Computer Numerical Control

Professor Dimitris MOURTZIS

Laboratory for Manufacturing Systems and Automation (LMS)

Department of Mechanical Engineering & Aeronautics

University of Patras

Patras 2022

Table of Contents

Speed and Feeds

Introduction

The efficiency and the life of a cutting tool depend on the cutting feed and the feed rate at which it is run

Cutting Speed

- The cutting speed is the edge or circumferential speed of a tool
- In a machining center or milling machine the cutting speed refers to the edge speed of the rotating cutter
- In a turning center or lathe application the cutting speed refers to the edge speed of the rotating workpiece
- Cutting Speed (CS) is expressed in surface feet per minute (sfm)
- CS is the number of feet a given point on a rotating part moves in one minute
- Proper CS varies from material to material – the softer the material the higher the cutting speed

Speed and Feeds

Introduction

Feed rate is the velocity at which the tool is fed into the workpiece

Feed rates are expressed in two ways:

- inches per minute of spindle travel or Inches per revolution of the spindle
- For milling applications feedrates are generally given in inches per minute (ipm) of spindle travel
- For turning applications feedrates are given in inches per revolution (ipr) of the spindle

WHY Feed Rates are critical for the effectiveness of a job?

- A too heavy feed rate will result in premature burning of the tool
- A too light feed rate will result in tools chipping which rapidly leads to tool burning and breakage

Turning

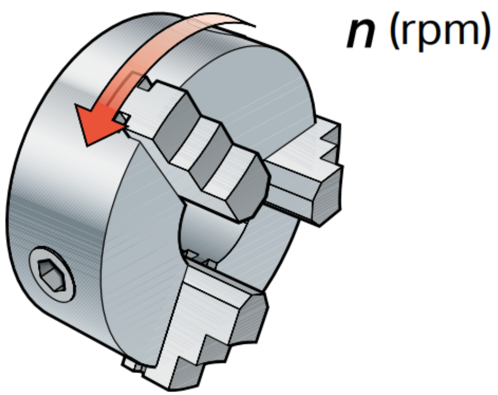
Turning

Definitions of terms

Cutting Speed (CS) and Spindle rpm are two different things:

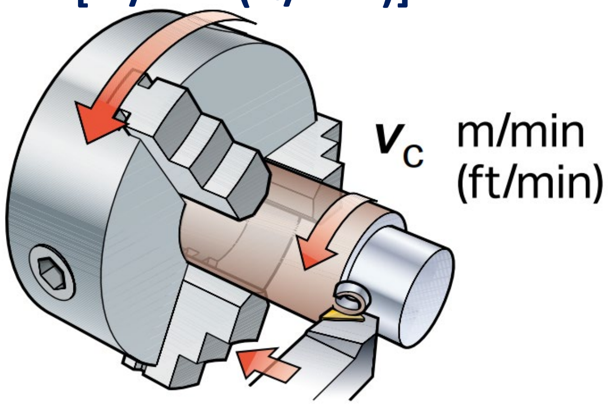
Spindle speed

is the rotation of the chuck and workpiece in rpm (revolution per minute)



Cutting speed

is the surface speed at which the tool moves along the workpiece in feet (meters) per minute , [m/min (ft/min)]



For Turning applications the Diameter of the Workpiece rather than the tool diameter is used to determine the cutting speed and spindle speed

Definition of cutting speed

Metric

$$v_c = \frac{\pi \times D_m \times n}{1000} \quad \text{m/min}$$

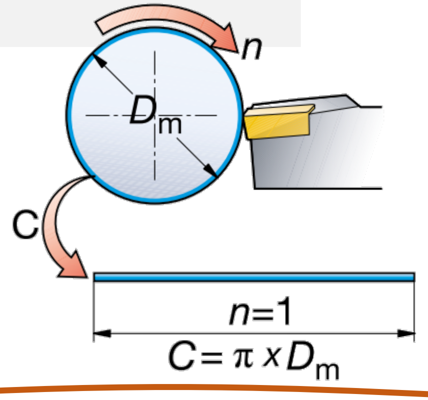
Inch

$$v_c = \frac{\pi \times D_m \times n}{12} \quad \text{ft/min}$$

Where:

- v_c = cutting speed, m/min (ft/min)
- D_m = machined diameter, mm (inch)
- n = spindle speed, (rpm)
- C = Circumference, $\pi \times D_m$ mm (inch)
- π (pi) = 3.14

The circumference (C) is the distance the cutting edge moves in a revolution.



Turning

Example of cutting speed calculation

- The cutting speed differs depending on the workpiece diameter.

Given:
Spindle speed, n = 2000 rpm
Diameter, D_{m1} = 50 mm (1.969 inch)
Diameter, D_{m2} = 80 mm (3.150 inch)

Metric

$$v_c = \frac{\pi \times D_m \times n}{1000} \text{ m/min}$$

$$v_{c1} = \frac{3.14 \times 50 \times 2000}{1000} = \textbf{314 m/min}$$

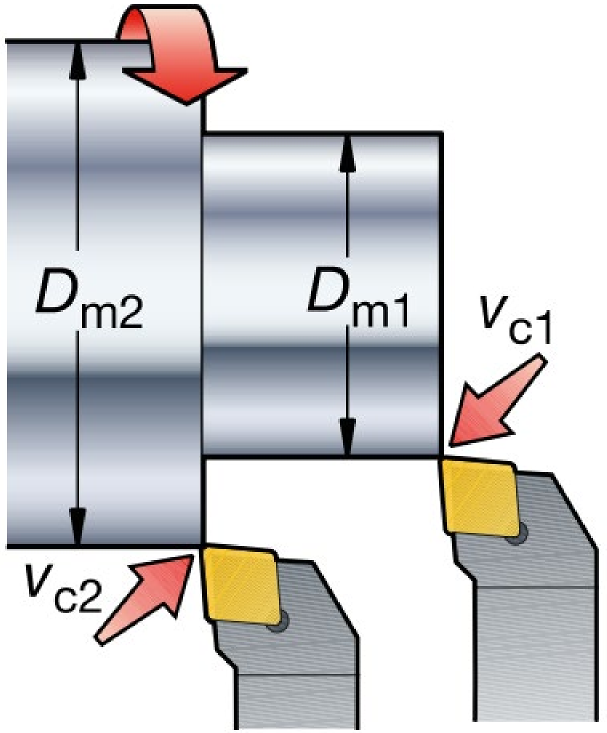
$$v_{c2} = \frac{3.14 \times 80 \times 2000}{1000} = \textbf{502 m/min}$$

