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

Objectives of Chapter 3

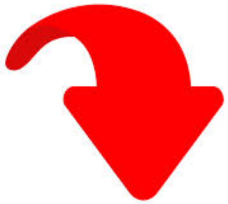
- List the steps involved in **process planning**
- List the **factors** that influence the selection of an NC machine, work-holding devices, and tooling
- Describe the **types of tools** available for **hole operations**
- Describe the **types of tools** available for **milling operations**
- Determine 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

(Chryssolouris G., «Manufacturing Systems: Theory and Practice», 2nd Edition, 2006)

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

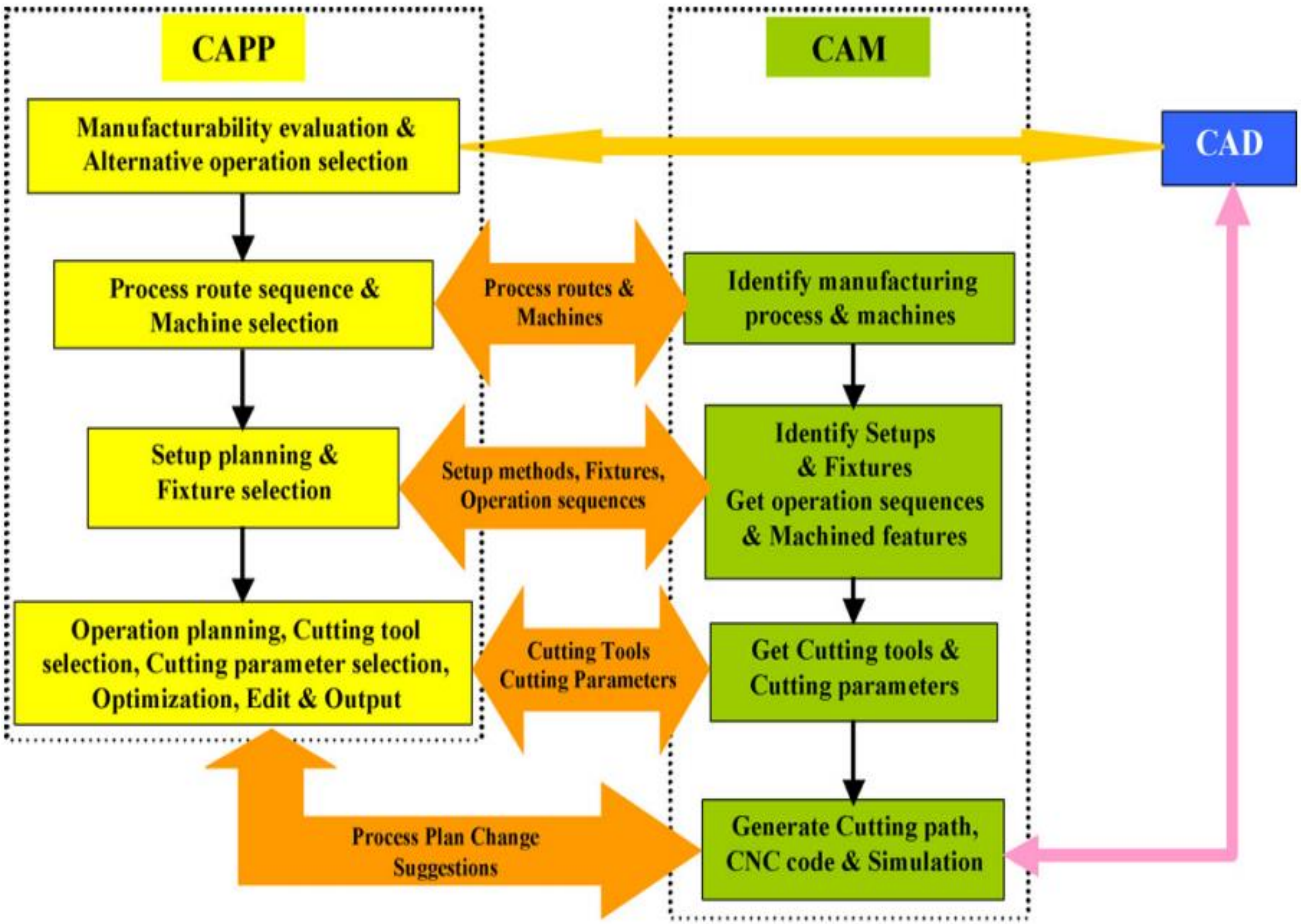


Figure 3-8:The collaboration between CAM, CAPP and CAD systems (Ming et al. 2008)

Process Planning

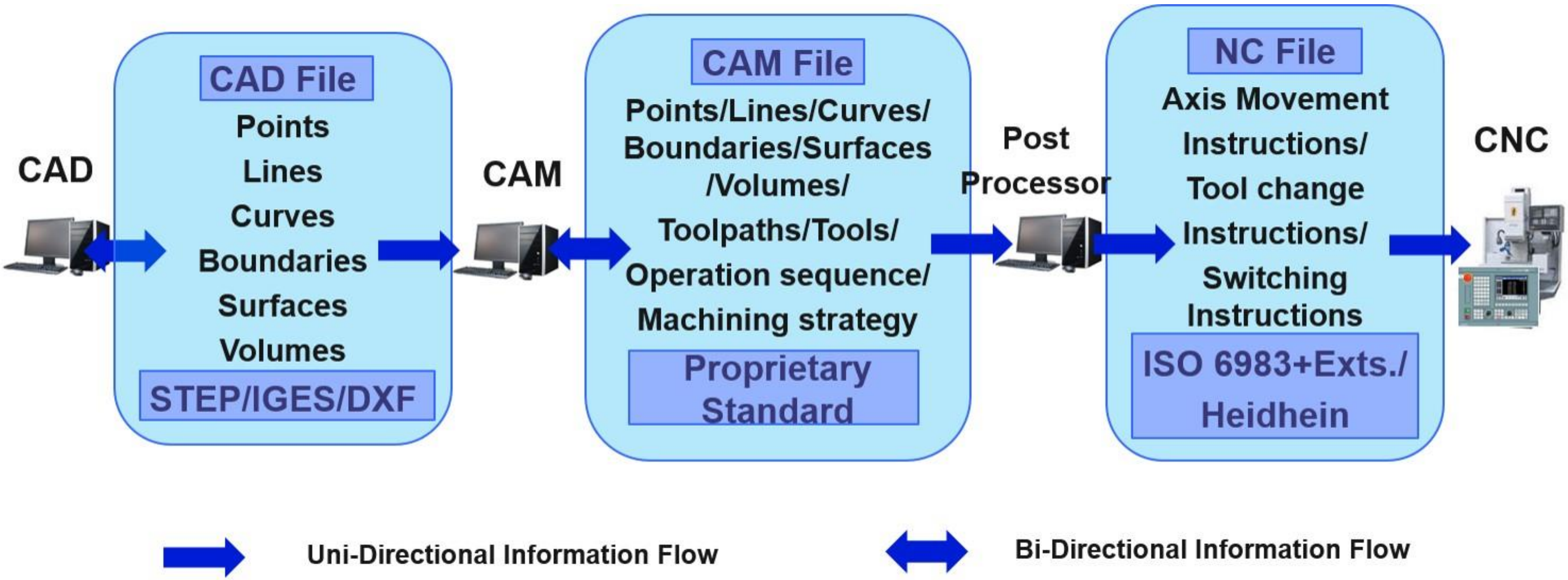


Figure 3-9: Manufacturing information flow in the state-of-the-art CAD/CAM/CNC chain (Newman et al. 2008)

