COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece
Chapter 3:
Process Planning and Tool Selection
Objectives of Chapter 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 feed rates
**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?
The collaboration between CAM, CAPP and CAD systems (Ming et al. 2008)
Role of CAM in a typical product lifecycle (Zeid 1991)
Manufacturing information flow in the state-of-the-art CAD/CAM/CNC chain (Newman et al. 2008)
Process Planning

CAD/CAM links and flow of a computer-aided system (Rehg and Kraebber 2005)
## Evolution of turning machines

<table>
<thead>
<tr>
<th>Year</th>
<th>Round parts</th>
<th>Square parts</th>
<th>Complex shape parts</th>
</tr>
</thead>
<tbody>
<tr>
<td>'90</td>
<td><img src="image1" alt="Round part" /></td>
<td><img src="image2" alt="Square part" /></td>
<td><img src="image3" alt="Complex part" /></td>
</tr>
<tr>
<td>'91</td>
<td><img src="image4" alt="Round part" /></td>
<td><img src="image5" alt="Square part" /></td>
<td><img src="image6" alt="Complex part" /></td>
</tr>
<tr>
<td>'92</td>
<td><img src="image7" alt="Round part" /></td>
<td><img src="image8" alt="Square part" /></td>
<td><img src="image9" alt="Complex part" /></td>
</tr>
<tr>
<td>'93</td>
<td><img src="image10" alt="Round part" /></td>
<td><img src="image11" alt="Square part" /></td>
<td><img src="image12" alt="Complex part" /></td>
</tr>
<tr>
<td>'94</td>
<td><img src="image13" alt="Round part" /></td>
<td><img src="image14" alt="Square part" /></td>
<td><img src="image15" alt="Complex part" /></td>
</tr>
<tr>
<td>'95</td>
<td><img src="image16" alt="Round part" /></td>
<td><img src="image17" alt="Square part" /></td>
<td><img src="image18" alt="Complex part" /></td>
</tr>
<tr>
<td>'96</td>
<td><img src="image19" alt="Round part" /></td>
<td><img src="image20" alt="Square part" /></td>
<td><img src="image21" alt="Complex part" /></td>
</tr>
<tr>
<td>'97</td>
<td><img src="image22" alt="Round part" /></td>
<td><img src="image23" alt="Square part" /></td>
<td><img src="image24" alt="Complex part" /></td>
</tr>
<tr>
<td>'98</td>
<td><img src="image25" alt="Round part" /></td>
<td><img src="image26" alt="Square part" /></td>
<td><img src="image27" alt="Complex part" /></td>
</tr>
<tr>
<td>'99</td>
<td><img src="image28" alt="Round part" /></td>
<td><img src="image29" alt="Square part" /></td>
<td><img src="image30" alt="Complex part" /></td>
</tr>
<tr>
<td>'00</td>
<td><img src="image31" alt="Round part" /></td>
<td><img src="image32" alt="Square part" /></td>
<td><img src="image33" alt="Complex part" /></td>
</tr>
<tr>
<td>'01</td>
<td><img src="image34" alt="Round part" /></td>
<td><img src="image35" alt="Square part" /></td>
<td><img src="image36" alt="Complex part" /></td>
</tr>
<tr>
<td>'02</td>
<td><img src="image37" alt="Round part" /></td>
<td><img src="image38" alt="Square part" /></td>
<td><img src="image39" alt="Complex part" /></td>
</tr>
<tr>
<td>'03</td>
<td><img src="image40" alt="Round part" /></td>
<td><img src="image41" alt="Square part" /></td>
<td><img src="image42" alt="Complex part" /></td>
</tr>
<tr>
<td>'04</td>
<td><img src="image43" alt="Round part" /></td>
<td><img src="image44" alt="Square part" /></td>
<td><img src="image45" alt="Complex part" /></td>
</tr>
</tbody>
</table>

*Laboratory for Manufacturing Systems and Automation*  
Dr. Dimitris Mourtzis
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.
Three possible types of machine structure

Vertical type  Horizontal type  Double column type
Sample configuration of recent multi-functional machine tool

- 4 spindles  9 axes
- 2-axis Lathe + 5-axis Machining center

Headstock 1

Headstock 2

Tool spindle (Upper turret)

Lower turret
An example of combined multi-axis machine tool (Source: Mori Seiki)

- No.1 Headstock
- No.1 Tool post
- No.2 Headstock
- No.2 Tool post
The optimum angle is indexed and fixed with two rotating axes according to the shape to be machined. Normal orthogonal 3-axis machining is performed at that location.
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?
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?
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.

