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
Associate professor

Patras, 2016
Chapter 3:  
Process Planning and Tool Selection
# Table of Contents

Chapter 3: Process Planning and Tool Selection

3.1 Steps involved in Process Planning .......................................................... 5
3.2 A Processing Example ........................................................................... 6
3.3 Tooling for Numerical Control ............................................................... 13
3.4 Fundamentals of Machining ................................................................. 21
3.5 Turning Operations ................................................................................ 34
3.6 Milling Operations ................................................................................ 37
3.7 Hole Operations .................................................................................... 42
3.8 Speed & Feeds ...................................................................................... 57

---

Laboratory for Manufacturing Systems and Automation
Dr. Dimitris Mourtzis
Objectives of Chapter 3

- List the steps involved in **process planning**
- List the **factors** that influence the selection of an NC machine, work-holding 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

Process planning can be defined as the function, which establishes the sequence of the manufacturing processes to be used in order to convert a part from an initial to a final form, where the process sequence incorporates process description, the parameters for the process and possibly equipment and/or machine tool selection.


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?

("By failing to prepare, you are preparing to fail" - Ben Franklin)
Figure 3-8: The collaboration between CAM, CAPP and CAD systems (Ming et al. 2008)
**Process Planning**

![Diagram showing the flow of information in CAD/CAM/CNC systems](image)

**Figure 3-9**: Manufacturing information flow in the state-of-the-art CAD/CAM/CNC chain (Newman et al. 2008)
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.
Machine Selection Example

Figure 3-1: Three possible configurations of machine structure (T. Moriwaki, 2008)
Fixturing: Decision on how the workpiece should be held

- Will standard holding devices (clamps, mill vises, chucks, etc.) suffice, or will special fixturing need to be developed?

- What quantity of parts will be run?
  
  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?
  
  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?
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?
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?
- What are the blueprint tolerances?
- What machine is being used?

*Note: If a large number of parts are in the order, special timesaving tools can be made or purchased.*
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.

Figure 3-2: NC Setup Sheet for a CNC machining center
Process Planning

NC Setup Sheet

- **Special instructions** to the setup personnel or machine operators should be included
- **Special notes** regarding tooling should also be included

Figure 3-3 NC Setup Sheet for a CNC lathe
A Processing Example

Example:

- A part to be machined from aluminium casting
- The cast has 0.250-inch diameter of stock to be removed from 4.000 and 3.000–inch diameters
- The center of the cast was cored to 1.000-inch
- The 1.000-inch height was cast at 1.250 inch
- The 4.000-inch diameter and the 0.38-inch are to be done on a conventional lathe
- The part will be routed on a Vertical NC Machining Center
- A fixture for clamping the part on the CNC vertical machining center is needed

Figure 3-4: Part drawing
A Processing Example

The sequence of the machining operation at the vertical NC machining center was planned as follows:

1. Face the 1.000 and .25 dimensions using a 3\(\frac{1}{4}\) carbide inserted face mill.
2. Center drill the .188 and .250 diameter holes. A 90-degree center drill was chosen. The 90-degree chamfer will provide an edge break at the drilled hole, thereby reducing the amount of deburr time.
3. Drill the .188 diameter holes using a 3\(\frac{1}{16}\) drill. Since drills almost always drill .001 or more oversize, the hole will be comfortably within tolerance.
4. Drill the .250 diameter hole using a 1\(\frac{1}{4}\) drill.
5. Mill the 3.000 diameter using a 1\(\frac{1}{4}\) diameter inserted helical end mill. The end mill has inserts up the sides of the insert, allowing side cutting up to 2.00 deep.
6. Using the same end mill, mill the 1.500 diameter bore.
A Processing Example