Inch

$$v_c = \frac{\pi \times D_m \times n}{12} \text{ ft/min}$$

$$v_{c1} = \frac{3.14 \times 1.969 \times 2000}{12} = \textbf{1030 ft/min}$$

$$v_{c2} = \frac{3.14 \times 3.150 \times 2000}{12} = \textbf{1649 ft/min}$$



Turning

Definitions of terms

Spindle speed

The workpiece rotates in the lathe, with a certain spindle speed (n), at a certain number of revolutions per minute (rpm).

Surface/cutting speed

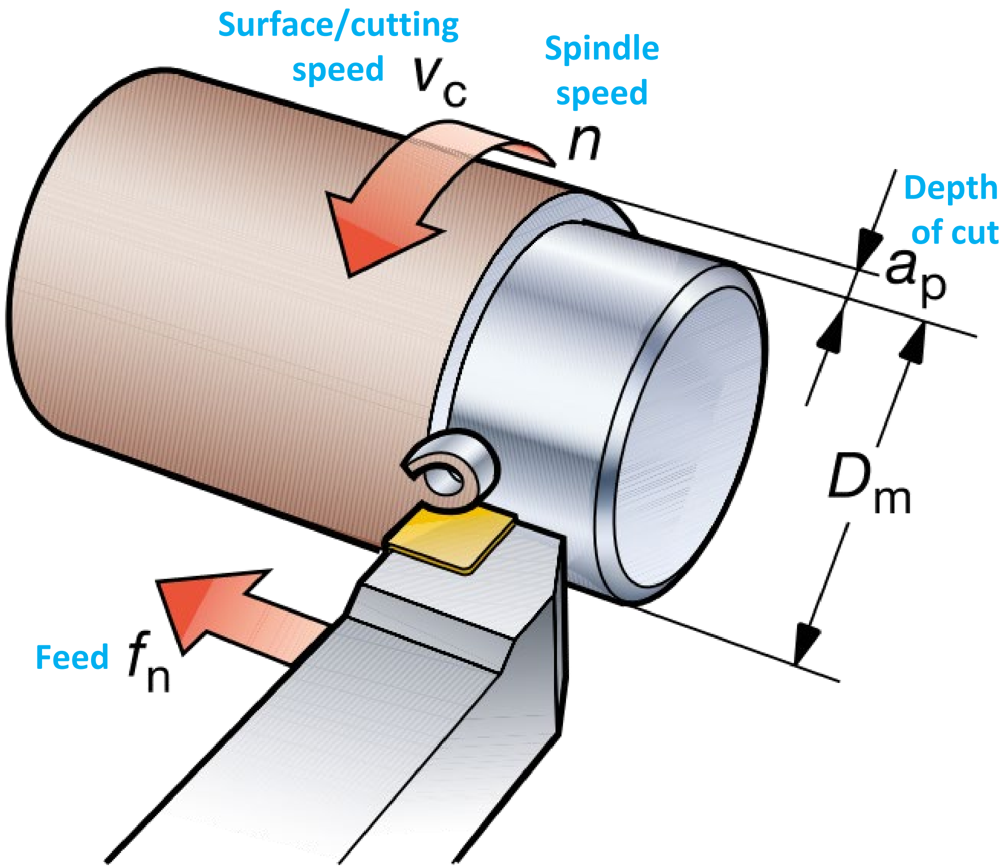
The cutting speed (v_c) in m/min (ft/min) at which the periphery of the cut workpiece diameter passes the cutting edge

Feed

The cutting feed (f_n) in mm/r (inch/r) is the movement of the tool in relation to the revolving workpiece

Depth of cut

The cutting depth (a_p) in mm (inch) is half of the difference between the un-cut and cut diameter of the workpiece.



n = spindle speed (rpm)
v_c = cutting speed m/min (ft/min)
f_n = cutting feed mm/r (inch/r)
a_p = depth of cut mm (inch)

Turning

Calculating cutting data

Given:
Cutting speed, v_c = 400 m/min (1312 ft/min)
Diameter D_m = 100 mm (3.937 inch)

Example of how to calculate the spindle speed (n) from cutting speed (v_c)

Metric

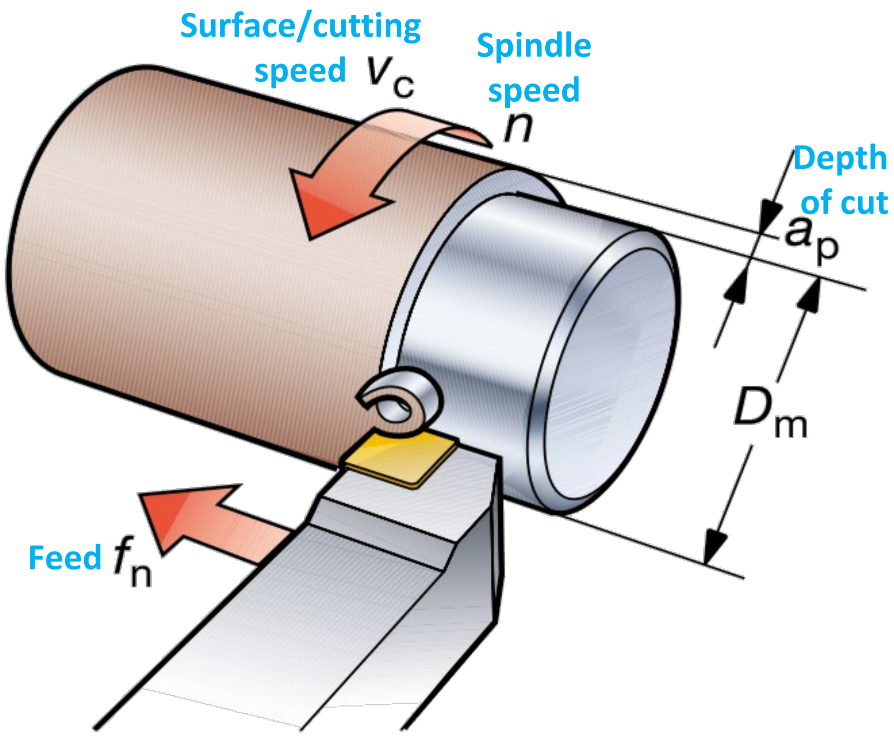
$$n = \frac{v_c \times 1000}{\pi \times D_m} \quad \text{r/min}$$