**FIGURE 3-1: NC Setup Sheet for a 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>—</td>
<td>3.0 DIA. INSERTED FACE MILL</td>
</tr>
<tr>
<td></td>
<td></td>
<td>W. .015 R GRADE 883 INSERTS</td>
</tr>
<tr>
<td>2</td>
<td>D12</td>
<td>.500 DIA. 4-FLUTE SOLID CARB. END MILL</td>
</tr>
<tr>
<td>3</td>
<td>—</td>
<td>NO. 4 × 90° C DRILL</td>
</tr>
<tr>
<td>4</td>
<td>—</td>
<td>1/4 DRILL (.250 DIA.)</td>
</tr>
<tr>
<td>5</td>
<td>—</td>
<td>.262 DIA. BORING BAR</td>
</tr>
</tbody>
</table>

**TAPE NUMBER: 1053**

**FIXTURE: 6 IN. MILL VISE**

**TABLE LAYOUT:**

**NOTES:** DRILL POINT ANGLES TO BE 118° INCL.

**DRWN: WSS**

**PROG: WSS**

**DATE: 1-10-89**

**B/P REV: C**

**MACHINE: VERTICAL 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 3-2 NC Setup Sheet for a CNC lathe
Process Planning

- **Turning**
  - External turning
  - Drilling
  - Boring
  - Internal threading

- **Milling**
  - External milling
  - Drilling
  - Ball endmilling
  - Inclined drilling

- **Cutting with 2nd Spindle**
  - Parting - off
  - Face turning
  - Inclined drilling
  - Drilling and tapping
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 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

- 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 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

Laboratory for Manufacturing Systems and Automation

Dr. Dimitris Mourtzis
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 aluminum
  - 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*
There are four basic hole operations that are performed on NC machinery:

- Drilling
- Reaming
- Boring
- Tapping
Tooling for Hole Operations

Drilling

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

FIGURE 3-4 Center drill
Tooling for Hole Operations

Drilling

- If the hole tolerance is closer 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. 3-5).
- The flat blades in spade drills allow good chip flow and economical replacement of the drill tip.

FIGURE 3-5 Spade drill
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. 3-6)
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 (Fig. 3-7).

**FIGURE 3-7** Straight flute chucking reamer
(Photo courtesy of DoALL Manufacturing)
Tooling for Hole Operations

Reaming

- Spiral fluted reamers (Fig. 3-8)
- 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

FIGURE 3-8 Spiral flute chucking reamers
(Photo courtesy of DoALL Manufacturing)
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**
Tapping

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

- 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** (Fig. 3-10) are used for soft, stringy material (e.g. Aluminum).

FIGURE 3-9 Machining screw tap

FIGURE 3-10 High spiral coated tap
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

![Thread Hob Image](Photo courtesy of GTE Valenite)

**FIGURE 3-11 Thread hob (Photo courtesy of GTE Valenite)**
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

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

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

**FIGURE 3-12** Single end, multiple flute end mill, standard length flutes (Photo courtesy of Sharpaloy Division, Precision Industries Inc.)