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

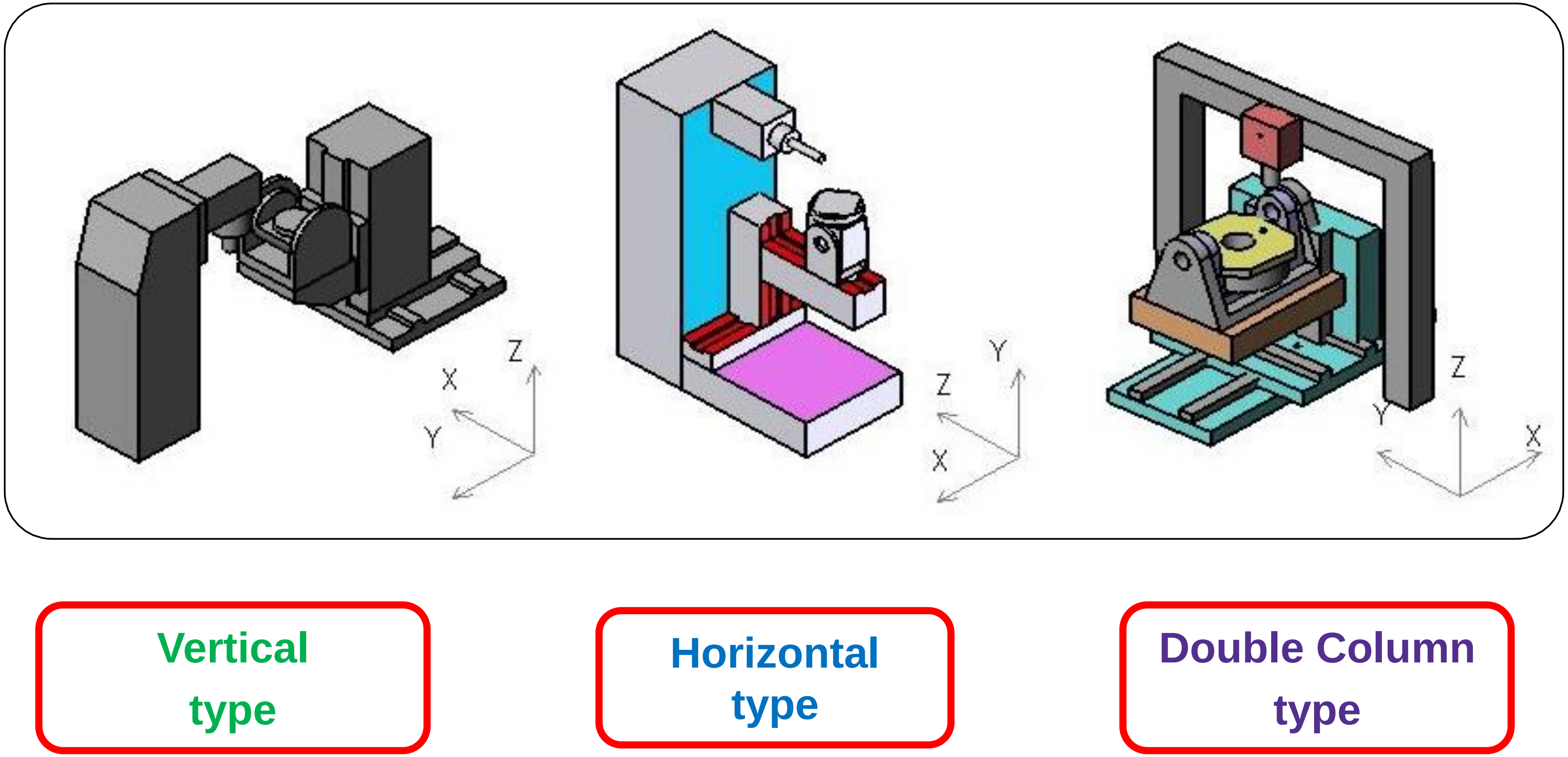


Figure 3-1: Three possible configurations of machine structure (T. Moriwaki,2008)

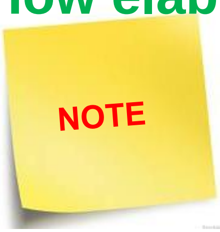
Fixturing: Decision on how the **workpiece should be held**

- 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.

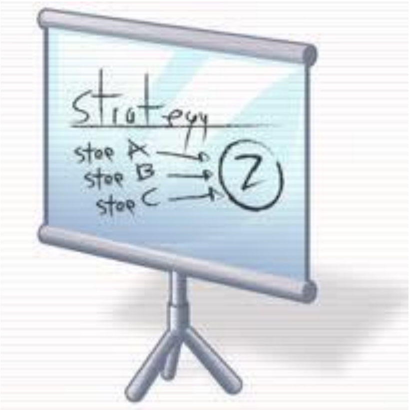
- What will make the best **quality** part?

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?



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?

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?

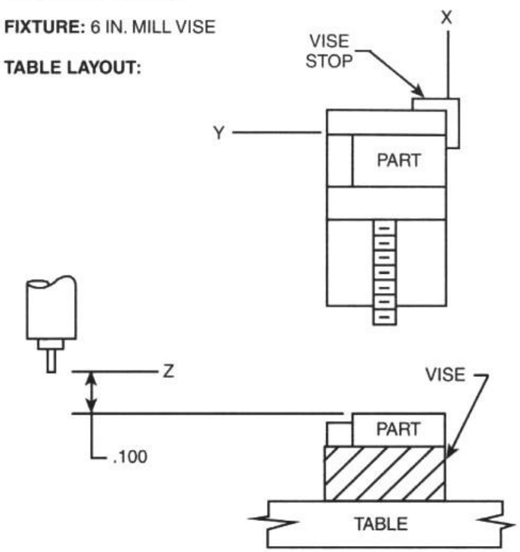
NC Setup Sheet

STA. NO.	CRO REG.	TOOL DESCRIPTION
1	—	3.0 DIA. INSERTED FACE MILL
		W/ .015 R GRADE 883 INSERTS
2	D12	.500 DIA. 4-FLUTE SOLID CARB. END MILL
3	—	NO. 4 × 90° C'DRILL
4	—	1/4 DRILL (.250 DIA.)
5	—	.262 DIA. BORING BAR

TAPE NUMBER: 1053

FIXTURE: 6 IN. MILL VISE

TABLE LAYOUT:



NOTES: DRILL POINT ANGLES TO BE 118° INCL.

