute, S-Code)
V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut

Legend table listing all variables and their definitions for feed and speed calculations.

SPINDLE SPEED

$$n = V_c \times 318 \div D$$

**TABLE FEED
END MILL**

$$V_f = f_z \times Z \times n$$

**TABLE FEED
DRILLS**

$$V_f = f_n \times n$$

	Cutting Speed (m/min)
π	Pi (3.14, our $\varnothing$ to Circ. ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, Revolutions/Minute, S-Code)
V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut

Feed rate and spindle speed formulas for end mills and drills, with the spindle speed formula circled in red and a legend of variables below.

Using catalogs and digital tools instead of manual calculation

51. Review the same three core formulas for spindle speed and table feed before moving to catalog- or software-based methods.

SPINDLE SPEED

$$n = V_c \times 318 \div D$$

**TABLE FEED
END MILL**

$$V_f = f_z \times Z \times n$$

**TABLE FEED
DRILLS**

$$V_f = f_n \times n$$

V_c	Cutting Speed (m/min)
π	Pi (3.14, our $\varnothing$ to Circ. ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, Revolutions/Minute, S-Code)
V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut

Feed rate and spindle speed formulas for end mills and drills, with table feed formulas circled in red and a legend of variables below.

52. Check whether your tooling catalog already provides RPM and feed rate directly. Many catalogs eliminate manual calculation entirely. For example, a "Z-Carb-HPR" series chart for carbon and alloy steels lists material type, hardness, operation type (profile or slot), cutting speed (V_c), tool diameter (6–20 mm), RPM, feed per tooth (F_z), and feed rate (mm/min) for each diameter.



Close-up of a tractor tire with the calculation "100 mm × 3.14 mm = 314 mm" and "314 mm CIRCUMFERENCE" circled in red.

SPINDLE SPEED

$n = V_c \times 318 \div D$

**TABLE FEED
END MILL**

$V_f = f_z \times Z \times n$

**TABLE FEED
DRILLS**

$V_f = f_n \times n$

V_c	Cutting Speed (m/min)
π	Pi (3.14, our ϕ to Circ. ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, Revolutions/Minute, S-Code)
V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut

Tooling catalog chart for the Z-Carb-HPR series, showing recommended RPM, feed per tooth, and feed rate for various tool diameters and materials.

53. Use the table values directly when provided, without applying the formulas. For example, for a 12 mm tool in alloy steel at a 96 m/min cutting speed:

- RPM = 2545
- Feed per tooth (Fz) = 0.055 mm
- Feed rate = 702 mm/min

SPINDLE SPEED

$n = V_c \times 318 \div D$

**TABLE FEED
END MILL**

$V_f = f_z \times Z \times n$

**TABLE FEED
DRILLS**

$V_f = f_n \times n$

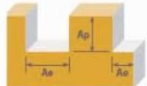
V_c	Cutting Speed (m/min)
π	Pi (3.14, our ϕ to Circ. ratio)
D	Tool Diameter (mm)
n	rev/min (RPM, Revolutions/Minute, S-Code)
V_f	Table Feed (mm/min, F-Code)
f_z	Feed per Tooth (mm)
f_n	Feed per Revolution (mm)
Z	Number of Flutes
a_e	Width of Cut, Radial Depth of Cut
a_p	Depth of Cut, Axial Depth of Cut

Close-up of a feed rate table for alloy steel, showing RPM, feed per tooth, and feed rate for diameters 6–25 mm.

54. Consider using a digital speeds-and-feeds tool. Smartphone apps or PC software can calculate feed rates and speeds quickly and accurately as an alternative to manual formulas.

METRIC

Z-Carb-HPR



Series Z5MCR Metric	Hardness	Ae x D1	Ap x D1	Vc (m/min)	Diameter (D1) (mm)								
					6	8	10	12	16	20			
CARBON STEELS 1018, 1040, 1080, 1090, 10L50, 1140, 1212, 12L15, 1525, 1536	≤ 275 Bhn or ≤ 28 HRC	Profile 	≤ 0.5	≤ 1.5	169	RPM	8967	6725	5380	4484	3363	2690	
					(135-203)	Fz	0.029	0.049	0.061	0.074	0.087	0.099	
					Feed (mm/min)	1291	1650	1650	1668	1463	1327		
		Slot 	1	≤ 1	134	RPM	7109	5332	4265	3555	2666	2133	
					(107-161)	Fz	0.029	0.049	0.061	0.074	0.087	0.099	
					Feed (mm/min)	1024	1308	1308	1322	1160	1052		
	ALLOY STEELS 4140, 4150, 4320, 5120, 5150, 8630, 86L20, 50100	≤ 375 Bhn or ≤ 40 HRC	Profile 	≤ 0.5	≤ 1.5	96	RPM	5089	3817	3054	2545	1909	1527
						(77-115)	Fz	0.022	0.036	0.045	0.055	0.067	0.075
						Feed (mm/min)	550	692	692	702	635	570	
			Slot 	1	≤ 1	76	RPM	4039	3029	2424	2020	1515	1212
(61-91)						Fz	0.022	0.036	0.045	0.055	0.067	0.075	
Feed (mm/min)						436	549	549	557	504	452		
Profile 	≤ 0.5	≤ 1.5	56	RPM	2989	2242	1793	1495	1121	897			
			Fz	0.017	0.030	0.037	0.043	0.051	0.059				

CNC shop workbench, representing the use of digital apps or software to calculate feed rates and speeds quickly.