**FIGURE 3-13** Solid carbide, two-flute, end mill (Photo courtesy of DoALL Manufacturing)
End Mills

- **Inserted cutters** are preferred for NC applications (Fig. 3-14, 3-15)
- 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
Milling Cutters

End Mills

- **Ball End Mills** using inserts (Fig. 3-16, 3-17)
- **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

FIGURE 3-16 Ball nose end mills featured round inserts (Photo courtesy of GTE Valenite)

FIGURE 3-17 Ball nose end mills featuring triangular inserts (Photo courtesy of GTE Valenite)
Milling Cutters

End Mills

- **Inserted End Mill (Cyclo Mill)** designed by VALENITE GTE (Fig. 3-18)
- **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 3-18 “Cyclo Mill” special multi-inserted milling cutter (Photo courtesy of GTE Valenite)
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*

![Figure 3-19](image-url) A common type of Carbide inserted face mill (Photo courtesy of GTE Valenite)
Milling Cutters

Face Mills

- **Large Diameter Face Mill** (Fig. 3-20)
- **Large number of inserts used**
- **Cyclo Mill** was developed for NC use

**FIGURE 3-20** Large inserted face mill – note number of inserts on cutter (Photo courtesy of GTE Valenite)
Milling Cutters

Face Mills

- Plunge and Profile Cutter (Fig. 3-21)
- It is designed to plunge into the material first and then begging the cutting path
- The design is a cross between End Mill and Face Mill

FIGURE 3-21 Plunge and profile inserted milling cutter (Photo courtesy of GTE Valenite)
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 catalogs to become acquainted with current tooling developments.

- Figures 3-22 through 3-24 illustrate some of the current tooling ideas developed *specifically for NC applications*.
Special Inserted Cutters

FIGURE 3-23 (Photo courtesy of GTE Valenite)
Special inserted tooling for use with NC. From top to bottom:

- an inserted milling cutter with interchangeable tooling extensions
- a machine tap in a tap holder with interchangeable tooling extensions
- an inserted drill mounted in a holder with interchangeable extensions

FIGURE 3-22
Special small diameter inserted end mill (Photo courtesy of GTE Valenite)
Special Inserted Cutters

An NC tooling system featuring:
- tool adapters
- interchangeable extensions
- tool bodies
- boring heads
- arbors
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
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

Identification System

FIGURE 3-25 Carbide insert identification system (Courtesy of Carboloy Inc., A Seco Tools Company)
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 (Fig 3-26)
- The programmer is necessary to consult the individual manufacturers catalog to arrive the proper grade number

<table>
<thead>
<tr>
<th>INSERT GRADE APPLICATION CHART</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cast iron and nonferrous materials</td>
</tr>
<tr>
<td>C-1: Roughing</td>
</tr>
<tr>
<td>C-2: General Purpose</td>
</tr>
<tr>
<td>C-3: Finishing</td>
</tr>
<tr>
<td>C-4: Precision Finishing</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MANUFACTURER’S GRADE DESIGNATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSI</td>
</tr>
<tr>
<td>Class</td>
</tr>
<tr>
<td>C-8</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>C-7</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>C-6</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>C-5</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>C-4</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>C-3</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>C-2</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>C-1</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

Note: Most manufacturers produce more than one grade per insert class. Consult the manufacturer’s catalog for a complete listing.

**FIGURE 3-26** Carbide insert grades
Special Inserted Cutters

FIGURE 3-27 Toolholder and boring bar sections – boring bar styles (Courtesy of Carboloy Inc., A Seco Tools Company)

FIGURE 3-28 Tool holder and boring bar selection – tool holder styles (Courtesy of Carboloy Inc., A Seco Tools Company)

Laboratory for Manufacturing Systems and Automation

Dr. Dimitris Mourtzis
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
A Processing Example

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

- Face the 1.000 and .25 dimensions using a 3\(\frac{1}{4}\) carbide inserted face mill
- 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
- Drill the .188 diameter holes using a \(\frac{3}{16}\) drill. Since drills almost always drill .001 or more oversize, the hole will be comfortably within tolerance
- Drill the .250 diameter hole using a \(\frac{1}{4}\) drill
- 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
- Using the same end mill, mill the 1.500 diameter bore
### A Processing Example