DRWN: WSS	**NC SETUP SHEET FOR:**
PROG: WSS	**MACHINE:** VERTICAL MACHINING CENTER
DATE: 1-10-89	
B/P REV: C	

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

- 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**

NC Setup Sheet

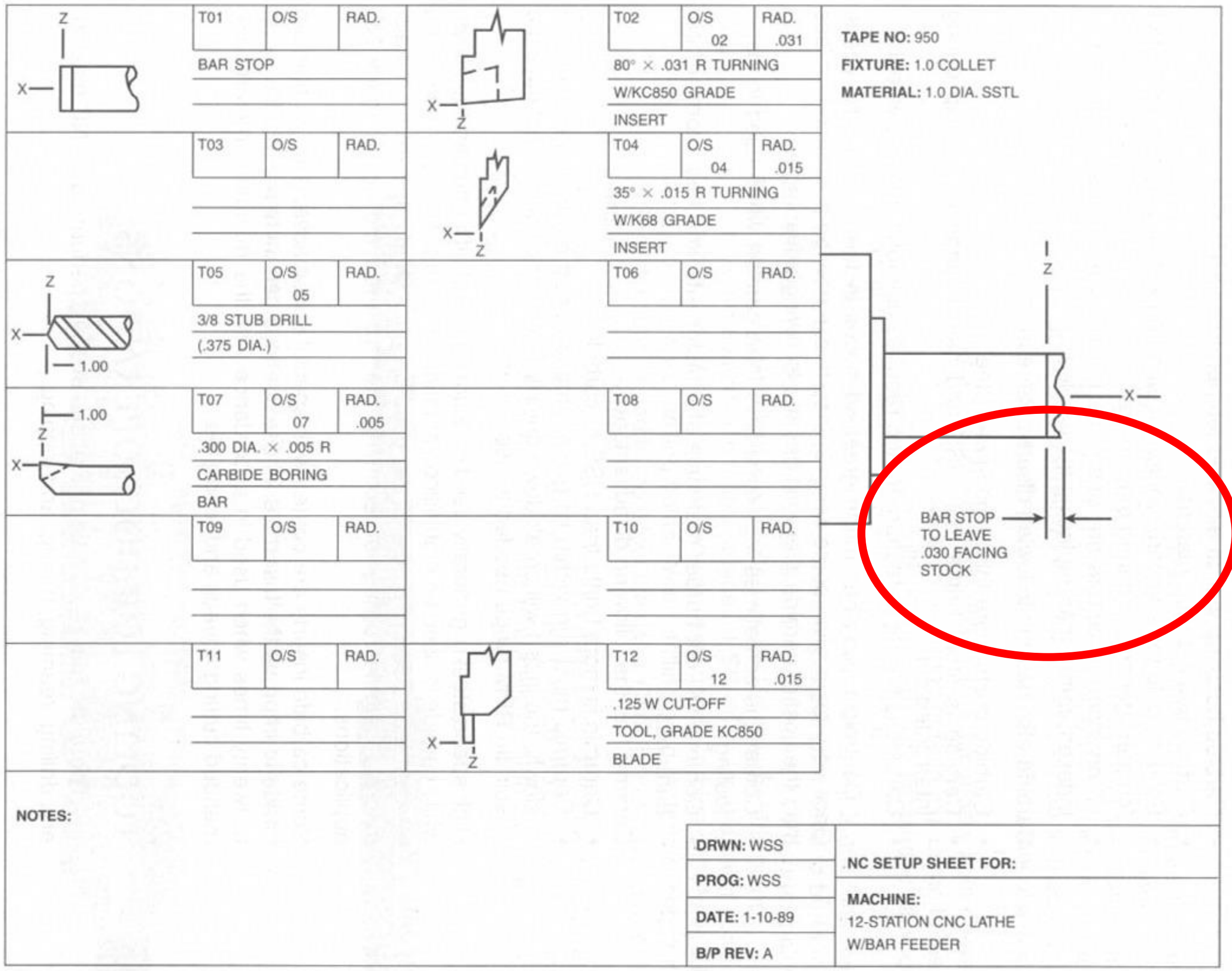


Figure 3-3 NC Setup Sheet for a CNC lathe

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

A Processing Example

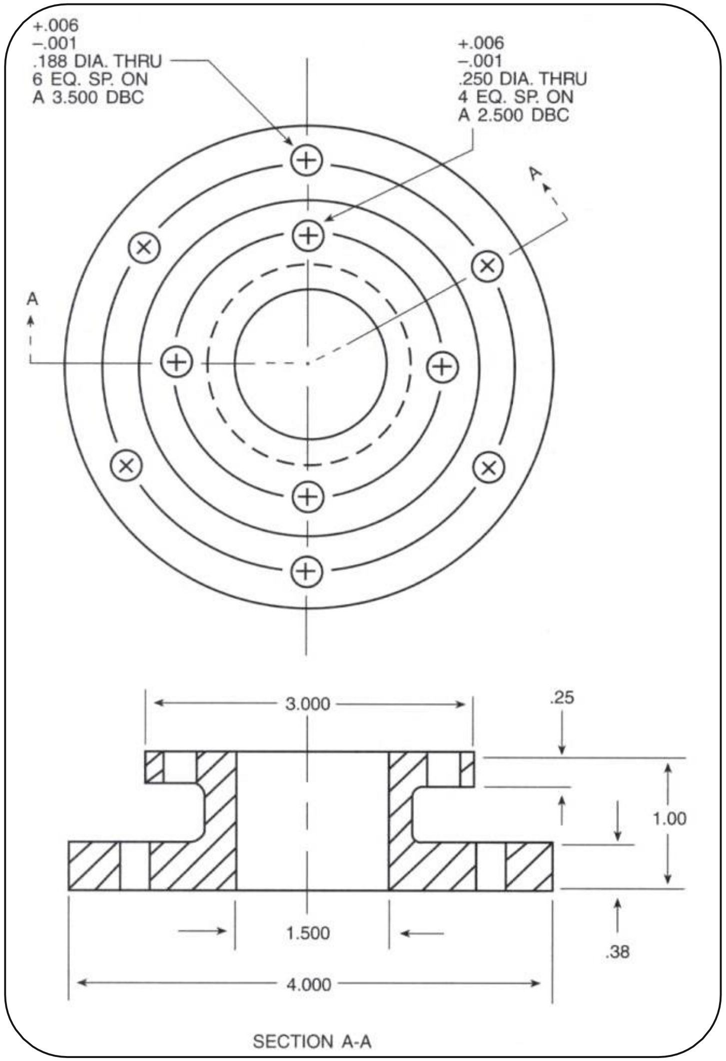


Figure 3-4: Part drawing

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:

1. Face the 1.000 and .25 dimensions using a $3^1/_4$ 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 $^3/_{16}$ 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 $^1/_4$ drill
5. Mill the 3.000 diameter using a $1^1/_4$ 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

Part Number: Adapter
Run Quantity: 200

Job Number: 000-000-001
Material: Alum. Casting.

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

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

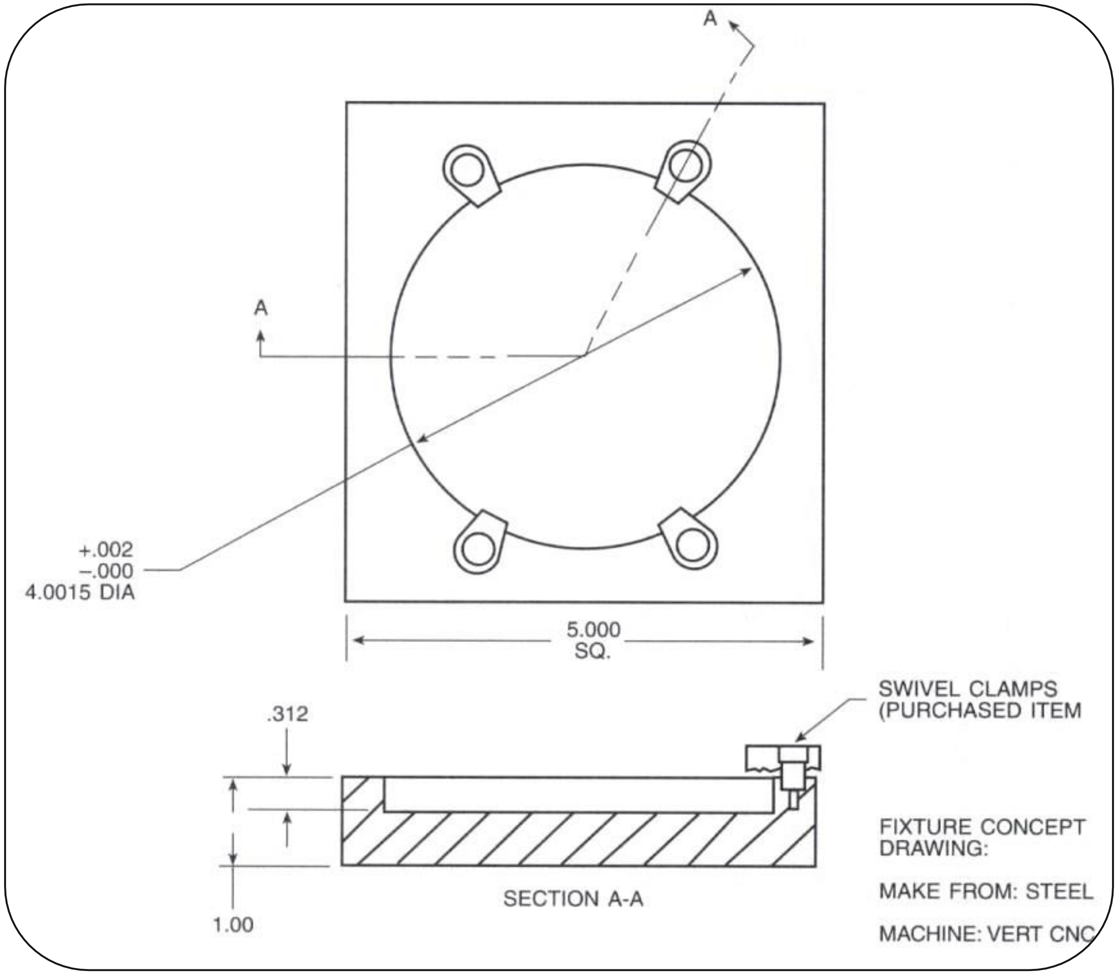


Figure 3-6 Fixture concept

- 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

A Processing Example

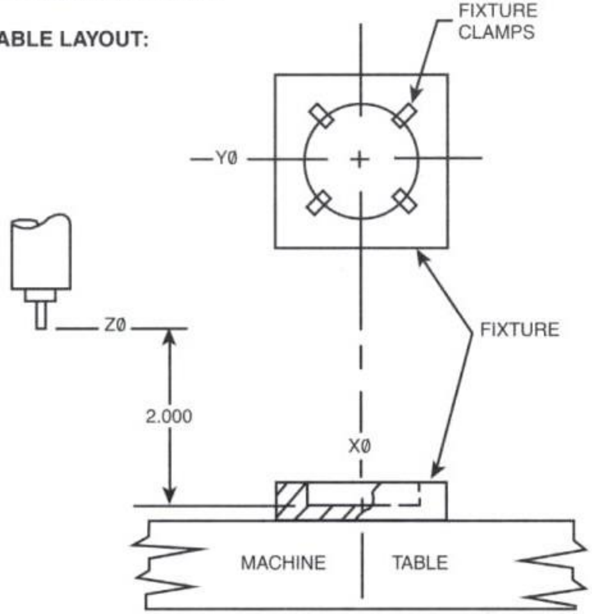
STA. NO.	CRO REG.	TOOL DESCRIPTION
1	D11	3 1/4 INSERTED CARBIDE FACE MILL
2		NO. 4 × 90° C'DRILL
3		3/16 DRILL (.1875 DIA.)
4		1/4 DRILL (.250 DIA.)
5	D15	1 1/4 INSERTED CARBIDE HELICAL END MILL

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

TAPE NUMBER: 1000

FIXTURE: NCF-000-100

TABLE LAYOUT:

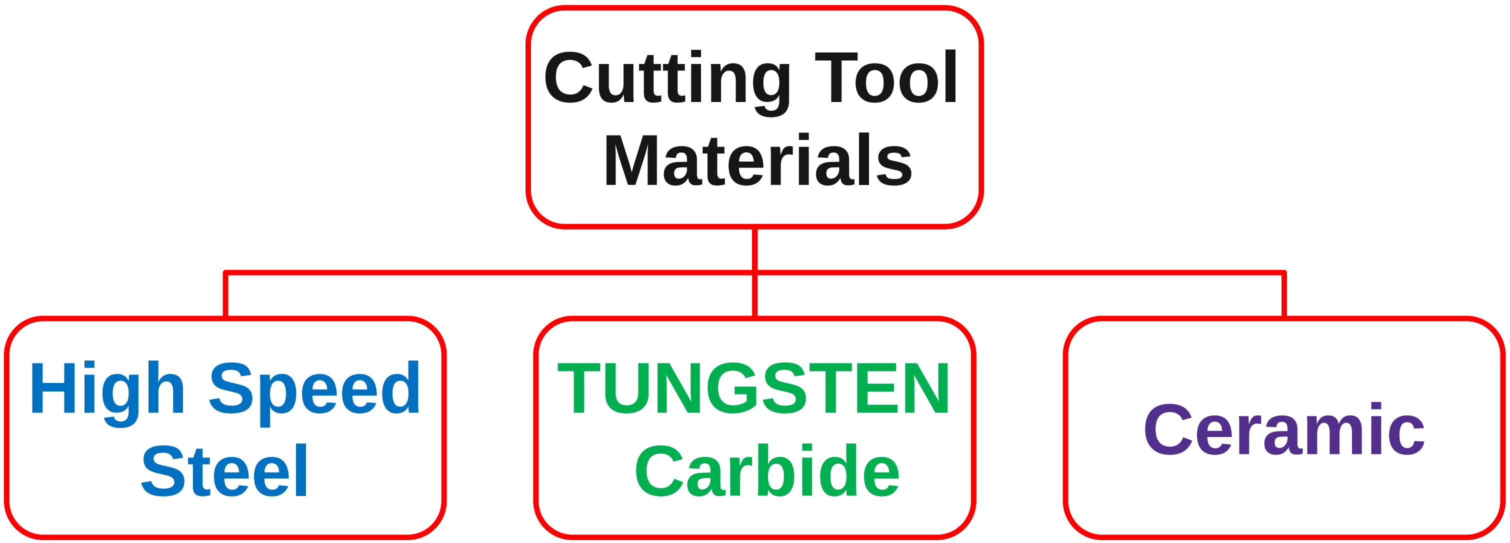


DRWN: WSS	**SETUP SHEET FOR CNC MACHINING CENTER**
PROG: WSS	**MACHINE:** UNIVERSAL VERT. MACH. CENTER
DATE: 3-4-89	
B/P REV: A	**OPER. NO:** 030

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

Cutting Tool Materials

Cutting Tools are available in three basic types:



High Speed Steel (HSS)

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

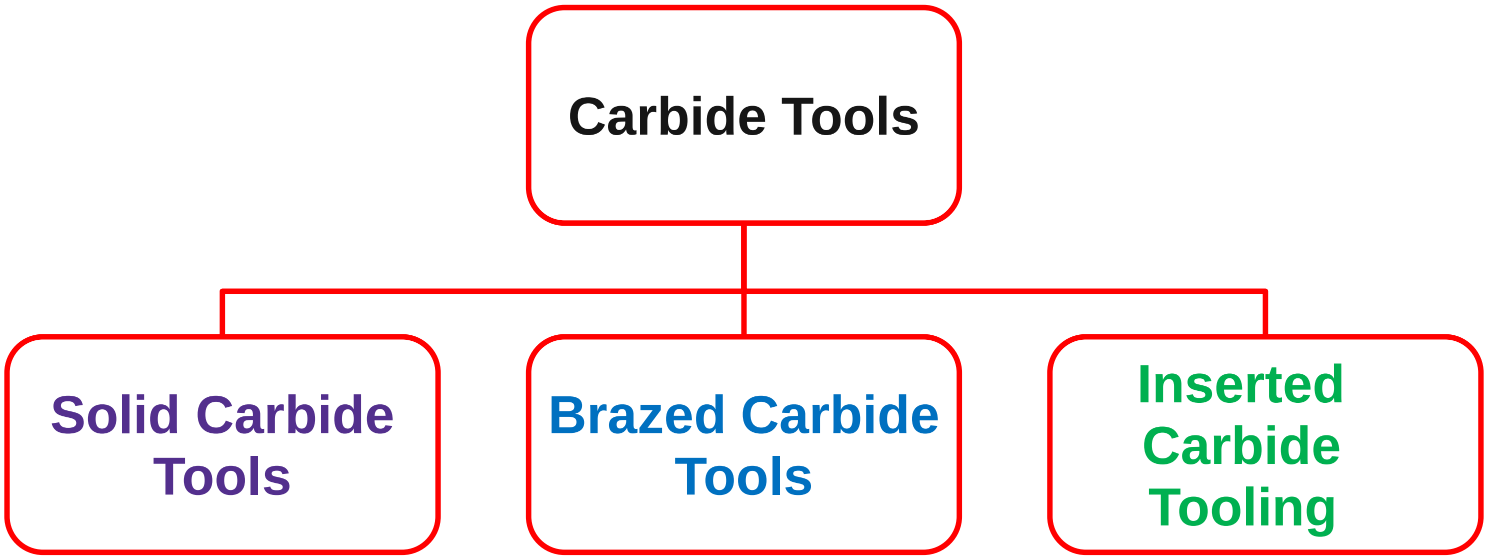
- HSS ***costs less*** than Carbide or Ceramic tooling
- HSS is ***less brittle*** and not as likely to break during interrupted cuts
- The tools can be ***re-sharpened*** easily

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:



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

Properties of Tungsten-Carbide Tools

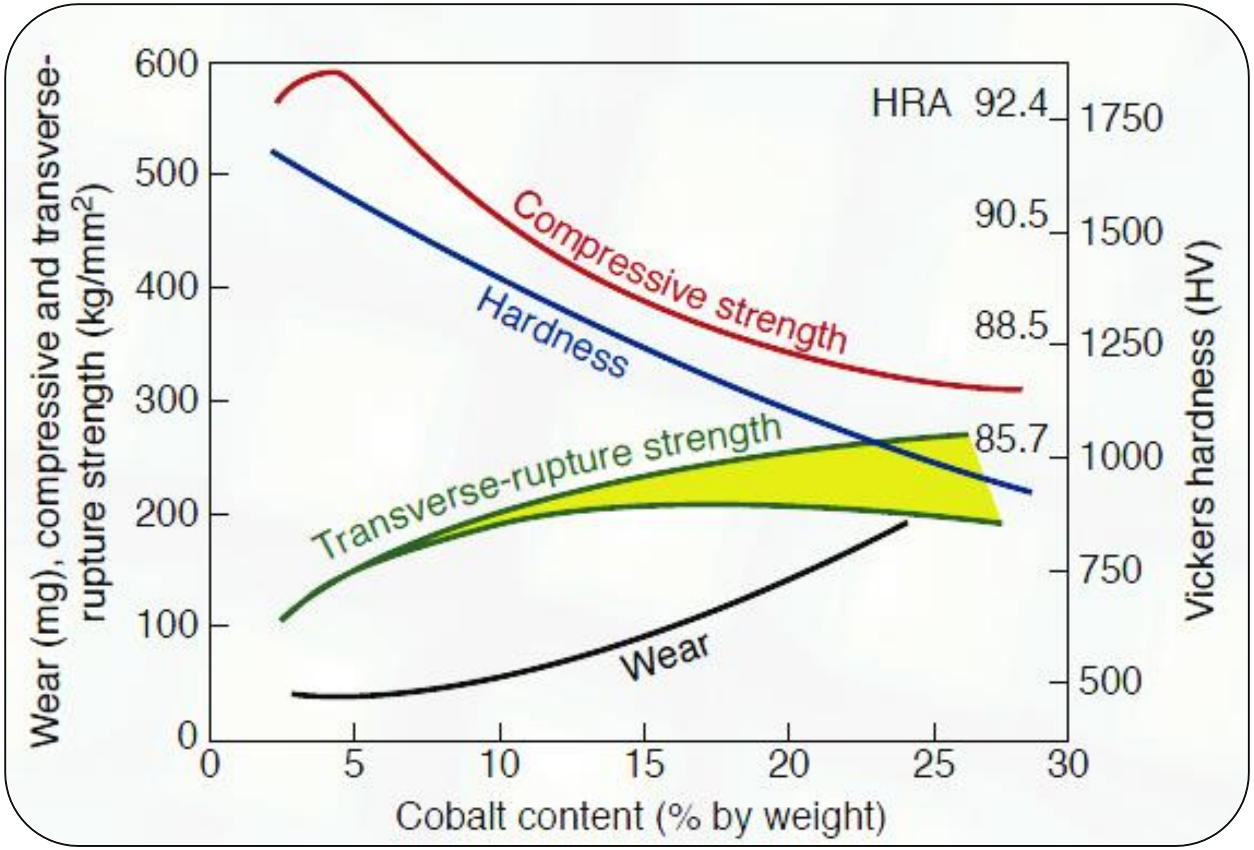


Figure 3-10: Effect of cobalt content in tungsten-carbide tools on mechanical properties. Note that hardness is directly related to compressive strength and hence, inversely to wear