55. On machines with next-generation controls, use the built-in milling calculator by pressing the Current Commands button.

		Diameter (D ₁) (mm)						
		6	8	10	12	16	20	25
96	RPM	5089	3817	3054	2545	1909	1527	1221
(77-115)	Fz	0.022	0.036	0.045	0.055	0.067	0.075	0.080
	Feed (mm/min)	550	692	692	702	635	570	489

CNC machine control panel where the built-in milling calculator can be accessed via the Current Commands button.

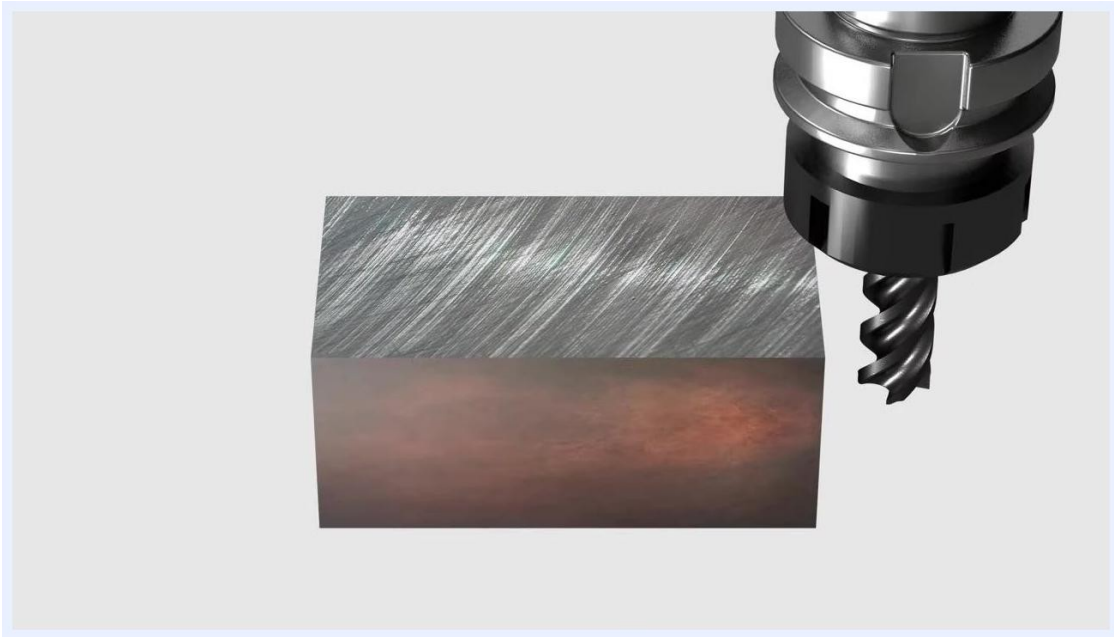
56. Consult additional resources for tapping feed rate calculations, since tapping uses its own set of considerations not covered by the end mill and drill formulas above.

Verification and summary

Before running your CNC program, confirm the following:

- You have identified the correct ISO material group and manufacturer catalog code for your workpiece material.
- You have selected a cutting speed (V_c) appropriate to your material, tool coating, and setup rigidity.
- You have calculated spindle speed using $n = V_c \times 318 \div D$ and entered it as the S-code.
- You have calculated feed rate using $V_f = f_z \times Z_n \times n$ (end mills) or $V_f = f_n \times n$ (feed per revolution) and entered it as the F-code, adjusting f_z downward for slotting versus side milling as recommended.

- Where available, you have cross-checked your calculated values against direct RPM and feed rate tables in the tooling catalog.



Rendered image of a metal block and an end mill, illustrating the concept of tool and material interaction in machining.

What's next

With spindle speed (S-code) and feed rate (F-code) calculated and entered into your program, you are ready to proceed with running the CNC job. For tapping operations specifically, consult additional resources on tapping feed rate calculations, as tapping requires separate considerations beyond the end mill and drill formulas covered in this procedure.



Instructor gesturing to illustrate the importance of setup rigidity for cutting speed selection.

A video player interface showing three "Tip of the Day" videos. The first video, titled "Faster Tapping with Setting", shows a close-up of a drill bit with "SETTING 1.30 = 1" and "SETTING 1.10 = 4" labels, with a duration of 4:02. The second video, titled "Simple Peck Tapping Using", shows a drill bit tapping a hole, with a duration of 3:12. The third video, titled "Easily Program a Tap in Met", shows a close-up of a tap with "13 THREADS" and "1 INCH" labels, with a duration of 7:27. Each video has a "CC" (Closed Caption) button.

CNC shop workbench, indicating that additional resources are available for tapping feed rate calculations.