**MANUFACTURING PROCESS**

<table>
<thead>
<tr>
<th>Part Number: Adapter</th>
<th>Job Number: 000-000-001</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>OPERATION NUMBER</th>
<th>OPERATION CODE</th>
<th>DESCRIPTION OF OPERATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>010</td>
<td>issue</td>
<td>Issue 356 alum. castings</td>
</tr>
<tr>
<td>020</td>
<td>manual lathes</td>
<td>Chuck on 3.250 as cast dia.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Turn 4.000 ± .010 b/p dim to 4.000 ± .001 dia. (tooling dimension).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Face .38 b/p dim.</td>
</tr>
<tr>
<td>030</td>
<td>vert. mach. center</td>
<td>Locate parts in fixture NCF-000-100</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Drill .188 + .006 – .001 dia. thru 6 plcs.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Drill .250 + .006 – .001 dia. thru 4 plcs.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Bore 1.500 ± .010 dia. thru 1 plc.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Mill the 3.000 ± .010 dia., hole the 1.000 and .25 dims.</td>
</tr>
<tr>
<td>040</td>
<td>burr</td>
<td>• Deburr parts as required.</td>
</tr>
<tr>
<td>050</td>
<td>insp</td>
<td>• Inspect parts for b/p conformance.</td>
</tr>
</tbody>
</table>

Figure 3-5: Manufacturing process for part shown in Figure 3-4
A Processing Example

The fixture design was based on the following factors:

- The 4.000 diameter and .38 dimensions were completed in the previous operation, making this feature the logical choice for locating the part.

- The run quantity is only 200 parts. The fixture design is simple, making it economical to build.

- The design is easy to load.
A Processing Example

Fixture Concept

- The fixture is used to hold the part
- The fixture is developed by the NC programmer
- The part will be nested in the 4.0015-inch diameter fixture bore
- The part will be clamped with 4 swivelling clamps
- The swivelling clamps are purchased from the tooling supplier

Figure 3-6 Fixture concept
## A Processing Example

### Figure 3-7: NC setup sheet for CNC machining center

<table>
<thead>
<tr>
<th>STA. NO.</th>
<th>CRO REG.</th>
<th>TOOL DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>D11</td>
<td>3 1/4 INSERTED CARBIDE FACE MILL</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>NO. 4 x 90° C’DRILL</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>3/16 DRILL (.1875 DIA.)</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>1/4 DRILL (.250 DIA.)</td>
</tr>
<tr>
<td>5</td>
<td>D15</td>
<td>1 1/4 INSERTED CARBIDE HELICAL END MILL</td>
</tr>
</tbody>
</table>

**NOTES:** TOOL NO. 2 REQUIRES 1.125 MIN EFF. LENGTH

**TAPE NUMBER:** 1000  
**FIXTURE:** NCF-000-100  
**TABLE LAYOUT:**

**DRWN:** WSS  
**PROG:** WSS  
**DATE:** 3-4-89  
**B/P REV:** A  
**MACHINE:** UNIVERSAL VERT. MACH. CENTER  
**OPER. NO:** 030
Cutting Tool Materials

Cutting Tools are available in three basic types:

- High Speed Steel
- TUNGSTEN Carbide
- Ceramic
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 *re-sharpened* 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
Tungsten Carbide (Carbide)

Carbide Tools come in one of three basic types:

- Solid Carbide Tools
- Brazed Carbide Tools
- Inserted Carbide Tooling
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 has 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
Figure 3-10: Effect of cobalt content in tungsten-carbide tools on mechanical properties. Note that hardness is directly related to compressive strength and hence, inversely to wear

(Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid)
Tooling for Numerical Control

Tungsten Carbide

TUNGSTEN Carbide has 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
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

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

- **High Speed Steel** is used on:
  - Aluminum alloys
  - Other non ferrous alloys

- **Carbide** is used on:
  - High silicon aluminum
  - Steels
  - Stainless steels
  - Exotic metals

- **Ceramic inserts** are used on:
  - Hard steels
  - Exotic metals

*NOTE*

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

*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*


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° Diamond
  - 55° Diamond
  - Round
Figure 3-11: Relative edge strength and tendency for chipping and breaking of inserts with various shapes. Strength refers to that of the cutting edge shown by the included angles. (Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid)
Carbide Inserts and their Selection