#### FIGURE 3-30 Manufacturing process for part shown in Figure 3-29

<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>
</tbody>
</table>
| 020              | manual lathes  | Chuck on 3.250 as cast dia.  
|                  |                | Turn $4.000 \pm .010$ b/p dim to $4.000 \pm .001$ dia. 
|                  |                | (tooling dimension).  
|                  |                | Face $0.38$ b/p dim.  |
| 030              | vert. mach. center | Locate parts in fixture NCF-000-100  
|                  |                | Drill $0.188 + .006 - .001$ dia. thru 6 plcs.  
|                  |                | Drill $0.250 + .006 - .001$ dia. thru 4 plcs.  
|                  |                | Bore $1.500 \pm .010$ dia. thru 1 plc.  
|                  |                | Mill the $3.000 \pm .010$ dia., hole the $1.000$ and $0.25$ dim.  |
| 040              | burr           | Deburr parts as required.  |
| 050              | insp           | Inspect parts for b/p conformance.  |
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 3-31 Fixture concept**
A Processing Example

<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 × 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>

TAPE NUMBER: 1000
FIXTURE: NCF-000-100

TABLE LAYOUT:

![Diagram of NC setup sheet for CNC machining center]

FIGURE 3-32 NC setup sheet for CNC machining center
Speed and Feeds

**Insert Shape**

**Figure 3-2** The shape of the insert will have a great effect on the strength of the tool. Select the largest included angle that will cut the part.

**Figure 3-3** This figure is a diagram of a typical triangular insert.
Speed and Feeds

**Figure 3-4** Side view of back rake angles.

**Figure 3-5** Lead or side-cutting edge angle is determined by the tool holder type. The lead angle can be positive, neutral, or negative.
FIGURE 3-6 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 surfaces.
Speed and Feeds

**FIGURE 3-28** Tool feed.

**FIGURE 3-29** Tool feed, width of cut and chip thickness.
Speed and Feeds

**FIGURE 3-30** Climb milling.

**FIGURE 3-31** Conventional milling.
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:**

<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>

Laboratory for Manufacturing Systems and Automation

Dr. Dimitris Mourtzis
# Speed and Feeds

## 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>
Speed and Feeds

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
- \( \pi = 3.1416 \)
Speed and Feeds

Cutting Speed

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

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

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

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

\[
CS = \frac{\text{rpm} \times D}{4}
\]
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 when the first piece is run during the job setup**
Drilling Feedrates

- Drilling feedrates 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, magnesium</td>
<td>300</td>
</tr>
</tbody>
</table>

\[ \text{ipm} = \text{rpm} \times \text{iqr} \]

Where:
- \( \text{ipm} \) = the required feedrate expressed in inches per minute
- \( \text{rpm} \) = the programmed spindle speed in revolutions per minute
- \( \text{iqr} \) = the drill feedrate to be used expressed in inches per revolution
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:

\[ F = R \times T \times \text{rpm} \]

Where:
- \( F \) = the milling feedrate expressed in inches per minute
- \( R \) = the chip load per tooth
- \( T \) = the number of teeth on the cutter
- \( \text{rpm} \) = 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 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:

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

Where:
- CT = the chip thickness
- W = the width of the cut
- D = the diameter of the cutter
- R = the feed per tooth
Speed and Feeds

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 calculated
Speed and Feeds

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:

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

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

  \[
  \text{rpm} = \frac{1,000 \times 12}{1.75 \times 3.1416} = 2,183
  \]
Speed and Feeds

Speed and Feed Example

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

\[
F = R \times T \times \text{rpm}
\]

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

\[
F = 2.183 \times 4 \times 0.004 \quad F = 34.91 \text{ inches/min}
\]

- **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

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

- **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

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

\[
f = \sqrt{\frac{1.75}{1.000}} \times 0.008 = 0.010
\]

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

\[
F = 2138 \times 4 \times 0.010 = 87.32 \text{ inches/min}
\]

**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
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:
1. Determine the machine.
2. Determine the workholding.
3. Determine the machining strategy.
4. 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 chapter

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