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

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

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

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

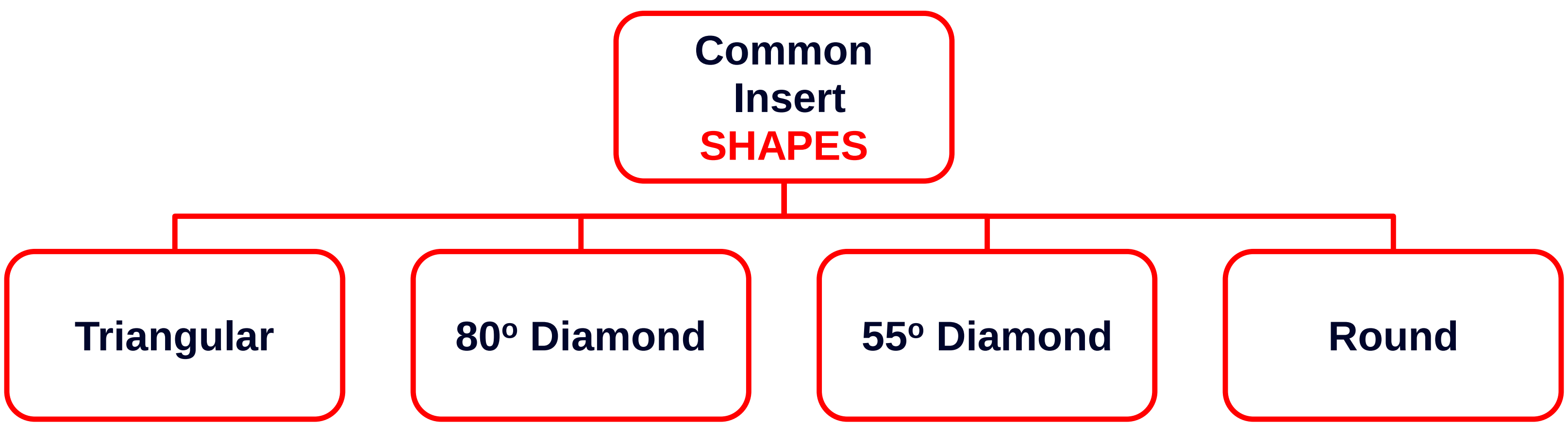
NOTE

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



Insert Shape

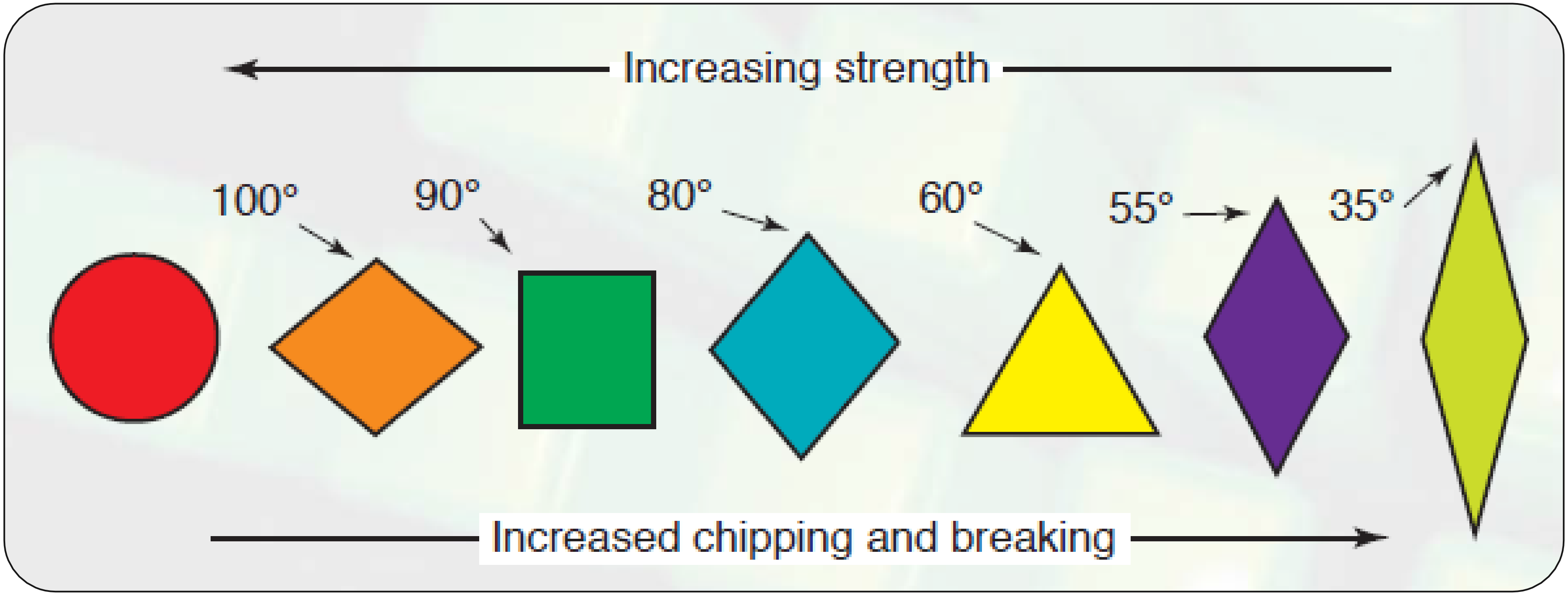


Figure 3-11:Relative edge strength and tendency for chipping and breaking of inserts with various shapes. Strength refers to that of the cutting edge shown by the included angles. (Manufacturing Processes for Engineering Materials, 5th ed. Kalpakjian • Schmid)

Inserts

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

Insert Grading System

Carbide Insert Grading System

INSERT GRADE APPLICATION CHART

Cast iron and nonferrous materials	Alloy and tool steels Stainless steels
C-1: Roughing	C-5: Roughing
C-2: General Purpose	C-6: General Purpose
C-3: Finishing	C-7: Finishing
C-4: Precision Finishing	C-8: Precision Finishing

MANUFACTURER'S GRADE DESIGNATION

ANSI Class	ISO Class	Carboloy	Iscar	Kennametal	Sandvik	Valenite
C-8	P-01 P-05	210	IC-80t	K7H	F02	VC-8
C-7	P-10 P-25	350	IC-70	K45	S1P	VC-7
C-6	P-25 P-35	370	IC-50	KC850	S4	VC-55
C-5	P-40 P-50	518	IC-54	—	S35	VC-5
C-4	K-01 K-05	999	IC-4	K11	—	VC-4
C-3	K-10 K-15	905	IC-20	K68	H10	VC-3
C-2	K-20 K-25	883	IC-2	K6	H20	VC-2
C-1	K-30 K-20	820	IC-28	K1	H	VC-1

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

Figure 3-12 Carbide insert grades

- 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 3-12)
- The programmer is necessary to consult the individual manufacturers catalog to arrive the proper grade number