$$n = \frac{400 \times 1000}{3.14 \times 100} = \mathbf{1274} \quad \text{r/min}$$

Inch

$$n = \frac{v_c \times 12}{\pi \times D_m} \quad \text{r/min}$$

$$n = \frac{1312 \times 12}{3.14 \times 3.937} = \mathbf{1274} \quad \text{r/min}$$

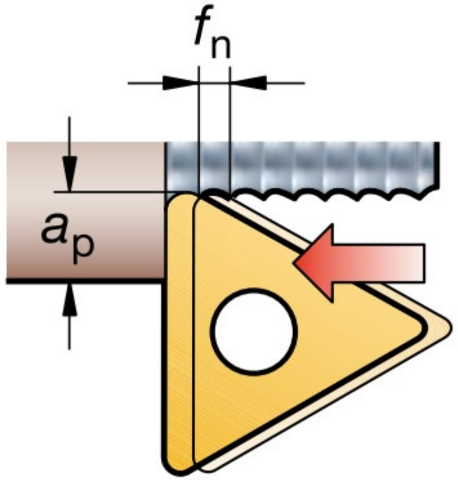


Turning

Definitions of terms

Feed rate

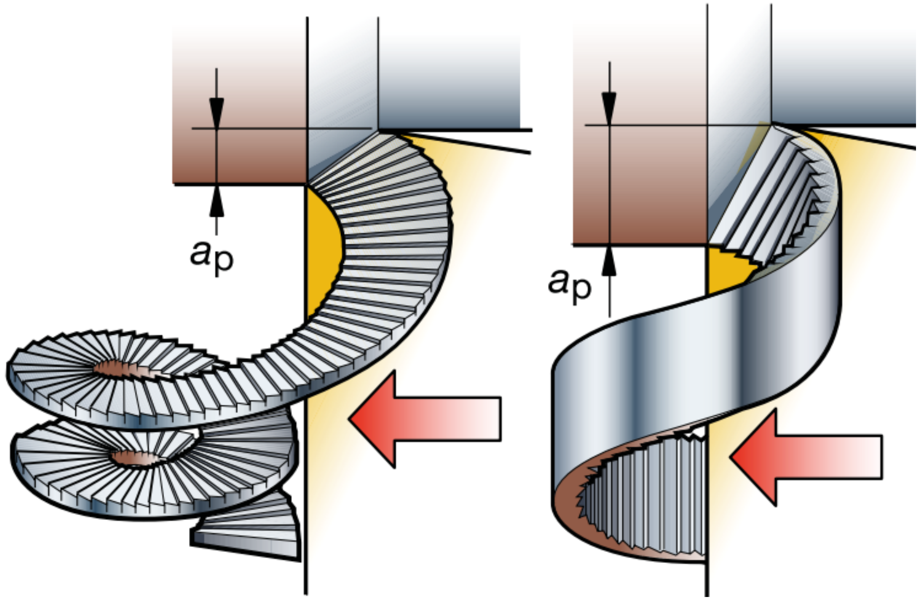
The feed rate (f_n) is the distance the edge moves along the cut per revolution



Cutting depth and chip formation

The cutting depth (a_p) is the length the edge goes into the workpiece

Chip formation varies with depth of cut, entering (lead) angle, feed, material and insert geometry



Turning Feed rates

- The vast majority of tools used with NC are inserted tools
- The feed rates vary with:
  - Material type
  - Insert Type
- Tables of manufacturers' catalogues and machining data handbooks are the best sources for turning feedrates

WHY the values given in tables are starting points?

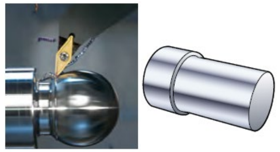
- Conditions which are also affect CS and feed rates are the following:
  - Part geometry
  - Machine rigidity
  - Machine setup
- The actual CS and feed rate used during the run will ultimately be determined when the first piece is run during the job setup

Computer Numerical Control

Turning

Process planning

1. Component



Dimensions,
type of operation,
Workpiece material,
quantity

Material:
- Machinability
- Hardness

Component:
- Quality demands
- Number of passes
- Tolerances

2. Machine

Machine parameters

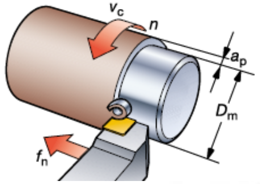
Important machine considerations:
- Stability, power and torque, Component clamping
- Tool position, Tool changing times/number of tools in turret
- Spindle speed (rpm) limitations

3. Choice of tool

Type of turning tool:
- External/internal
- Facing
- Profiling

Different types of tools and inserts have their pros and cons:
E.g. Turning with rhombic inserts:
- ✓ Operational versatility
- ✓ Good roughing strength.
- × Can cause vibration when turning small components.

4. How to apply



Cutting data,
tool path, etc.

