Αριθμητικός Έλεγχος Εργαλειομηχανών

Ενότητα 3: Process Planning and Tool Selection

Δημήτρης Μούρτζης, Επίκουρος Καθηγητής
Πολυτεχνική Σχολή
Τμήμα Μηχανολόγων & Αεροναυπηγών Μηχανικών

COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece

Dr. Dimitris Mourtzis

Assistant Professor

Patras, 2015

Objectives of Section 3

- List the steps involved in process planning
- List the factors that influence the selection of an NC machine, workholding devices, and tooling
- Describe the types of tools available for hole operations
- Describe the types of tools available for milling operations
- Determine the proper grade of carbide insert for a given material
- Describe some common NC turning tool types
- Determine the proper spindle RPM to obtain a given cutting speed
- Explain the importance of proper feedrates

Process Planning

DEFINITION: **Process Planning** is the term used to describe the development of an ***NC part program***

Decisions which must be made by the NC programmer to successfully program a part:

- **Machine Selection:** Which NC machine should be used?
- **Fixturing:** How will the part be held in the machine?
- **Strategy**: What machining operations & strategy will be used?
- **Tool Selection:** What cutting tools will be used?

Process Planning

Machine Selection: This decision is based on a number of factors:

- What is the programmer's ***experience***?
- What ***machines*** are available?
- ***How many parts*** are in the order?
- Are there enough parts to justify the ***setup time*** and higher per hour ***run cost*** on a more complex machine?
- Is the particular part best suited for a ***lathe*** or a ***milling*** machine application?
- Is the ***vertical*** or ***horizontal*** spindle preferred?

  Note: *Vertical spindles are advantageous for hole drilling and boring operations. The horizontal orientation of the spindle causes the chips to fall away from the tool, whereas vertical spindles tend to keep the chips packed around the tool*

Process Planning

Fixturing: Decision on how the workpiece should be held

- Will standard ***holding devices*** (clamps, mill vises, chucks, and so on) suffice, or will ***special fixturing*** need to be developed?

- What ***quantity*** of parts will be run?

  __Note__: A large number of parts mean that special fixturing to shorten the machining cycle may be feasible, even if conventional workholding methods would otherwise be used

- How ***elaborate*** does the fixturing need to be?

  __Note__: If many part runs are foreseen, a more durable fixture must be designed. If only one or two part runs are projected, a simpler fixture can be used.

- What will make the best ***quality*** part?

Process Planning

Machining Strategy:

Must be developed before the NC program can be written and ***machining sequences*** used in a part program are determined by the following decisions

- What is the programmer's ***experience***?
- What is the ***shape*** of the part
- What is the blueprint ***tolerance***?
- What ***tooling*** is available?
- ***How many*** parts are in the order?

Process Planning

Tool Selection:

The final important step in process planning based on the following decisions

- What tools are *available*?
- What ***machining strategy*** is to be used?
- ***How many parts*** are in the order?

  Note: If a large number of parts are in the order, special timesaving tools can be made or purchased
- What are the blueprint ***tolerances***?
- What ***machine*** is being used?