- 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. 3-12,3-13)

- *GRADE* of insert describes the *HARDNESS* of the insert
Carbide Insert Grading System

- Each GRADE of Carbide is designated by an ANSI “C” number from $C_1$ to $C_8$
- 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 (Fig 3-12)
- The programmer is necessary to consult the individual manufacturers catalog to arrive the proper grade number

**Figure 3-12 Carbide insert grades**
Figure 3-13: Carbide insert identification system  (Photo KENNAMETAL)
Fundamentals of Machining
Two-Dimensional Cutting Process

Orthogonal Cutting

- A two-dimensional cutting process, also called **orthogonal cutting**:
  
  a) **Orthogonal cutting** with a well-defined shear plane, also known as the **Merchant Model**. Note that the tool shape, depth of cut, to, and the cutting speed, V, are all independent variables.

  b) **Orthogonal cutting** without a well-defined shear plane.

Two-Dimensional Cutting Process

Oblique Cutting

a) Cutting with an oblique tool
b) Top view, showing the inclination angle, \( i \).
c) Types of chips produced with different inclination angles.

• Terminology used in a turning operation on a lathe, where $f$ is the feed (in mm/rev or in./rev) and $d$ is the depth of cut.

• Note that **feed in turning is equivalent to the depth of cut in orthogonal cutting**, and the **depth of cut in turning** is equivalent to the width of cut in orthogonal cutting

Right-hand Cutting Tool for Turning

Figure 3-14: Lead or side-cutting edge angle is determined by the tool holder type. The lead angle can be (1) Neutral, (2) Negative or (3) Positive. (Photo SANVIK Coromant)
Figure 3-15: Side and top view of rake angles (1) Neutral, (2) Negative and (3) Positive (Photo ENCO)
Figure 3-16: The effect of the lead angle on the strength of the insert. Increasing the lead angle will greatly reduce tool breakage when roughing or cutting interrupted faces.

(Photos: Photo SANVIK Coromant)
Milling Operation

Conventional & Climb Milling

FIGURE 3-17: (a) Illustration showing the difference between conventional milling and climb milling, (b) Slab-milling operation, showing depth of cut, \( d \); feed per tooth, \( f \); chip depth of cut, \( tc \) and workpiece speed, (c) Schematic illustration of cutter travel distance, \( lc \), to reach full depth of cut

(Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid)
Face-milling operation showing (a) action of an insert in face milling; (b) climb milling; (c) conventional milling; (d) dimensions in face milling. The width of cut, w, is not necessarily the same as the cutter radius. (Manufacturing, Engineering & Technology, Fifth Edition, by Serope Kalpakjian and Steven R. Schmid)
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:

- End Mills
- Face Mills
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 3-18: Gear hob (Photo Sandvik Coromant) and thread hob (Photo Star SU, LLC)
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

**NOTE**

Two-flute cutters with deeper gullets are well suited for roughing operations.

Four-flute end mills are more rigid because of their thicker core.
Milling Cutters

End Mills

Figure 3-19: Single end, multiple flute end mill, standard length flutes
(Photo TTC Production s Inc.)

Figure 3-20: Solid carbide, two-flute, end mill
(Photo MARITool)
Milling Cutters

End Mills

- **Inserted cutters** are preferred for NC applications (Fig. 3-21).

- **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 be used on many types of workpiece materials by changing the inserts from one designed for Aluminum to one designed for Stainless Steel.
Figure 3-21: (a) Inserted carbide end mills, (b) and (c) 2 and 3 flute inserted end mills

(Photo Tool Korea Co)
Milling Cutters

End Mills

- **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**

- **Ball End Mills** using inserts (Fig. 3-22, 3-23)
End Mills

**Figure 3-22:** Ball nose end mills featured round inserts *(Photo SANDVIK Coromant)*

**Figure 3-23:** Ball nose end mills featuring inserts with two cutting edges *(Photo SANDVIK Coromant)*
Milling Cutters

End Mills

- **Inserted End Mill (Cyclo Mill)** designed by VALENITE GTE (Fig. 3-24)