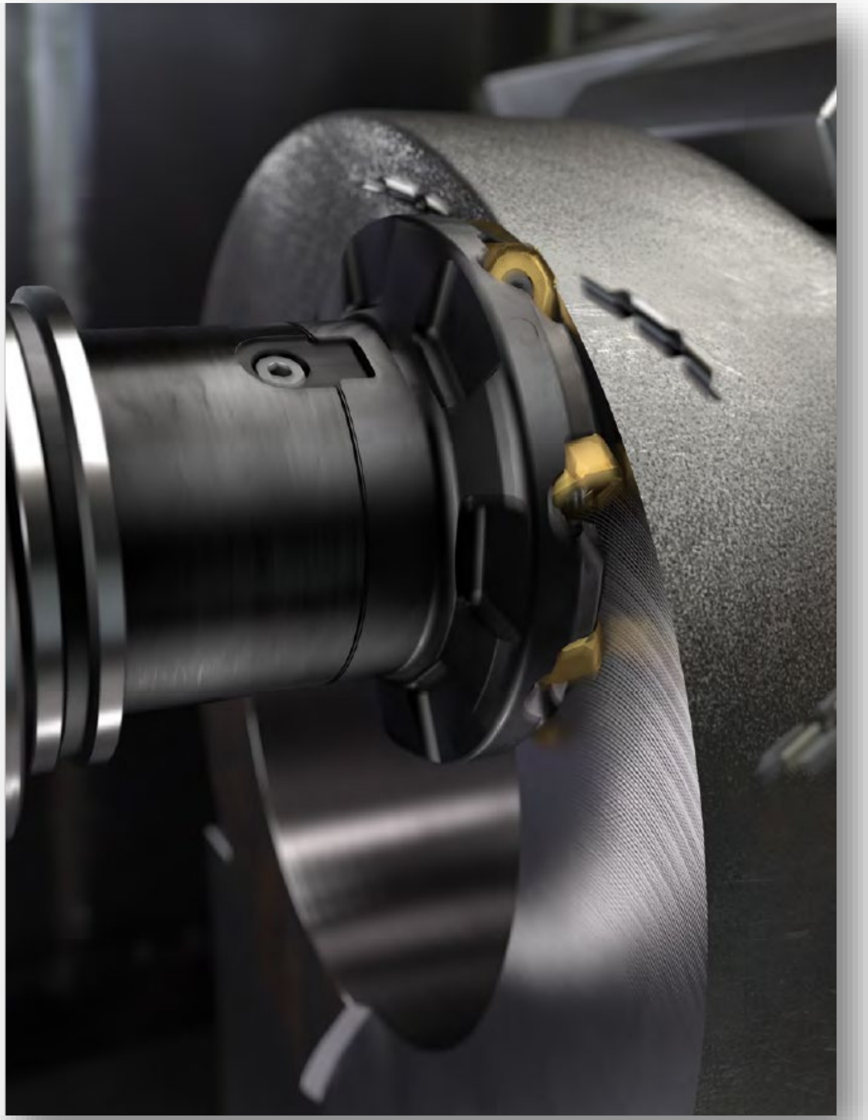
The tool path has a significant impact on the process:
- Chip control
- Insert wear
- Surface quality
- Tool life

5. Troubleshooting

Remedies
and solutions

Chip breaking, Insert wear

Milling



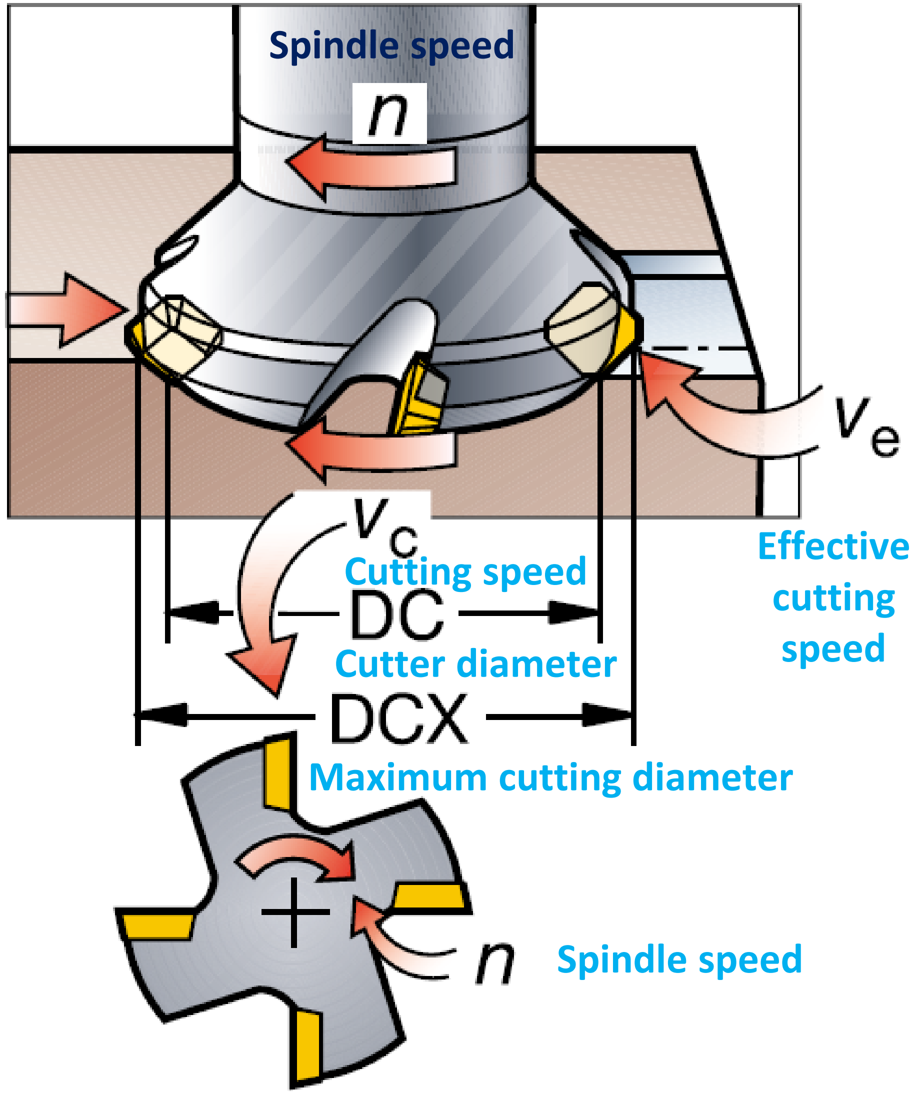
Milling

Definitions of terms

Spindle speed (n) in rpm is the number of revolutions the milling tool on the spindle makes per minute.