Process Planning

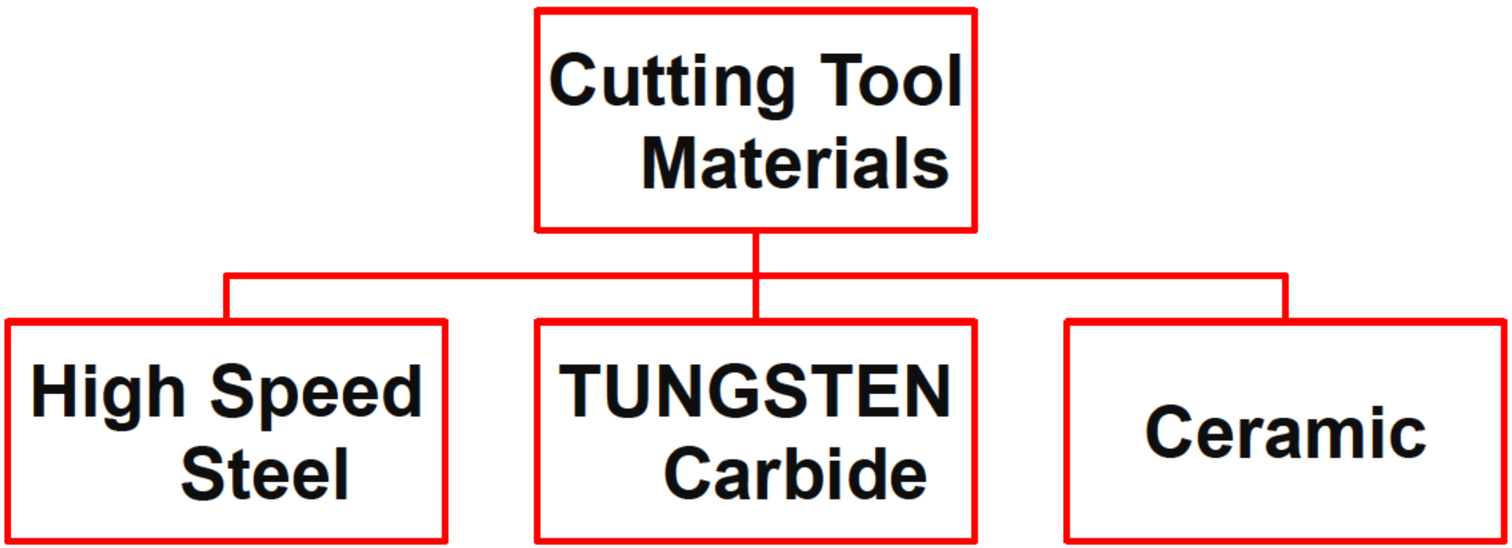
NC Setup Sheet:

- The programmer must communicate to the setup personnel in the shop what tools and fixtures are to be used in the NC program
- The information is placed on Setup Sheets
- The Setup Sheet should contain all necessary information to prepare for the job
- Special instructions to the setup personnel or machine operators should be included
- Special notes regarding tooling should also be included

Tooling for Numerical Control

Cutting Tool Materials

Cutting Tools are available in three basic types:



Tooling for Numerical Control

High Speed Steel (HSS)

HSS tools have the following *advantages* over *Carbide*:

- HSS ***costs less*** than Carbide or Ceramic tooling
- HSS is ***less brittle*** and not as likely to break during interrupted cuts
- The tools can be ***resharpened*** easily

HSS tools have the following *disadvantages*:

- HSS does not hold up as well as Carbide or Ceramic at the high temperatures generated during machining
- HSS does not cut hard materials well

Tooling for Numerical Control

Tungsten Carbide (Carbide)

Carbide Tools come in one of three basic types

- Carbide Tools
  - Solid Carbide Tools
  - Brazed Carbide Tools
  - Inserted Carbide Tooling

Tooling for Numerical Control

Tungsten

Symbol: W
Melting point: 3,422° C
Atomic number: 74
Boiling point: 5,552° C
Atomic mass: 183.84

Tungsten Carbide (Carbide)

- WC can be prepared by reaction of tungsten metal and carbon at 1400–2000 °C
- **Reacting tungsten hexachloride with hydrogen, as a reducing agent, and methane, as the source of carbon at 670 °C (1,238 °F)**

● $WCl_6 + H_2 + CH_4 \rightarrow WC + 6\ HCl$

- **Reacting tungsten hexafluoride with hydrogen, as reducing agent, and methanol, as source of carbon at 350 °C (662 °F)**

● $WF_6 + 2\ H_2 + CH_3OH \rightarrow WC + 6\ HF + H_2O$

Tooling for Numerical Control

Tungsten Carbide

- ***Solid Carbide Tools*** are made from a solid piece of carbide
- ***Brazed Carbide Tools*** use a carbide cutting tip brazed in a steel shank
- ***Inserted Carbide Tooling*** utilizes indexable inserts made of carbide which are held in steel tool holders

TUNGSTEN Carbide have the following *advantages* over *HSS*:

- Carbide ***holds up well*** at elevated temperatures
- Carbide can ***cut hard materials*** well
- Solid carbide tools ***absorb workpiece vibration*** and reduce the amount of ‘’chatter’’ generated during machining
- When inserted cutters are used, the ***inserts can be easily changed*** or indexed, rather than replacing the whole tool

Tooling for Numerical Control

Tungsten Carbide

TUNGSTEN Carbide have the following *disadvantages* over *HSS*:

- Carbide ***costs more*** than High Speed Steel Tools
- Carbide is ***more brittle*** than HSS and has a tendency to chip during interrupted cuts
- Carbide is ***harder to resharpen*** and requires diamond grinding wheels

Tooling for Numerical Control

Ceramic Tooling

- Has made great advances in the past several years
- Once very expensive – Some Ceramic inserts cost now less than a Carbide

Ceramic has the following *advantages*:

- Ceramic is sometimes ***less expensive than carbide*** when used in insert tooling
- Ceramic will cut ***harder materials at a faster rate***
- Ceramic has ***superior heat hardness***

Ceramic has the following *disadvantages*:

- Ceramic is ***more brittle*** than HSS or carbide
- Ceramic must run within its given surface speed parameters

Note: ***If run too slowly, the insert will break down quickly. Many machines do not have the spindle RPM range needed to use ceramics***

Tooling for Numerical Control

Fields of Application

- **High Speed Steel** is used on:
  - Aluminum alloys
  - Other non ferrous alloys
- **Carbide** is used on:
  - High silicon aluminums
  - Steels
  - Stainless steels
  - Exotic metals
- **Ceramic inserts** are used on:
  - Hard steels
  - Exotic metals

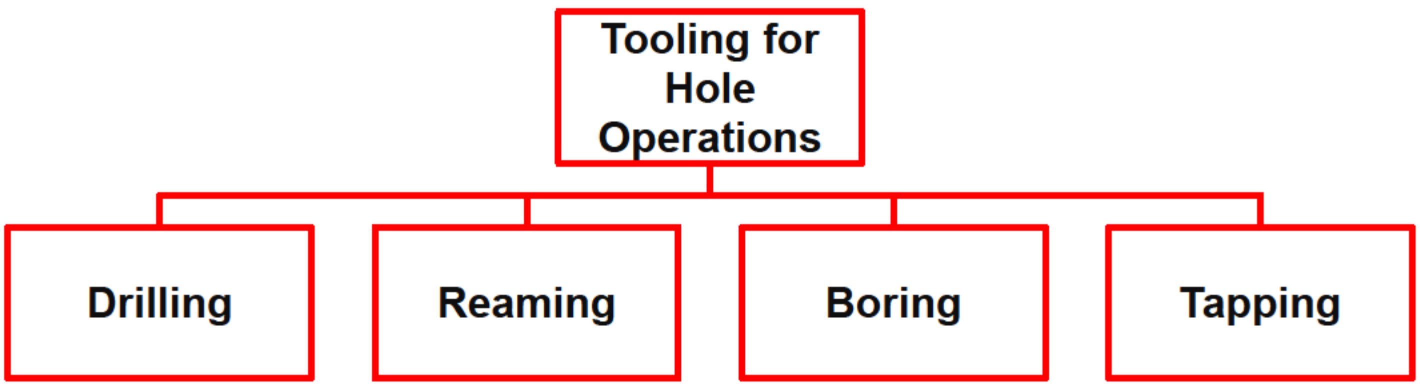
***Note 1:* Inserted Carbide Tooling is becoming the preferred for any CNC application**

***Note 2:* Some Carbide inserts are coated with special substances (e.g. titanium nitride) increasing tool life up to 20 time – using recommended cutting speeds and feedrates**

Tooling for Hole Operations

Tooling for Hole Operations

There are four basic hole operations that are performed on NC machinery



Tooling for Hole Operations

Drilling

- Drills are available in different styles for different materials
- Twist drills remain one of the most common tools for making holes
- Drills have a tendency to walk as drill, resulting in a hole that it is not truly straight
- Center drills (Fig. 3-4) are often used to predrill a pilot hole to help twist drill to start straight
- Drills also produce triangular- shaped holes