- **Cyclo Mill** uses a series of round inserts staggered on a helical pattern

- **Cyclo Mill** can remove large amount of material at high speeds

- **Cyclo Mill** was developed for NC use

Figure 3-24: “Cyclo Mill” special multi-inserted milling cutter

(Photo GTE Valenite)
Milling Cutters

Face Mills

● **Face Mills** are designed to remove large amounts of material from the face of the workpiece (Fig. 3-25, 3-26)

● **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**

*The cost of HSS and Brazed Carbide limit their application to special situations*
Milling Cutters

Face Mills

Figure 3-25: A common type of Carbide inserted face mill (Photo Fiora Machinery)

Figure 3-26: Large inserted face mill – note number of inserts on cutter (Photo Sandvik Coromant)

Carbide Inserts

High number of inserts on the periphery of the cutter
Milling Cutters

Face Mills

- **Plunge and Profile Cutter**
  (Fig. 3-27)
  - It is designed to plunge into the material first and then begin the cutting path
  - The design is a cross between End Mill and Face Mill

Figure 3-27: Plunge and profile inserted milling cutter
(Photo Sandvik Coromant)
There are four basic hole operations that are performed on NC machinery:

- Drilling
- Reaming
- Boring
- Tapping
Drilling

- **Drills** are available in different styles for **different materials** (Fig. 3-29 shows a standard twist drill)

- Twist drills remain one of the most **common tools** for **making holes**

- Drills have a tendency to walk as they drill, resulting in a hole that is not truly straight

- **Center drills** (Fig. 3-30) are often used to predrill a pilot hole to help twist drill to start straight

- Drills also produce triangular-shaped holes
Hole Operations

Common Types of Drills

a) **Chisel-point drill**: The function of the pair of margins is to provide a bearing surface for the drill against walls of the hole as it penetrates into the workpiece. Drills with four margins (*double-margin*) are available for improved drill guidance and accuracy. Drills with chip-breaker features also are available.

b) **Crankshaft drills**: Have good centering ability, and because chips tend to break up easily, these drills are suitable for producing deep holes.
Tooling for Hole Operations

Various Types of Drills

Figure 3-28: Various types of drills and drilling operations
(Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid)
Tooling for Hole Operations

Figure 3-29: Tapered shank twist drill

Figure 3-30: Center drill
Drilling

- If the hole **tolerance is less than 0.003 inch** a secondary hole operation should be used to size the hole, such as **Boring** or **Reaming**

- **Large holes** are sometimes produced by **spade drills** (Fig. 3-31)

- The flat blades in spade drills allow good chip flow and economical replacement of the drill tip

**Figure 3-31 : Spade drill featuring inserts** (Photo ALLIED MAXCUT)
Tooling for Hole Operations

- **Drill point angle** must be considered when selecting a drill
- The harder the material to be cut the greater 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
- Special drills with Carbide inserts (Fig. 3-32)