Insert Grading System

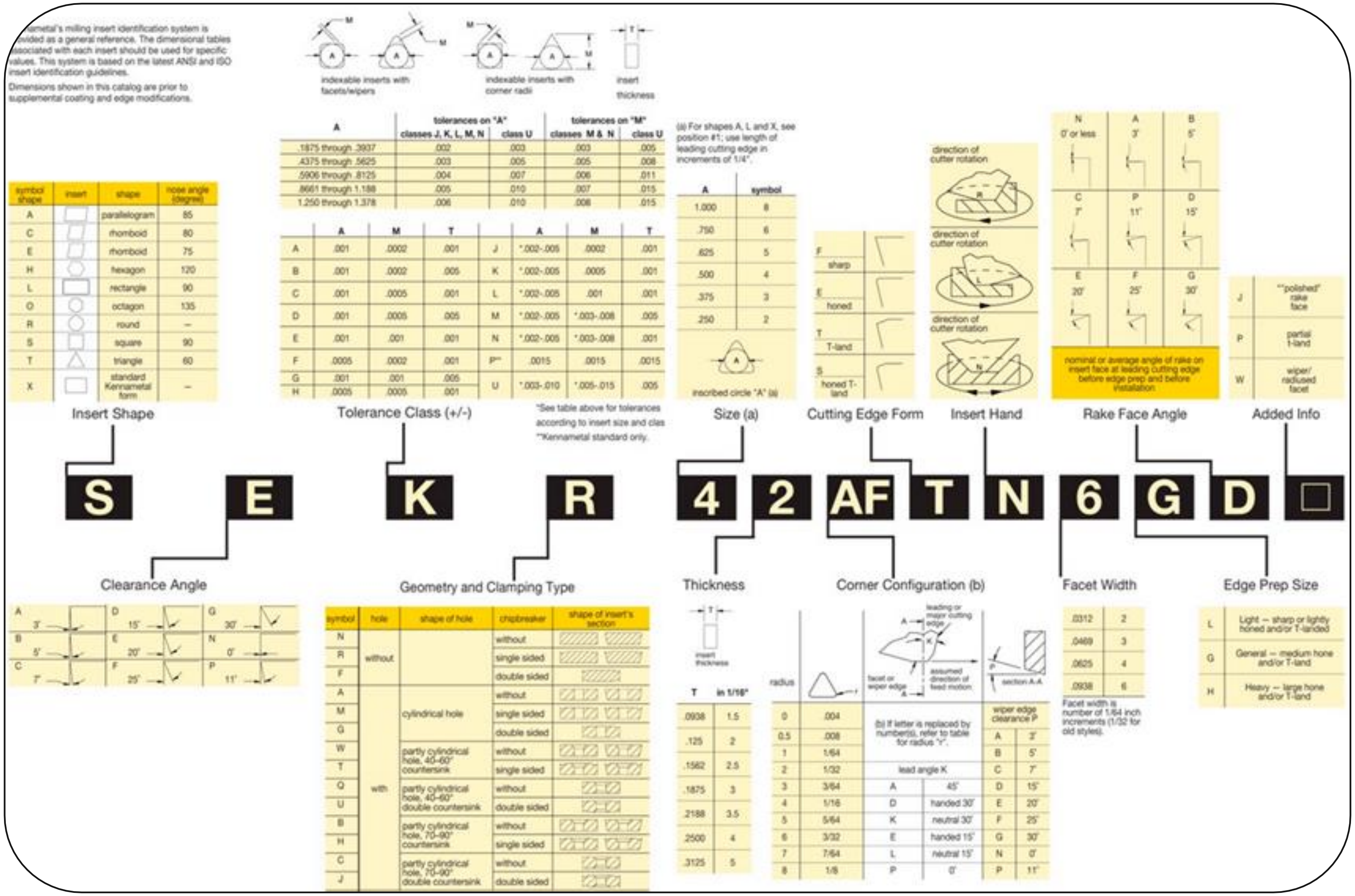
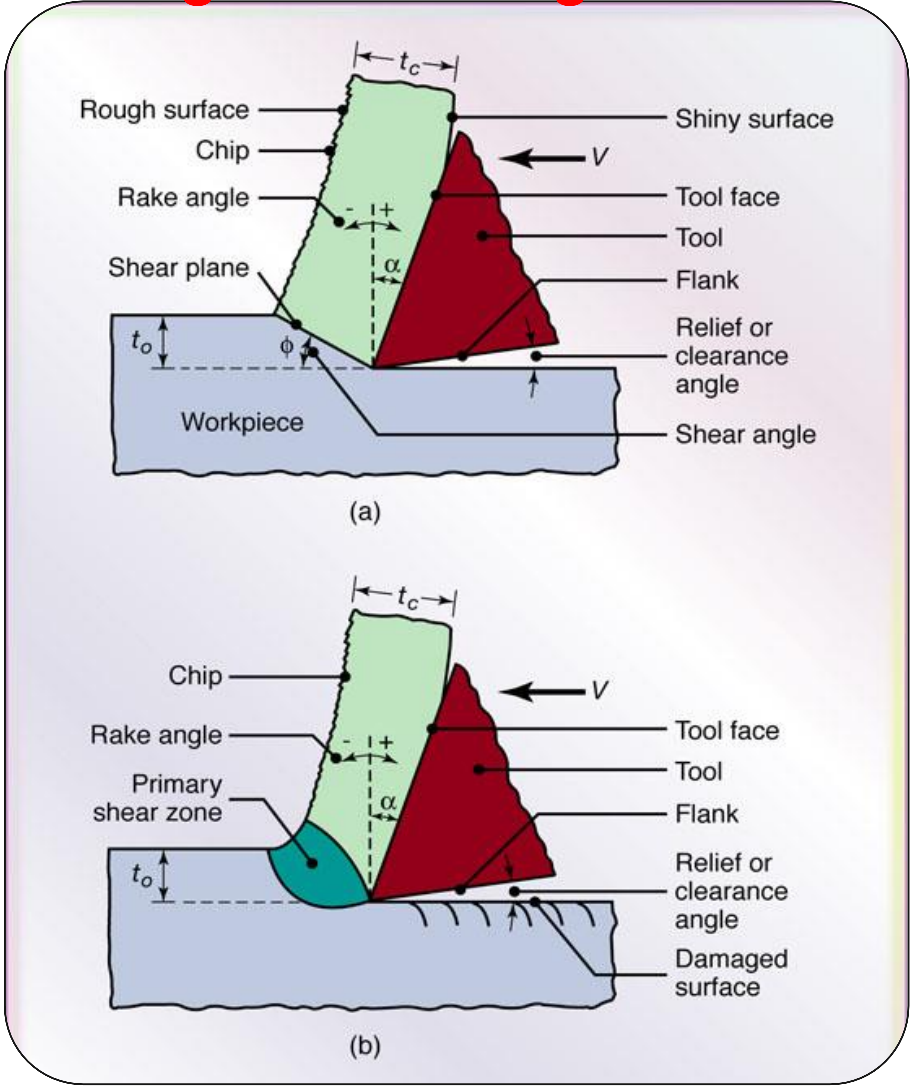


Figure 3-13: Carbide insert identification system *(Photo KENNAMETAL)*

Fundamentals of Machining

Orthogonal Cutting



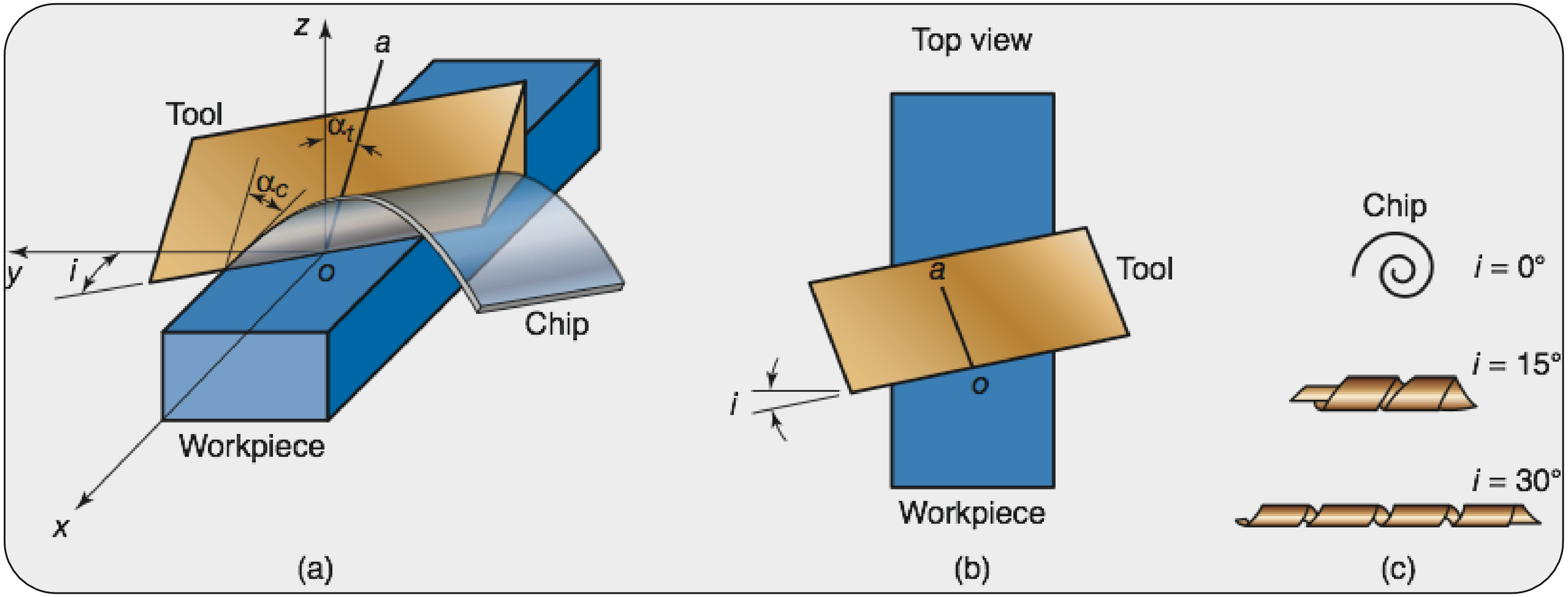
- A two-dimensional cutting process, also called **orthogonal cutting**:

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

(Manufacturing, Engineering & Technology, Fifth Edition, by S.Kalpakjian and S.R. Schmid)

Oblique Cutting



a) Cutting with an oblique tool

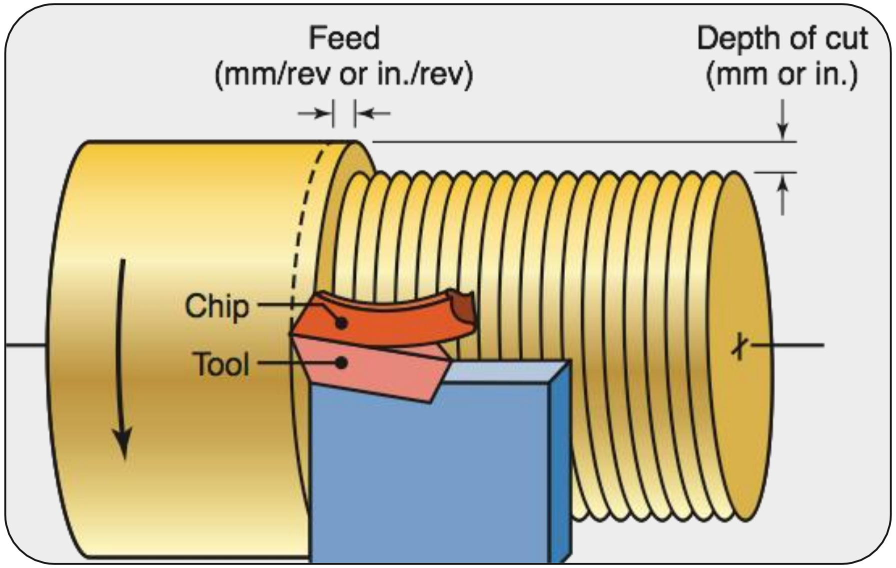
b) Top view, showing the inclination angle, i.

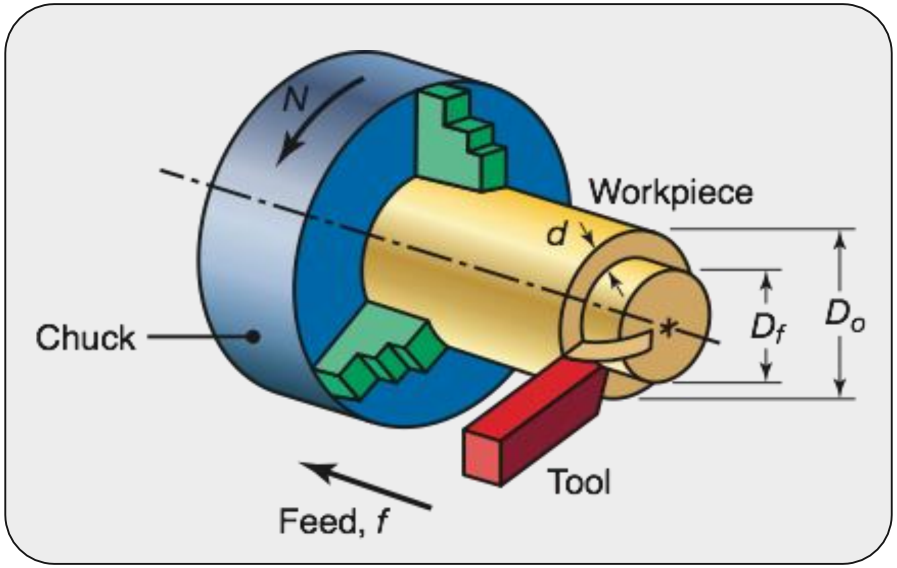
c) Types of chips produced with different inclination angles.

(Manufacturing, Engineering & Technology, Fifth Edition, by S.Kalpakjian and S.R. Schmid)

Turning Operation

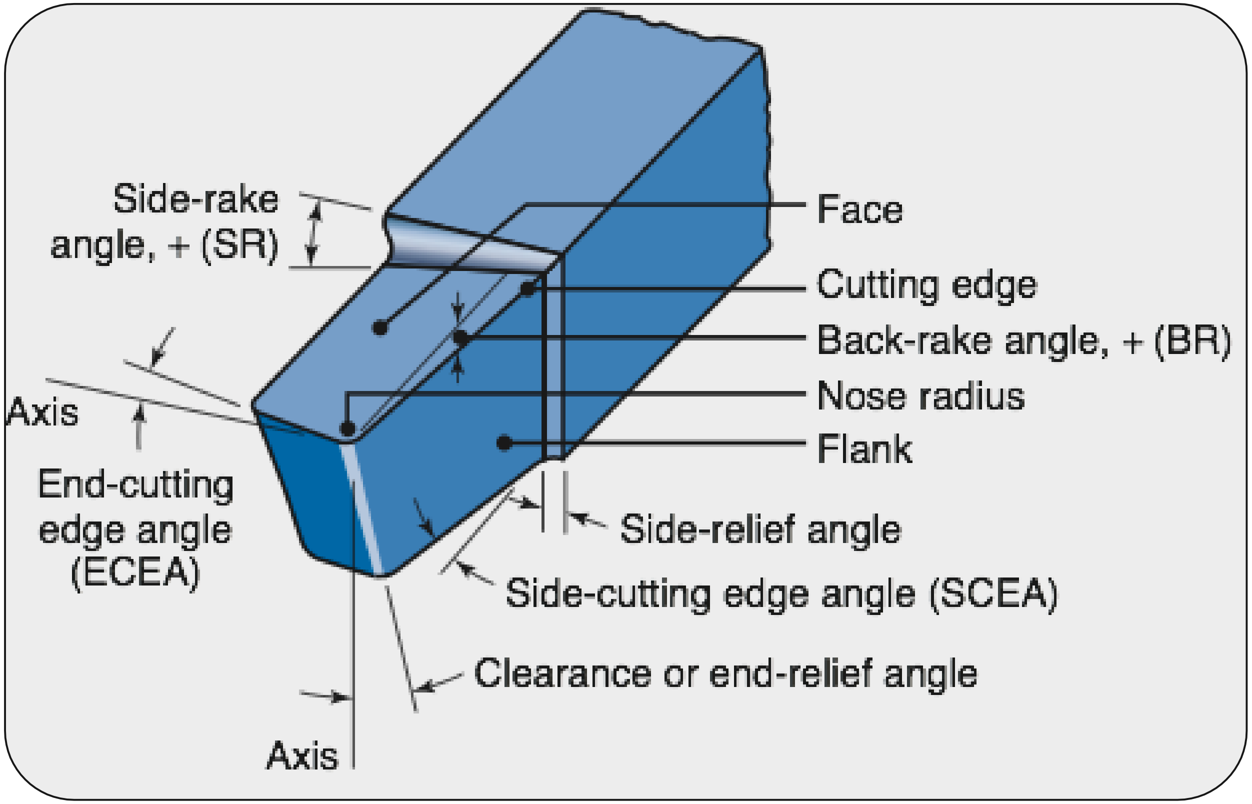
- Terminology used in a turning operation on a lathe, where **f** is the feed (in mm/rev or in./rev) and **d** is the depth of cut.
- Note that **feed in turning is equivalent to the depth of cut in orthogonal cutting**, and the **depth of cut in turning is equivalent to the width of cut in orthogonal cutting**





(Manufacturing, Engineering & Technology, Fifth Edition, by S.Kalpakjian and S.R. Schmid)

Right-hand Cutting Tool for Turning



(Manufacturing, Engineering & Technology, Fifth Edition, by S.Kalpakjian and S.R. Schmid)