Cutting speed (v_c) in m/min (ft/min) indicates the surface speed at which the cutting edge machines the workpiece.

Specified cutter diameter (DCX), having an effective cutting depth to diameter (DC), which is the basis for the cutting speed v_c or v_e

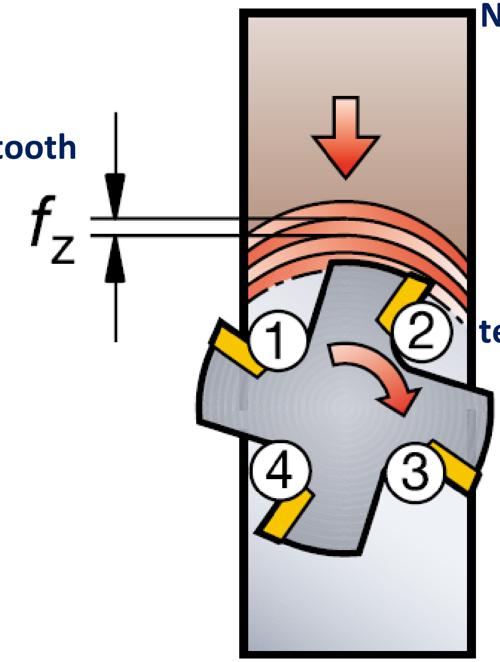


For Milling applications the Diameter of the Tool is used to determine the cutting speed and spindle speed

Milling

Definitions of terms

- **Feed per tooth, f_z** mm/tooth (inch/tooth), is a value in milling for calculating the table feed. The feed per tooth value is calculated from the recommended maximum chip thickness value.
- **Feed per minute, v_f** mm/min (inch/min), also known as the table feed, machine feed or feed speed is the feed of the tool in relation to the workpiece in distance per time-unit related to feed per tooth and number of teeth in the cutter.
- The number of available cutter teeth in the tool (z_n) varies considerably and is used to determine the table feed while the effective number of teeth (z_c) is the number of effective teeth in cut.
- Feed per revolution (f_n) in mm/rev (inch/rev) is a value used specifically for feed calculations and often to determine the finishing capability of a cutter.



$$V_f = f_z \times Z_c \times n$$

z = Feed per tooth mm/tooth (inch/tooth)

v_f = Table feed mm/min (inch/min)

z_n = Number of cutter teeth (pcs)

z_c = Effective number of teeth (pcs) [in engagement]

f_n = Feed per revolution mm/rev (inch/rev) $[f_n = f_z \times z_c]$

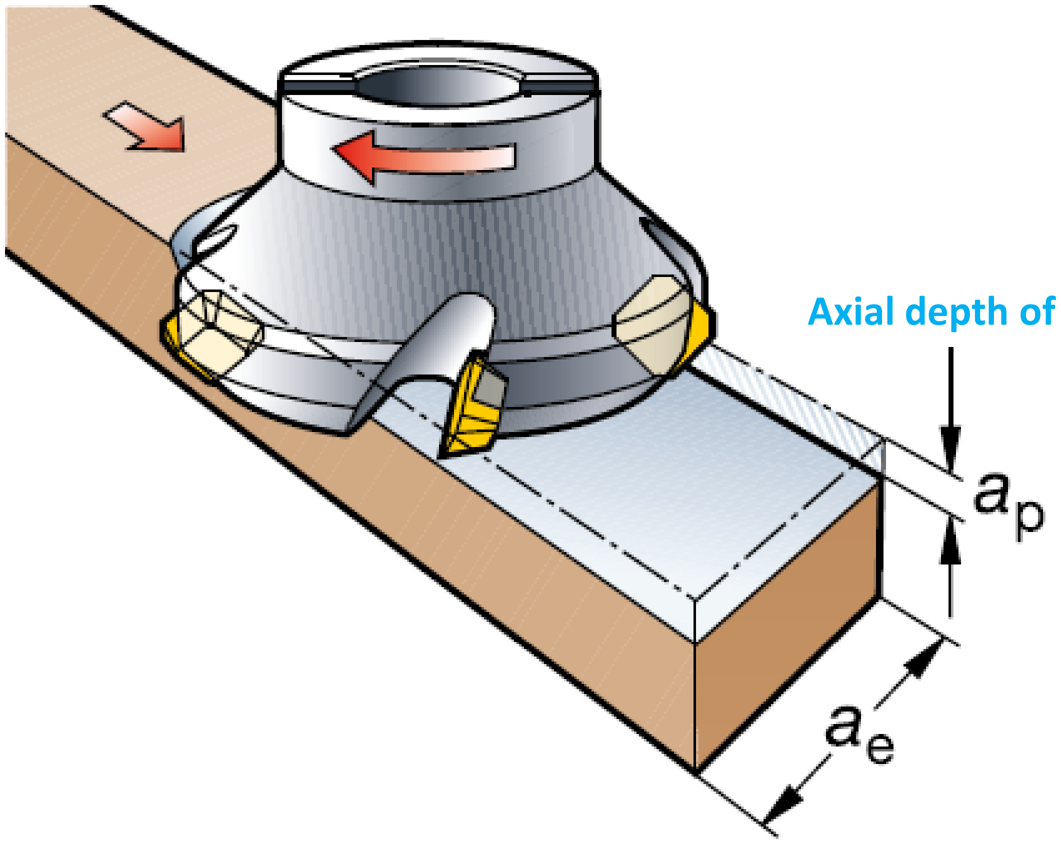
n = Spindle speed (rpm)

Milling

Definitions of terms

- Axial depth of cut, a_p mm (inch), is what the tool removes in metal on the face of the workpiece. This is the distance the tool is set below the unmachined surface.