**Figure 3-32:** (Courtesy Carboloy Inc., A Seco Tools Company)
# Speeds and Feeds in Drilling

<table>
<thead>
<tr>
<th>Workpiece Material</th>
<th>Surface Speed m/min</th>
<th>Feed, mm/rev (in./rev) Drill Diameter</th>
<th>Spindle speed (rpm) Drill Diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1.5 mm (0.060 in.)</td>
<td>12.5 mm (0.5 in.)</td>
</tr>
<tr>
<td>Aluminum alloys</td>
<td>30-120</td>
<td>0.025 (0.001)</td>
<td>0.30 (0.012)</td>
</tr>
<tr>
<td>Magnesium alloys</td>
<td>45-120</td>
<td>0.025 (0.001)</td>
<td>0.30 (0.012)</td>
</tr>
<tr>
<td>Copper alloys</td>
<td>15-60</td>
<td>0.025 (0.001)</td>
<td>0.25 (0.010)</td>
</tr>
<tr>
<td>Steels</td>
<td>20-30</td>
<td>0.025 (0.001)</td>
<td>0.30 (0.012)</td>
</tr>
<tr>
<td>Stainless steels</td>
<td>10-20</td>
<td>0.025 (0.001)</td>
<td>0.18 (0.007)</td>
</tr>
<tr>
<td>Titanium alloys</td>
<td>6-20</td>
<td>0.010 (0.0004)</td>
<td>0.15 (0.006)</td>
</tr>
<tr>
<td>Cast irons</td>
<td>20-60</td>
<td>0.025 (0.001)</td>
<td>0.30 (0.012)</td>
</tr>
<tr>
<td>Thermoplastics</td>
<td>30-60</td>
<td>0.025 (0.001)</td>
<td>0.13 (0.005)</td>
</tr>
<tr>
<td>Thermosets</td>
<td>20-60</td>
<td>0.025 (0.001)</td>
<td>0.10 (0.004)</td>
</tr>
</tbody>
</table>

*Note: As hole depth increases, speeds and feeds should be reduced. Selection of speeds and feeds also depends on the specific surface finish required.*

**TABLE 1: General recommendations for speeds and feeds in drilling**

*Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid*
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**.
Reaming

- Spiral fluted reamers (Fig. 3-34)
- Spiral fluted reamers produce better surface finishes than straight flutes
- Spiral fluted reamers are more difficult to re-sharpen than straight fluted
- Reamers are available in three basic tool materials:
  - HSS
  - Brazed carbide
  - Solid carbide
Tooling for Hole Operations

Reaming

Figure 3-33: Straight flute chucking reamer
(Photo DoALL Manufacturing)

Figure 3-34: Spiral flute chucking reamers
(Photo DoALL Manufacturing)
Figure 3-35: Terminology for a spiral fluted reamer

(Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid)
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
Tapping

Taping is used to produce internally threaded holes (Milling, Turning)

- They are available in different flute designs:
  - **Standard machine screw taps** (Fig. 3-36) 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** (Fig. 3-37) are used for soft, stringy material (e.g. Aluminum)
Tooling for Hole Operations

Tapping

Figure 3-236(upper): Machining screw tap

Figure 3-37(bottom): High spiral coated tap
Figure 3-38: (a) Terminology for a tap; (b) illustration of tapping of steel nuts in high production

(Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid)
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 spend time looking at tooling catalogues to become acquainted with current tooling developments

- Figures 3-39, 3-40 illustrate some of the current tooling ideas developed specifically for NC applications
Special Inserted Cutters

Figure 3-39: Special inserted tooling for use with NC. From left to right:

- an inserted milling cutter with interchangeable tooling extensions (*Iscar*)
- a machine tap in a tap holder with interchangeable tooling extensions (*Softsynchro® HD and MQL Modular System*)
- an inserted drill mounted in a holder with interchangeable extensions (*Sandvik Coromant*)
Figure 3-40: Indexable Inserted end mill suitable for multi-functional milling

(Photo BIG Kaiser Precision Tooling Inc.)
## Special Inserted Cutters

<table>
<thead>
<tr>
<th></th>
<th>CoroBore® 820</th>
<th>DuoBore™</th>
<th>Heavy duty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boring depth</td>
<td>4 x $D_5m$</td>
<td>4 x $D_5m$</td>
<td>6 x $D_c$</td>
</tr>
<tr>
<td>Hole tolerance</td>
<td>IT9</td>
<td>IT9</td>
<td>IT9</td>
</tr>
<tr>
<td>Material</td>
<td>PMK</td>
<td>PMK</td>
<td>PMK</td>
</tr>
<tr>
<td>Number of cutting edges</td>
<td>3</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Insert types</td>
<td>T-Max P CoroTurn® 107</td>
<td>T-Max P CoroTurn® 107</td>
<td>CoroTurn® 107</td>
</tr>
<tr>
<td>Power requirement</td>
<td>Medium, high</td>
<td>(Low), medium</td>
<td>(Low), medium</td>
</tr>
<tr>
<td>Lead angle</td>
<td>6° (15°), 0°, -5°</td>
<td>15°, 6°, 0°</td>
<td>15°, 0°</td>
</tr>
</tbody>
</table>