FIGURE 1 Tapered shank twist drill

FIGURE 2 Center drills

Tooling for Hole Operations

Drilling

- If the hole tolerance is ***closer than 0.003 inch*** a secondary hole operation should to used to size the hole such as ***Boring*** or ***Reaming***
- Large holes are sometime produced by ***spade drills*** (Fig. 3-5)
- The flat blades in spade drills allow good chip flow and economical replacement of the drill tip

FIGURE 3 Spade drill

Tooling for Hole Operations

- **Drill point angle** must be considered when selecting a drill
- The harder the material to be cut the grater the drill point angle needs to be to maintain satisfactory tool life
- Mild steel is usually cut with a 118-degree included angle drill point
- Stainless steels often use a 135-degree drill point

Types of Drills

- HSS drills are the most common
- Brazed carbide and solid carbide
- Carbide drill chip when drilling holes
- When drilling hard materials Cobalt drills are used (HSS with Cobalt)
- Cobalt drills have greater heat hardness than HSS drills

Tooling for Hole Operations

Reaming

- Reaming is used to remove a small amount of metal from an existing hole as a finishing operation
- Reaming is a precision operation which will hold a tolerance of +/- 0.0002 inch easily
- Reaming needs a pilot hole
- Reamers are expensive
- Straight fluted reamers

FIGURE 4 Morse taper reamer

Tooling for Hole Operations

Reaming

- Spiral fluted reamers
- Spiral fluted reamers produce better surface finishes than straight flutes
- Spiral fluted reamers are more difficult to resharpen than straight fluted
- Reamers are available in three basic tool materials:
  - HSS
  - Brazed carbide
  - Solid carbide

Tooling for Hole Operations

Boring

Boring removes metal from an existing hole with a single-point boring bar

- **Boring heads are available in two designs**:
  - ***Offset*** in which the boring bar is a separate tool inserted into the head
  - ***Cartridge*** which use an adjustable insert in place of a boring bar
- **Boring bars are available in four material types:**
  - High Speed Steel (HSS)
  - Solid carbide – up to ½-inch diameter
  - Brazed carbide – up to ½-inch diameter
  - Inserted carbide - for large holes
- **Boring Bars move of-centre, produce very round, straight hole, tight specs**

Tooling for Hole Operations

Tapping

Taping is used to produce internally threaded holes <u>(Milling, Turning)</u>

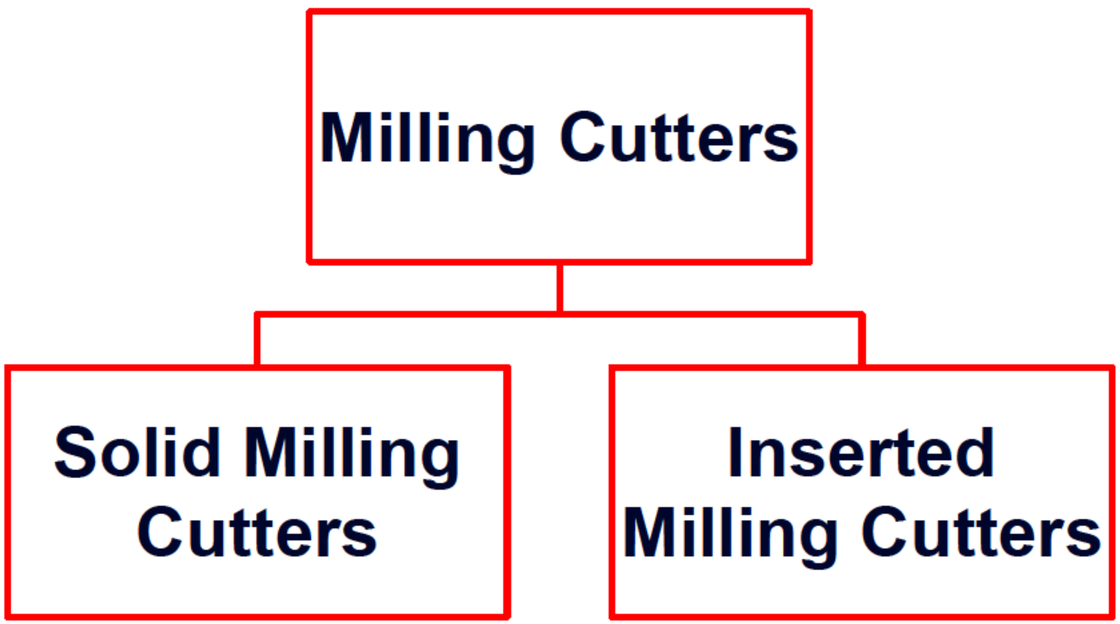
- **They are available in different flute designs**:
  - ***Standard machine screw taps*** (Fig. 3-9) are widely used when tapping blind holes
  - ***Spiral pointed taps*** (gun taps) which are preferred for thru-hole operations – shoot chips forward and out of the bottom of the hole
  - ***High-spiral taps*** are used for soft, stringy material (e.g. Aluminum)