- Radial cutting width, a_e mm (inch), is the width of the component engaged in cut by the diameter of the cutter. It is the distance across the surface being machined or, if the tool diameter is smaller, that is covered by the tool.



Milling

Calculating cutting data

Example in face milling

Given:
Cutting speed, v_c = 225 m/min (738 ft/min)
Feed per tooth, f_z = 0.21 mm (.0082 inch)
Number of cutter teeth Z_n=5
Cutter diameter DC= 125mm (4.92inch)

Need:
- Spindle speed, n (rpm)
- Table feed, v_f (mm/min (inch/ min)

1. Spindle speed

Given: vc = 225 m/min (738 ft/min)

Metric

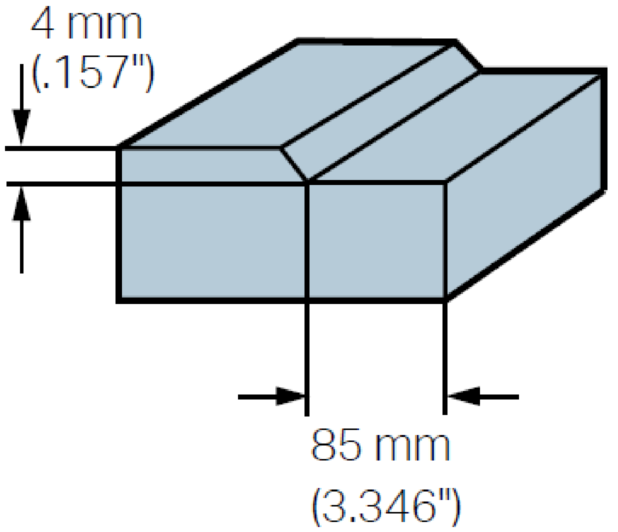
$$n = \frac{v_c \times 1000}{\pi \times D_m} \quad \text{r/min}$$

$$n = \frac{225 \times 1000}{3.14 \times 125} = \mathbf{575} \ \text{r/min}$$

Inch

$$n = \frac{v_c \times 12}{\pi \times D_m} \quad \text{r/min}$$

$$n = \frac{738 \times 12}{3.14 \times 4.921} = \mathbf{575} \ \text{r/min}$$



2. Table feed

$v_f = n \times f_z \times z_n$ mm/min

$v_f = 575 \times 0.21 \times 5 = 600$ mm/min

$v_f = n \times f_z \times z_n$ inch/min

$v_f = 575 \times .0082 \times 5 = 23.6$ inch/min

Milling Feed rates

- Feeds used in milling not only depend on the spindle rpm but also on the number of teeth on the cutter
- The milling feed rate is calculated to produce a desired chip load on each tooth of the cutter
- Example: In end milling chip load should be 0.002 inch to 0.006 inch
- The recommended chip loads for various mill cutters are given in machinists' handbooks
- For inserted cutters manufacturers' catalogue will list recommended chip loads for a given insert

Milling Feed rates

- To calculate the feed rate for a mill cut the following formula is used

$$\mathbf{F = R \times T \times rpm}$$

Where :

- F = the milling feed rate expressed (inches/minute)
- R = the chip load per tooth
- T = the number of teeth on the cutter
- rpm = the spindle speed in revolutions per minute

- Milling feed rates are also affected by:
  - Machine rigidity
  - Set up
  - Part geometry

Milling Feed rates

- In the case of inserted milling cutters Chip Thickness affects feed rates too
- This is not the chip load on the tooth but the actual thickness of the chip produced at a given feedrate
- Chip thickness will vary with the geometry of the cutter:
  - Positive Rake
  - Negative Rake
  - Neutral Rake

(Rake Angle is the angle the chips flow away from the cutting area)

- Chip thickness values: 0.004 inch to 0.008 inch
- Chip thickness less than or greater than these values will place either too little or too great pressure on the insert for efficient machining
- Once a feed rate is calculated the chip thickness it produces should be derived
- IF the chip thickness is out of the limit THEN the feedrate should be adjusted to bring it in to acceptable limits

Milling Feed rates

- Chip Thickness can be calculated by the following formula:

$$CT = \sqrt{\frac{W}{D}} \times R$$

Where:

- CT = the chip thickness
- W = the width of the cut
- D = the diameter of the cutter
- R = the feed per tooth

Milling Feed rates

- IF the Chip Thickness is too small a modification of the preceding formula can be used to determine an acceptable feed rate

$$f = \sqrt{\frac{D}{W}} \times CT$$

Where :

- f = the feed per tooth being calculated
- D = the diameter of the cutter
- CT = the desired chip thickness

- The new calculated value of the Feed per Tooth can be then substituted back into the feed rate formula and a new Feed rate is then calculated

Speed and Feed Example

Given:
Diameter, D = 1,750 inch
number of teeth on the cutter T= 4

An aluminium workpiece is to be milled using a carbide inserted mill cutter. The cutter is 1,750 inch diameter x 4 flute

Need:
What should be the appropriate Spindle rpm and Milling Feed rate?

- **Step 1: Calculate Spindle Speed (rpm) with the following formula:**

$$rpm = \frac{CS\,x12}{D\,x\,\pi}$$

- **Step 2: Select CS = 1000 sfm (surface feet per minute) for Aluminum**

$$rpm = \frac{1000 \times 3,82}{1,75} = 2183$$

(3.82 is derived from 12 divided (π))
The number 12 is used to convert the inch value of the part diameter into feet
Remember, we measure our parts in inches but use feet in cutting speed calculations.