**Figure 3-41:** Boring tool selection – boring tool styles  *(Photo Sandvik coromant)*
### Special Inserted Cutters

#### Figure 3-42 Fine boring tool selection *(Photo Sandvik coromant)*

<table>
<thead>
<tr>
<th></th>
<th>Fine boring head</th>
<th>CoroBore® 825 – Fine boring tools</th>
<th>CoroBore® 825 – Damped fine boring tools</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boring depth</td>
<td>5 x Dc</td>
<td>4 x D5m</td>
<td>4 x D5m</td>
</tr>
<tr>
<td>Hole tolerance</td>
<td>IT6</td>
<td>IT6</td>
<td>IT6</td>
</tr>
<tr>
<td>Material</td>
<td>PMK NSH</td>
<td>PMK NSH</td>
<td>PMK NSH</td>
</tr>
<tr>
<td>Lead angle</td>
<td>0°, -1°, -2°</td>
<td>-2°</td>
<td>-2°</td>
</tr>
</tbody>
</table>
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 (sfpm)
- **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**
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:

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>CUTTING SPEED (sfm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool steel</td>
<td>50</td>
</tr>
<tr>
<td>Cast iron</td>
<td>60</td>
</tr>
<tr>
<td>Mild steel</td>
<td>100</td>
</tr>
<tr>
<td>Brass, soft bronze</td>
<td>200</td>
</tr>
<tr>
<td>Aluminum, magnesium</td>
<td>300</td>
</tr>
</tbody>
</table>
# Cutting Speed Data

## Cutting Speed for DRILLS

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>CUTTING SPEED (sfm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool steel</td>
<td>50</td>
</tr>
<tr>
<td>Cast iron</td>
<td>60</td>
</tr>
<tr>
<td>Mild steel</td>
<td>100</td>
</tr>
<tr>
<td>Brass, soft bronze</td>
<td>200</td>
</tr>
<tr>
<td>Aluminum, magnesium</td>
<td>300</td>
</tr>
</tbody>
</table>

## Cutting speeds for MILLING

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>CUTTING SPEED (sfm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool steel</td>
<td>40</td>
</tr>
<tr>
<td>Cast iron</td>
<td>50</td>
</tr>
<tr>
<td>Mild steel</td>
<td>80</td>
</tr>
<tr>
<td>Brass, soft bronze</td>
<td>160</td>
</tr>
<tr>
<td>Aluminum, magnesium</td>
<td>200</td>
</tr>
</tbody>
</table>
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:

\[
\text{rpm} = \frac{\text{CS} \times 12}{D \times \pi}
\]

Where:
- **CS** = cutting speed in surface feet per minute (sfm)
- **D** = diameter in inches of the tool or workpiece diameter for lathe
- **\(\pi\)** = 3.1416
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:

\[ rpm = \frac{CS \times 4}{D} \]
\[ CS = \frac{rpm \times D}{4} \]
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*
Feedrate

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

Feedrates are expressed in two ways:

1. inches per minute of spindle travel
2. 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
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’ 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 feedrates are the following:
  - Part geometry
  - Machine rigidity
  - Machine setup
- The actual CS and feedrate used during the run will ultimately be determined when the first piece is run during the job setup
Drilling Feedrates

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

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>CUTTING SPEED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool steel</td>
<td>50</td>
</tr>
<tr>
<td>Cast iron</td>
<td>60</td>
</tr>
<tr>
<td>Mild steel</td>
<td>100</td>
</tr>
<tr>
<td>Brass, soft bronze</td>
<td>200</td>
</tr>
<tr>
<td>Aluminum</td>
<td>250</td>
</tr>
<tr>
<td>Magnesium</td>
<td>300</td>
</tr>
</tbody>
</table>

Table 1 Cutting Speeds for common materials
Drilling feed rate is calculated by using the formula below:

\[ ipm = rpm \times ipr \]