FIGURE 5 Machining screw tap

Milling Cutters

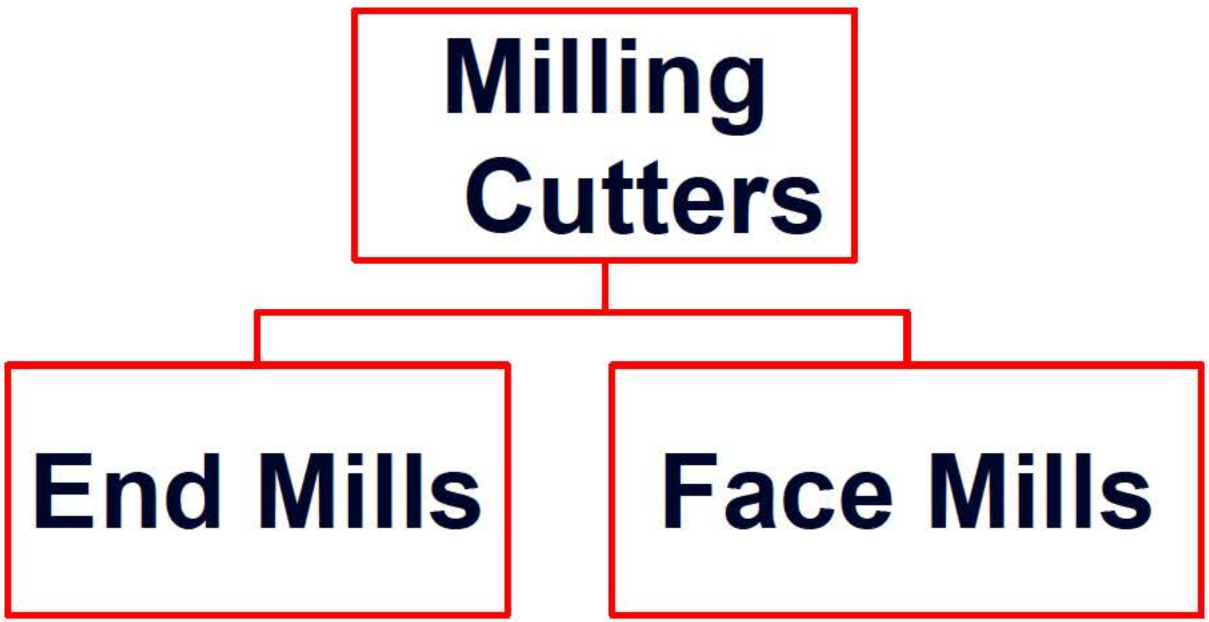
Milling Cutters

- The greatest advances in tooling for NC have taken in the area of ***Inserted Milling Cutters***
- ***Milling*** allows the ***contouring*** capabilities of the NC machine to be used to efficiently perform operations that would require special tooling if done manually



Milling Cutters

Can also be further classified in:



Milling Cutters

Thread Hob

- A special milling cutter used to mill a thread in a workpiece
- ***Thread hobs*** make use of an NC machine's helical interpolation capabilities

FIGURE 6 Milling cutter

Milling Cutters

End Mills

- **End Mills** are available in:
  - High Speed Steel (HSS)
  - Solid Carbide

- **End Mills** are available in diameters:
  - From 0.032 inch to 0.500 inch

- **Inserted End Mills** are available in diameters:
  - From 0.500 inch to 3 inch

FIGURE 7 Several types of end mills

***Note 1**: Two-flute cutters with deeper gullets are well suited for*

***Note 2**: Four-flute end mills are more rigid because of their thic*

Milling Cutters

End Mills

- **Inserted cutters** are preferred for NC applications Inserts are less expensive to replace than an entire tool
- By indexing the inserts four or six cutting edges can be used on one insert
- When the insert is used up it is thrown away rather than re-sharpened
- Inserted cutters may used on many types of workpiece materials by changing the inserts from one designed for Aluminum to one designed for Stainless Steel

- **Ball End Mills** using inserts
- ***Ball End Mills*** are also available in ***HSS*** and ***Solid Carbide***
- ***Ball Mills*** are used for three, four or five – axis contouring work where Z axis is used
- They are also used to produce a radius in a part

Milling Cutters

Face Mills

- **Face Mills** are designed to remove large amounts of material from the face of the workpiece

- **Face Mills** are manufactured in:

  - High Speed Steel (HSS)
  - Brazed Carbide
  - Inserted Carbide (the most common type of facing tool)

- **Face Mills** are available in two sizes: From 2 inch to over 8 inch in diameter

- ***Note 1: The cost of HSS and Brazed Carbide limit their application to special situations***

Special Inserted Cutters

- A number of ***special tools*** have been developed for ***use with CNC***
- The NC programmer is always confronted with new ideas to ***improve productivity***
- ***Prospective and experienced*** programmers should spent time looking at tooling catalogs to become acquainted with current tooling developments

Special Inserted Cutters

Carbide Inserts and their Selection

- ***Carbide Inserts*** **are manufactured in a variety of** ***TYPES*** **and** ***GRADES***
- **The** ***TYPE*** **of the insert describes the** ***SHAPE*** **of the insert**

Common Insert SHAPES

- **Triangular**
- **80^{o} Diamond**
- **55^{o} Diamond**
- **Round**

Special Inserted Cutters

Carbide Inserts and their Selection

- The ***GRADE*** of insert describes the ***HARDNESS*** of the insert and the application for which it was developed

- Each ***TYPE*** of insert is identified by a ***Designation Code***

- The Identification System used on an insert will vary depending on the manufacturer (Fig. 3-25)

Special Inserted Cutters

Carbide Insert Grading System:

- Each ***GRADE*** of Carbide is designated by an ***ANSI "C"*** number from ***C1 to C8***
- Each ***GRADE*** of Carbide has also been classified by ***ISO***
- The ISO designation uses ***"K" or "P"*** number depending on insert hardness
- In the ***USA*** the ***ANSI*** system is generally used
- In ***other countries*** the ***ISO*** is followed
- Manufacturers develop their own GRADE system based on the ANSI or ISO rating
- The programmer is necessary to consult the individual manufacturers catalog to arrive the proper grade number

Speed and Feeds

Speed and Feeds

The efficiency and the life of a cutting tool depend on the cutting feed and the feedrate 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

Cutting Speed Data

- The following rates are averages for ***high-speed steel (HSS)*** cutters
- For ***carbide cutters***, ***double the cutting speed value***

Cutting speeds for LATHES:

MATERIAL	CUTTING SPEED (sfm)
Tool steel	50
Cast iron	60
Mild steel	100
Brass, soft bronze	200
Aluminum, magnesium	300

Speed and Feeds

Cutting Speed Data

Cutting Speed for DRILLS

MATERIAL	CUTTING SPEED (sfm)
Tool steel	50
Cast iron	60
Mild steel	100
Brass, soft bronze	200
Aluminum, magnesium	300