Speed and Feed Example

- Step 3: Calculate Feedrate with the following formula:

$$F = R \times T \times rpm$$

- Step 4: Select R = 0.004 (chip load per tooth) – values are 0.002 to 0.006

$$F = 2183 \times 4 \times 0,004$$

$$F = 34,91\, inches/min$$

- Step 5: Calculate the chip thickness to insure that the inserts will not break down prematurely: It is assumed Width of the Cut = 1.000 inch wide

$$CT = \sqrt{\frac{W}{D}} \times R$$

$$CT = \sqrt{\frac{1.000}{1,750}} \times 0.004$$

$$CT = 0,00302$$

- Step 6: CT is less than the recommended min of 0.004 and the feed per tooth must be calculated

Speed and Feed Example

- **Step 7:** Calculate **Feed per tooth** with the following formula and CT = 0,008

$$f = \sqrt{\frac{D}{W}} \times CT$$

$$f = \sqrt{\frac{1,75}{1.000}} \times 0,008$$

$$f = 0,010$$

- **Step 8:** The new value for the chip load per tooth is substituted in the feed rate formula and recalculate Feedrate:

$$F = 2183 \times 4 \times 0.010$$

$$F = 87.32\ inches/min$$

Conclusion:

- The 2813 rpm spindle speed and 87.32 inches per min feed rate are "book value" rates
- They will have to be adjusted up or down depending on the machine, fixture and workpiece

Calculating Feed Rates

- To calculate the feed rate for a mill cut the following formula can also be used:

$$F_{\mathrm{m}} = f_t \times n_t \times \mathrm{N}$$

Where :

- F_m = Milling feed rate expressed in inches per minute
- f_t = Feed in inches / tooth
- n_t = number of teeth on the tool
- N = Spindle speed in revolutions per minute(rpm)

(Oberg, E. & Jones F. D. & Horton, H. L. & Ryffell, H. H. (2000). Machinery's Handbook, 26th ed., New York, NY: Industrial Press Inc.
Kibbe, R.R., Neely, J.E., Meyer, R.O., & White, W.T. (2002). Machine tool practices. Upper Saddle River, NJ: Prentice Hall.)

Recommended Tool Feed

Table 3 : Tool Feed

	Tool Feed (in/tooth)		
Material	Face Mill	Side Mill	End Mill
Magnesium	.005-.020	.004-.010	.005-.010
Aluminum	.005-.020	.004-.010	.005-.010
Brass and Bronze	.004-.020	.004-.010	.005-.010
Copper	.004-.010	.004-.007	.004-.008
Cast Iron (Soft)	.004-.016	.004-.009	.004-.008
Cast Iron (Hard)	.004-.010	.002-.006	.002-.006
Milt Steel	.004-.010	.002-.007	.002-.010
Alloy Steel (Hard)	.004-.010	.002-.007	.002-.006
Tool Steel	.004-.008	.002-.006	.002-.006
Stainless Steel	.004-.008	.002-.006	.002-.006
Titanium	.004-.008	.002-.006	.002-.006
High Manganese Steel	.004-.008	.002-.006	.002-.006
Note: Double Speed for Carbide Cutting Tools			

Feed Rate Calculation Example

Calculate the Feed Rate for End Milling Aluminum with a 2 flute, ½ inch HSS end mill

- **Step 1:** Selection of f_t (Feed in inches / tooth) from table 3

Table 3 : Tool Feed

	Tool Feed (in/tooth)		
Material	Face Mill	Side Mill	End Mill
Magnesium	.005-.020	.004-.010	.005-.010
Aluminum	.005-.020	.004-.010	.005-.010
Brass and Bronze	.004-.020	.004-.010	.005-.010
Copper	.004-.010	.004-.007	.004-.008

f_t=0.005 in./ tooth

Feed Rate Calculation Example

- **Step 2:** Calculation of n_t (number of teeth on the tool) : $n_t = 2$

- **Step 3:** Calculation of Spindle Speed :

$$N = \text{rpm} = \frac{CS \times 4}{D} \Rightarrow N = \frac{250 \times 4}{0.5} \Rightarrow N = 2000 \text{ rpm}$$

Feed Rate Calculation Example

- **Step 4:** Calculation of the feed rate of the milling cutter using the formula below :

$$F_{\mathrm{m}} = f_t \times n_t \times \mathrm{N}$$

$$F_{\mathrm{m}} = 0.005 \times 2 \times 2000$$

$$F_{\mathrm{m}} = 20 \ in/min.$$

Feed Rate Calculation Example 2

Calculate the Feed Rate for Face Milling Aluminum with a 4 flute, ¾ inch HSS end mill

- **Step 1**: Selection of f_t (Feed in inches / tooth) from table 3

$$f_t = 0.005 \text{ in. / tooth}$$

- **Step 2:** Calculation of n_t (number of teeth on the tool) :

$$n_t = 4$$

- **Step 3:** Calculation of Spindle Speed :

$$\mathrm{N} = rpm = \frac{CS \times 4}{D}$$

$$\mathrm{N} = \frac{250 \times 4}{0.75}$$

$$\mathrm{N} = 1333.33 rpm$$

Feed Rate Calculation Example 2

- **Step 4**: Calculation of the feed rate of the milling cutter using the formula below :

$$F_m = f_t \times n_t \times N$$

$$F_m = 0.005 \times 4 \times 1333.33$$

$$F_m = 26.67 \text{ in/min.}$$

Milling

Process planning

1. Component

Geometric shape, Material, Tolerances

Material:
- Machinability
- Hardness
- Chip forming

Geometric shape
- Flat surface
- Deep cavities
- Thin walls/bases
- Slots

Tolerances:
- Dimensional accuracy
- Surface finish

2. Machine

Machine parameters

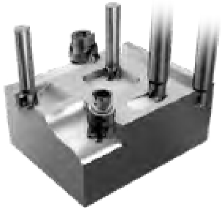
Important machine considerations:
- Age/condition – stability
- Horizontal/vertical, Spindle type and size
- Number of axes/configuration, Workpiece clamping.

3. Choice of tool

Type of cutter

Different types of tools and inserts have their pros and cons:
E.g. Cutters with round inserts:
- ✓ Very flexible for face milling and profiling
- ✓ High performance multi-purpose cutter
- × Round inserts require more stable machines

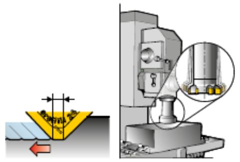
4. How to apply



Cutting data, method etc.