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
# Recommended Drilling Feeds

<table>
<thead>
<tr>
<th>Drill Diameter (in.)</th>
<th>Drill Feed Rate (ipr)</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt; $\frac{1}{8}$</td>
<td>.001-.002</td>
</tr>
<tr>
<td>$\frac{1}{8}$ – $\frac{1}{4}$</td>
<td>.002-.004</td>
</tr>
<tr>
<td>$\frac{1}{4}$ - $\frac{1}{2}$</td>
<td>.004 - .007</td>
</tr>
<tr>
<td>$\frac{1}{2}$ - 1</td>
<td>.007 - .015</td>
</tr>
<tr>
<td>&gt; 1</td>
<td>.015-.025</td>
</tr>
</tbody>
</table>

Table 2: Drilling Feeds
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:

  \[ ipm = rpm \times ipr \]

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

  \[ rpm = \frac{CS \times 4}{D} \]

  \[ rpm = \frac{250 \times 4}{0.375} \]

  \[ rpm = 2666 \]

- **Step 3:** Select Drill diameter: \( \frac{1}{4} - \frac{1}{2} \), Drill feed from table 1: .004 - .007

  \[ ipm = 2666 \times 0.005 \]

  \[ ipm = 13.33 \]
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

\[ F = R \times T \times rpm \]

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
**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 eep THEN the feedrate should be adjusted to bring it in to acceptable limits.
Milling Feedrates

- **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 Feedrates

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

\[ f = \sqrt{\frac{D}{W}} \times CT \]

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 feedrate formula and a new Feedrate is then 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:

\[
rpm = \frac{CS \times 12}{D \times \pi}
\]

• **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 (\(\pi\))

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 Feeds**

**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

  \[ F = 2183 \times 4 \times 0.004 \quad F = 34,91 \text{ inches/s/min} \]

- **Step 5:** Calculate the chip thickness to insure that the inserts will not break down prematurely: It is assumed \( \text{Width of the Cut} = 1.000 \text{ inch wide} \)

  \[ CT = \sqrt{\frac{W}{D}} \times R \quad CT = \sqrt{\frac{1.000}{1.750}} \times 0.004 \quad CT = 0.00302 \]

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

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 feedrate formula and recalculate Feedrate:
  \[ F = 2183 \times 4 \times 0.010 \]
  \[ F = 87.32 \text{ inches s/min} \]

**Conclusion:**

- The 2813 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
To calculate the feed rate for a mill cut the following formula can also be used:

\[ F_m = f_t \times n_t \times N \]

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)

### Recommended Tool Feed

<table>
<thead>
<tr>
<th>Material</th>
<th>Face Mill</th>
<th>Side Mill</th>
<th>End Mill</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magnesium</td>
<td>.005-.020</td>
<td>.004-.010</td>
<td>.005-.010</td>
</tr>
<tr>
<td>Aluminum</td>
<td>.005-.020</td>
<td>.004-.010</td>
<td>.005-.010</td>
</tr>
<tr>
<td>Brass and Bronze</td>
<td>.004-.020</td>
<td>.004-.010</td>
<td>.005-.010</td>
</tr>
<tr>
<td>Copper</td>
<td>.004-.010</td>
<td>.004-.007</td>
<td>.004-.008</td>
</tr>
<tr>
<td>Cast Iron (Soft)</td>
<td>.004-.016</td>
<td>.004-.009</td>
<td>.004-.008</td>
</tr>
<tr>
<td>Cast Iron (Hard)</td>
<td>.004-.010</td>
<td>.002-.006</td>
<td>.002-.006</td>
</tr>
<tr>
<td>Milt Steel</td>
<td>.004-.010</td>
<td>.002-.007</td>
<td>.002-.010</td>
</tr>
<tr>
<td>Alloy Steel (Hard)</td>
<td>.004-.010</td>
<td>.002-.007</td>
<td>.002-.006</td>
</tr>
<tr>
<td>Tool Steel</td>
<td>.004-.008</td>
<td>.002-.006</td>
<td>.002-.006</td>
</tr>
<tr>
<td>Stainless Steel</td>
<td>.004-.008</td>
<td>.002-.006</td>
<td>.002-.006</td>
</tr>
<tr>
<td>Titanium</td>
<td>.004-.008</td>
<td>.002-.006</td>
<td>.002-.006</td>
</tr>
<tr>
<td>High Manganese Steel</td>
<td>.004-.008</td>
<td>.002-.006</td>
<td>.002-.006</td>
</tr>
</tbody>
</table>