Cutting speeds for MILLING

MATERIAL	CUTTING SPEED (sfm)
Tool steel	40
Cast iron	50
Mild steel	80
Brass, soft bronze	160
Aluminum, magnesium	200

Cutting Speed

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

  Example:

  - A 0.250-inch diameter drill turning at 1,200 rpm has a CS of ca 75 sfm
  - A 0.500-inch diameter drill turning at 1,200 rpm has a CS of ca 150 sfm
- The spindle necessary ***rpm*** to achieve a ***given CS*** can be calculated by the formula:

$$\boxed{\mathrm{rpm} = \frac{\mathrm{CS\ x\ 12}}{\mathrm{D} \times \pi}}$$

Where : CS = cutting speed in surface feet per minute (sfm)

D = diameter in inches of the tool or workpiece diameter for lathe

π = 3.1416

Speed and Feeds

Cutting Speed

- The cutting speed of a ***particular tool*** can be determined from the rpm using the formula:

$$CS = \frac{D \times \pi \times rpm}{12}$$

- On the shop floor the formulas are often simplified
- The following formulas will yield results similar to the formulas just given:

$$\text{rpm} = \frac{\text{CS} \times 4}{\text{D}}$$

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

Speed and Feeds

Important Note

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

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

Speed and Feeds

Feedrate

Feedrate is the velocity at which the *tool is fed into the workpiece*

Feedrates are expressed in two ways:

- inches per minute of spindle travel
- 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?

- ***Too heavy a federate will result in premature burning of the tool***
- ***Too light a federate will result in tools chipping which rapidly leads to tool burning and breakage***

Speed and Feeds

TURNING Feedrates

- The vast majority of tools used with NC are inserted tools
- The feed rates vary with:
  - Material type
  - Insert Type
- Tables of manufacturers' catalogs 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 feedrates are the following:***
  - ***Part geometry***
  - ***Machine rigidity***
  - ***Machine setup***
- ***The actual CS and feedrate used during the run will ultimately be determined <u>when the first piece is run during the job setup</u>***

Speed and Feeds

Drilling Feedrates

- **Drilling feedrates depend on the drill diameter**
- Values for HSS drills from tables in machinists' handbooks

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

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

Where : ipm = the required feedrate expressed in inches per minute
rpm= the programmed spindle speed in revolutions per minute
ipr = the drill feedrate to be used expressed in inches per revolution

Speed and Feeds

Milling Feedrates

- Feeds used in milling not only depend on the ***spindle rpm*** but also on the ***number of teeth*** on the cutter
- The milling feedrate 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' catalog will list recommended ***chip loads*** for a given insert

Milling Feedrates

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

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

Where : F = the milling feedrate expressed in inches per minute

R = the chip load per tooth

T = the number of teeth on the cutter

rpm = the spindle speed in revolutions per minute

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

Speed and Feeds

Milling Feedrates

- In the case of inserted milling cutters ***Chip Thickness*** affects feedrates 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

***Note**: 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 *feedrate is calculated* the *chip thickness it produces* should be derived
- IF the chip thickness is out of the recommended range THEN the feedrate should be adjusted to bring it in to acceptable limits

Milling Feedrates

- ***Chip Thickness*** can be calculated by the following formula:

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

Where : CT = the chip thickness

W = the width of the cut

D = the diameter of the cutter

R = the feed per tooth

Milling Feedrates

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

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

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 <u>Feed per Tooth</u> can be then substituted back into the feedrate formula and a new Feedrate calculated***

Speed and Feed Example

- An aluminium workpiece is to be milled using a carbide inserted mill cutter
- The cutter is 1.750 inch diameter x 4 flute

What should be the appropriate Spindle rpm and Milling Feedrate?