Number of cutting edges,
Insert geometry, Stability, Chip formation through cutter positioning

5. Troubleshooting



Remedies and solutions

Insert wear and tool life, Unsatisfactory surface finish, Vibration

Drilling

Drilling

Definitions of terms

Definition of cutting speed

Metric

$$v_c = \frac{\pi \times DC \times n}{1000} \text{ m/min}$$

Inch

$$v_c = \frac{\pi \times DC \times n}{12} \text{ ft/min}$$

Spindle speed

Metric

$$n = \frac{v_c \times 1000}{\pi \times DC} \text{ r/min}$$

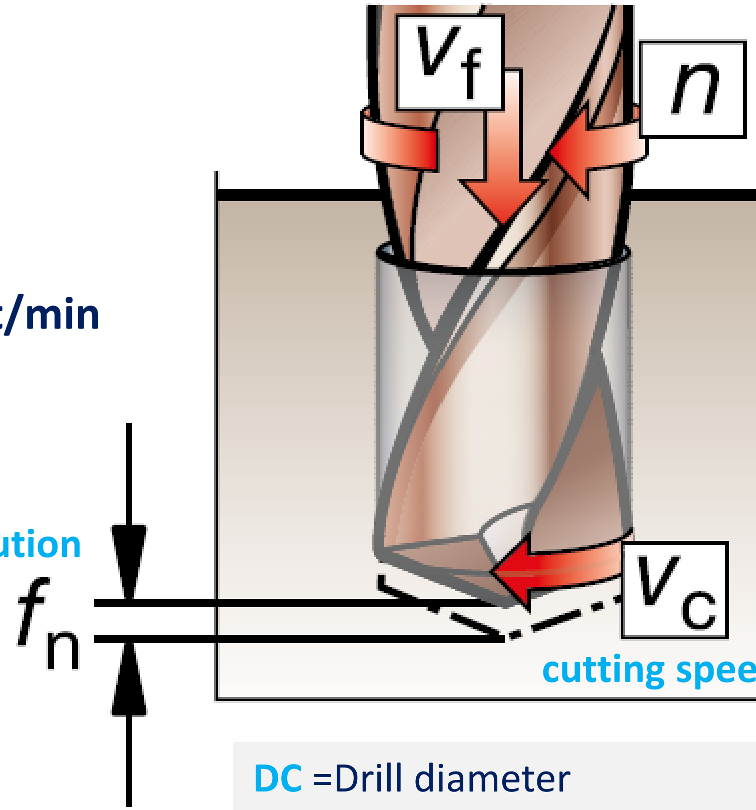
Inch

$$n = \frac{v_c \times 12}{\pi \times DC} \text{ r/min}$$

Penetration rate

$$v_f = f_n \times n \quad \text{mm/min (inch/min)}$$

Productivity in drilling is strongly related to the penetration rate, v_f



DC =Drill diameter
f_n =Feed per revolution
n = Spindle speed
v_c =Cutting speed
v_f =Penetration rate

Drilling Feed rates

- Drilling feed rates depend on the drill diameter
- Values for HSS drills from tables in machinists’ handbooks

Table 1. Cutting Speeds for common materials

MATERIAL	CUTTING SPEED
Tool steel	50
Cast iron	60
Mild steel	100
Brass, soft bronze	200
Aluminum	250
Magnesium	300

Drilling Feed rates

- Drilling feed rate is calculated by using the formula below

$$\text{ipm} = \text{rpm} \times \text{ipr}$$

Where :

ipm = the required feed rate expressed in inches per minute

rpm= the programmed spindle speed in revolutions per minute

ipr = the drill feed rate to be used expressed in inches per revolution

Recommended Drilling Feeds

Table 2 : Drilling Feeds

Drilling Feeds

Drill Diameter (in.)	Drill Feed Rate (ipr)
< ⅛	.001-.002
⅛ – ¼	.002-.004
¼ - ½	.004 - .007
½ - 1	.007 - .015
> 1	.015-.025

Drill Feed Example

What tool feed rate should be used for drilling a .375 inch hole in aluminum?

- **Step 1:** Tool Feed Rate (ipm) can be calculated by the following formula:

$$\text{ipm} = \text{rpm} \times \text{ipr}$$

- **Step 2:** Calculation of Spindle Speed (rpm) with the formula below (CS for Aluminum is selected by table 1, 250):

$$\text{rpm} = \frac{\text{CS} \times 4}{\text{D}} \rightarrow \text{rpm} = \frac{250 \times 4}{0.375} \rightarrow \text{rpm} = 2666$$

- **Step 3:** Select Drill diameter : ¼ - ½ , Drill feed from table 1 :.004 - .007

$$\text{ipm} = 2666 \times 0.005 \rightarrow \text{ipm} = 13,33$$

Drilling

Process planning

1. Component

Hole dimension and quality
Workpiece material, shape and quantity

Material:
- Machinability
- Hardness
- Chip breaking

Component:
- Clamping, hole size and depth.
- Component features, does something complicate the process?
- Is the component sensitive to feed force and/or vibrations?

2. Machine

Machine parameters

Important machine considerations:
- Machine stability
- Spindle speed, Coolant supply, Coolant flow and pressure
- Clamping of the workpiece, Tool magazine
- Horizontal or vertical spindle, Power and torque

3. Choice of tool

Type of tool and machining method

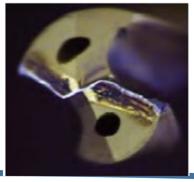
The basic parameters are:
- Diameter, Depth, Quality (tolerance, surface finish)
- The hole type, and the required precision affect tool choice.

4. How to apply

Cutting data, coolant etc.

Chip formation and evacuation is the dominant factor in drilling and affects hole quality.

5. Troubleshooting



Remedies and solutions

Insert wear and tool life, Chip evacuation, Hole quality and tolerances, Cutting data

Laboratory for Manufacturing Systems & Automation (LMS)
Dept. of Mechanical Engineering & Aeronautics
University of Patras, Greece

Professor Dimitris MOURTZIS

www.lms.mech.upatras.gr