**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>
<thead>
<tr>
<th>Material</th>
<th>Face Mill</th>
<th>Side Mill</th>
<th>End Mill</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magnesium</td>
<td>.005-.020</td>
<td>.004-.010</td>
<td>.005-.010</td>
</tr>
<tr>
<td>Aluminum</td>
<td>.005-.020</td>
<td>.004-.010</td>
<td>.005-.010</td>
</tr>
<tr>
<td>Brass and Bronze</td>
<td>.004-.020</td>
<td>.004-.010</td>
<td>.005-.010</td>
</tr>
<tr>
<td>Copper</td>
<td>.004-.010</td>
<td>.004-.007</td>
<td>.004-.008</td>
</tr>
</tbody>
</table>

Table 3 : Tool Feed

\[ f_t = 0.005 \text{ in.} / \text{tooth} \]
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:

  \[
  N = rpm = \frac{CS \times 4}{D} \rightarrow N = \frac{250 \times 4}{0.5} \rightarrow N = 2000rpm
  \]
**Step 4:** Calculation of the feed rate of the milling cutter using the formula below:

\[ F_m = f_t \times n_t \times N \]

\[ F_m = 0.005 \times 2 \times 2000 \]

\[ F_m = 20 \text{ in/min.} \]
Calculate the Feed Rate for Face Milling Aluminum with a 4 flute, \( \frac{3}{4} \) inch HSS end mill

- **Step 1:** Selection of \( f_t \) (Feed in inches / tooth) from table 3
  \[ f_t = 0.005 \text{ in.} / \text{tooth} \]

- **Step 2:** Calculation of \( n_t \) (number of teeth on the tool):
  \[ n_t = 4 \]

- **Step 3:** Calculation of Spindle Speed:
  \[ N = rpm = \frac{CS \times 4}{D} \]
  \[ N = \frac{250 \times 4}{0.75} \]
  \[ N = 1333.33 rpm \]
**Step 4:** Calculation of the feed rate of the milling cutter using the formula below:

\[ F_m = f_t \times n_t \times N \]

\[ F_m = 0.005 \times 4 \times 1333.33 \]

\[ F_m = 26.67 \text{ in/min.} \]
● **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.

● **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 chapter

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


References

17. http://www.use-enco.com/


30. Γ. Χρυσολούρης, «Συστήματα Παραγωγής Θεωρία και Πράξη» Μέρος Ι και ΙΙ, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2001,
References

31. Γ. Χρυσολούρης, Δ. Μούρτζης, Κ. Τσίρμπας, Σ. Καραγιάννης, “Ορθογωνική Κοπή”, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2000

32. Γ. Χρυσολούρης, Δ. Μούρτζης, και άλλοι, “Εργαστήρια Μηχανουργικής Τεχνολογίας Ι και ΙΙ”», Εκπαιδευτικές Σημειώσεις για το εργαστήριο του αντιστοίχου μαθήματος, Πανεπιστήμιο Πατρών, 2008 (4η Έκδοση)

33. Δ. Μούρτζης, “Αριθμητικός Έλεγχος Εργαλειομηχανών” Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2011 (3η Έκδοση)

34. Πετρόπουλου Π.Γ., «Μηχανουργική Τεχνολογία – ΙΙ. Τεχνολογία κατεργασιών κοπής των μετάλλων», 1998, Εκδόσεις Ζήτη

35. Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,Κ.Σαλωνίτης