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

$$\boxed{\mathrm{rpm} = \frac{\mathrm{CS \ x \ 12}}{\mathrm{D} \times \pi}}$$

- ***Step 2:*** Select CS = 1,000 sfm (surface feet per minute) for Aluminum

$$\boxed{rpm = \frac{1,000 \times 12}{1.75 \times 3.1416} = 2,183}$$

Speed and Feed Example

- ***Step 3:*** Calculate Feedrate with the following formula:

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

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

$$\boxed{F = 2.183 \times 4 \times 0.004} \qquad \boxed{F = 34.91\ inches / \min}$$

- ***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

$$\boxed{CT = \sqrt{\frac{W}{D}} \times R} \qquad \boxed{CT = \sqrt{\frac{1.000}{1.750}} \times 0.004} \qquad \boxed{CT = 0.00302}$$

- ***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 3:*** The new value for the chip load per tooth is substituted in the feedrate formula and recalculate Feedrate:

$$F = 2138 \times 4 \times 0.010$$

$$F = 87.32\ inches / \min$$

Conclusion:

- ***The 2,813 rpm spindle speed and 87.32 inches per min feedrate are "book value" rates***
- ***They will have to be adjusted up or down depending on the machine, fixture and workpiece***

Summary

- Process planning is the term used to describe the steps the programmer uses to develop and implement a part programming
- The steps in process planning are: determine the machine, determine the workholding, determine the machining strategy, select the tools to be used
- Tool selection is important to the efficiency of the NC program
- Cutting tools for NC are made in high-speed steel, tungsten carbide, and ceramic
- Inserted cutters are the preferred tools for NC use

Summary

- Inserts are manufactured in different grades with different applications intended
- Cutting speed is the edge speed of the tool; it is a function on the spindle rpm and the tool diameter
- Feedrates that are too heavy will result in excess tool wear and premature tool failure
- Feedrates that are too light will result in chipped tools and premature tool failure
- When calculating milling feedrates, chip thickness must be considered

Vocabulary Introduced in this Section

- Chip thickness
- Cutting speed (CS)
- Feedrate
- High speed steel (HSS)
- Methodizing
- Process planning
- NC setup sheet
- Tungsten carbide

End of Section

Funding

- This educational material has been developed in the teaching duties of the respective educator.
- The Project "**Open Academic Courses at the University of Patras**" has funded only the reformation of the educational material.
- The Project is implemented within the context of the Operational Programme "Education and Lifelong Learning" (EdLL) and is co-funded by the European Union (European Social Fund) and national resources.

European Union
European Social Fund
OPERATIONAL PROGRAMME
EDUCATION AND LIFELONG LEARNING
investing in knowledge society
MINISTRY OF EDUCATION, LIFELONG LEARNING AND RELIGIOUS AFFAIRS
MANAGING AUTHORITY
Co- financed by Greece and the European Union
NSRF
2007-2013
programme for development
EUROPEAN SOCIAL FUND

Reference Note

License Note

LMS

Notes Preservation

Any reproduction or modification of this material must include:

- the Reference Note
- the License Note
- the Notes Preservation statement
- the Third Parties’ Works Note (if exists)

as well as the accompanying hyperlinks.

Third Parties' Works Note

This Work makes use of the following works:

Figure 1: slide 19, CC BY-SA,
https://commons.wikimedia.org/wiki/File:DrillsMorseTaperShank1234.jpg

Figure 2: slide 19, CC BY-SA 2.0,
https://en.wikipedia.org/wiki/Drill_bit#/media/File:CenterDrills123456.jpg

Figure 3: slide 20, Free for Personal/Business use
http://publicphoto.org/industries/concrete-drill-bits/#

Figure 4: slide 22, CC BY-SA 2.0
https://en.wikipedia.org/wiki/Reamer#/media/File:ReamerMorseTaper3.jpg

Third Parties' Works Note

This Work makes use of the following works (cont.):

Figure 5: slide 25, CC BY-SA 2.5, https://commons.wikimedia.org/wiki/File:Machine-screw-tap-1.JPG

Figure 6: slide 28, CC 0 Public Domain, https://pixabay.com/en/taps-drill-milling-milling-machine-444506/

Figure 7: slide 29, CC BY-SA 2.0, https://en.wikipedia.org/wiki/End_mill#/media/File:MillingCutterSlotEndMillBallnose.jpg