Insert Angles

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)*

Insert Angles

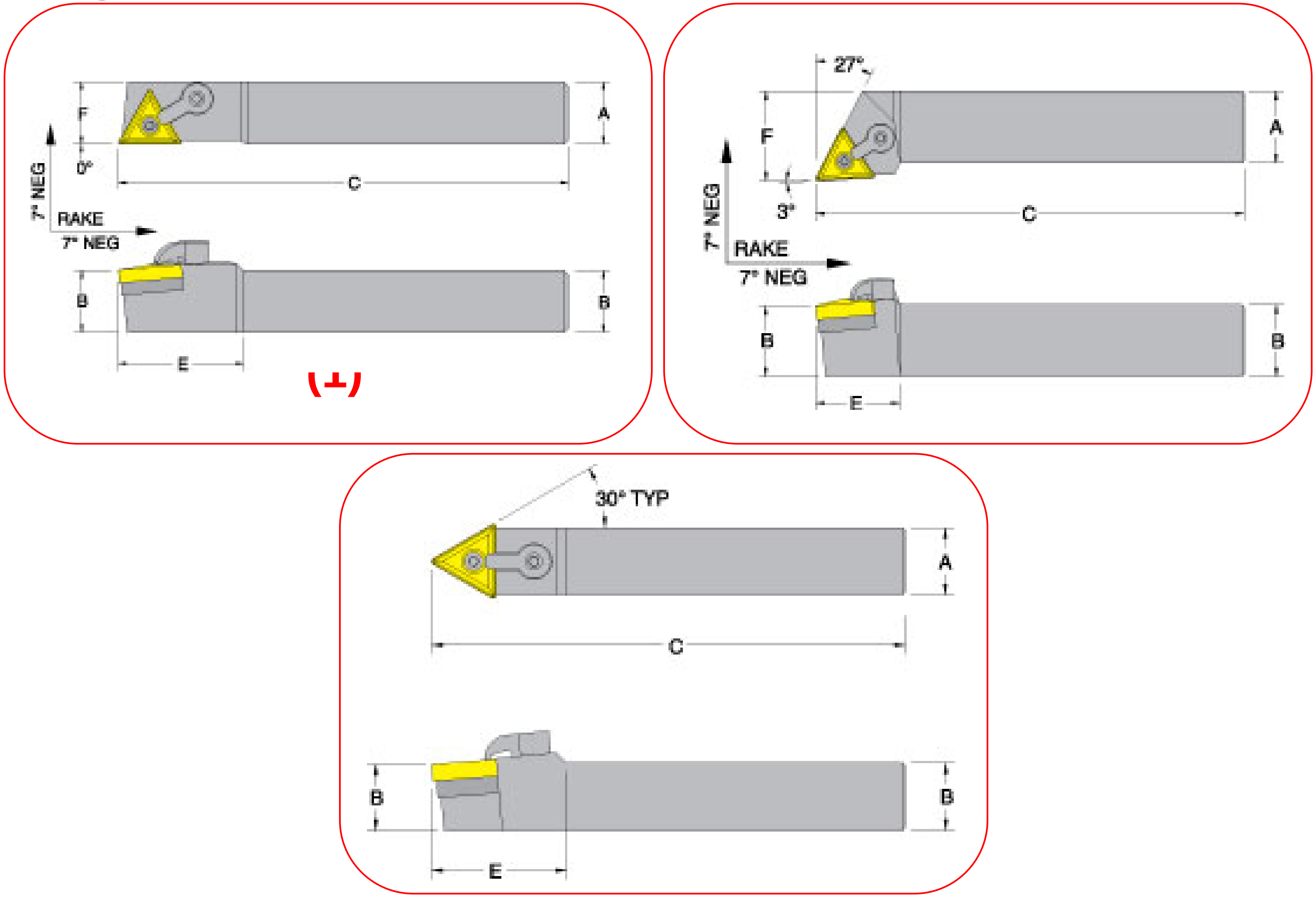
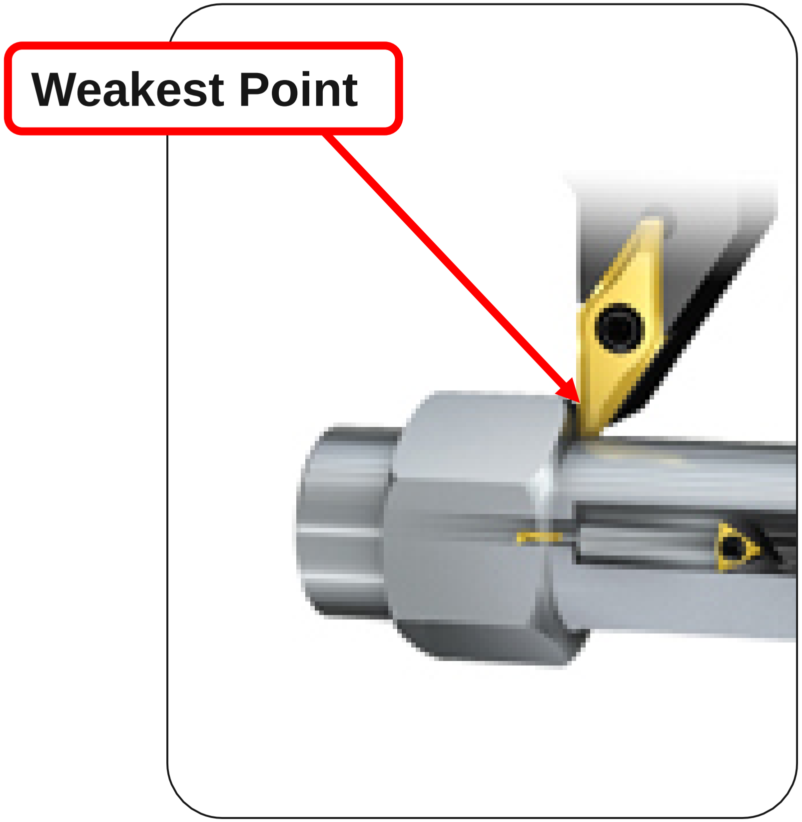


Figure 3-15: Side and top view of rake angles (1)Neutral,(2)Negative and (3)Positive *(Photo ENCO)*

Turning Operation

Negative Rake



Positive Rake

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)

Conventional & Climb Milling

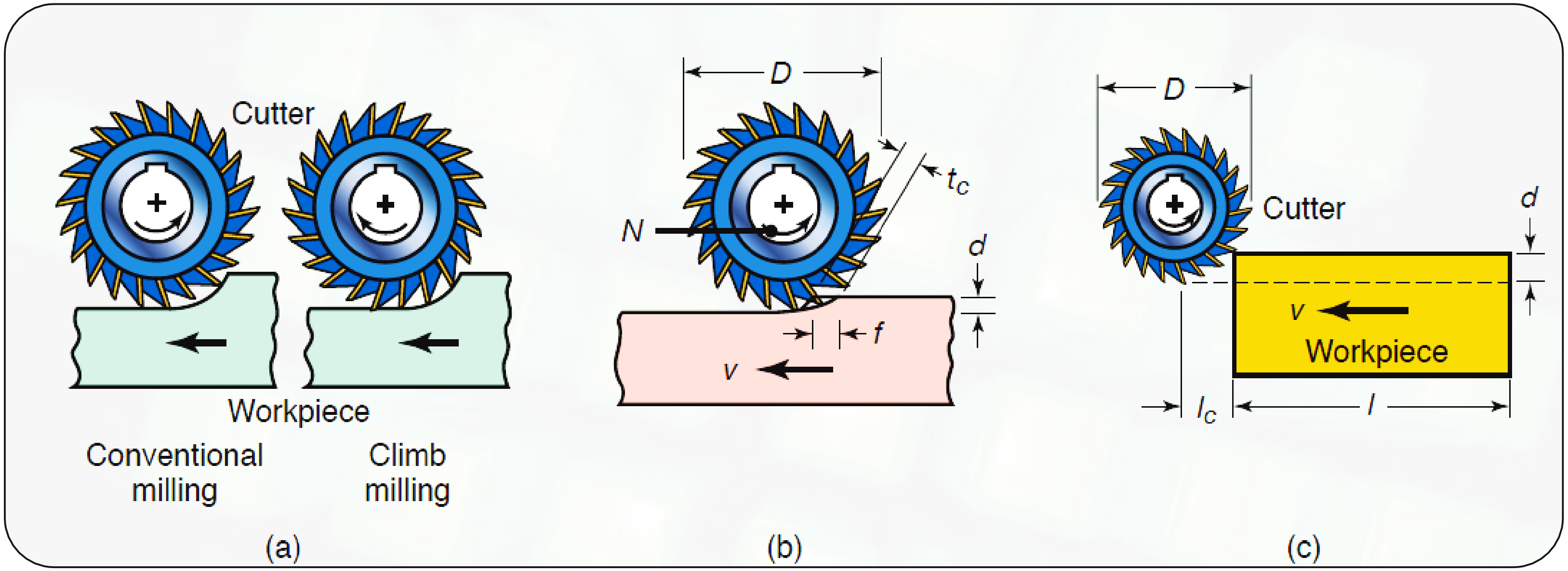
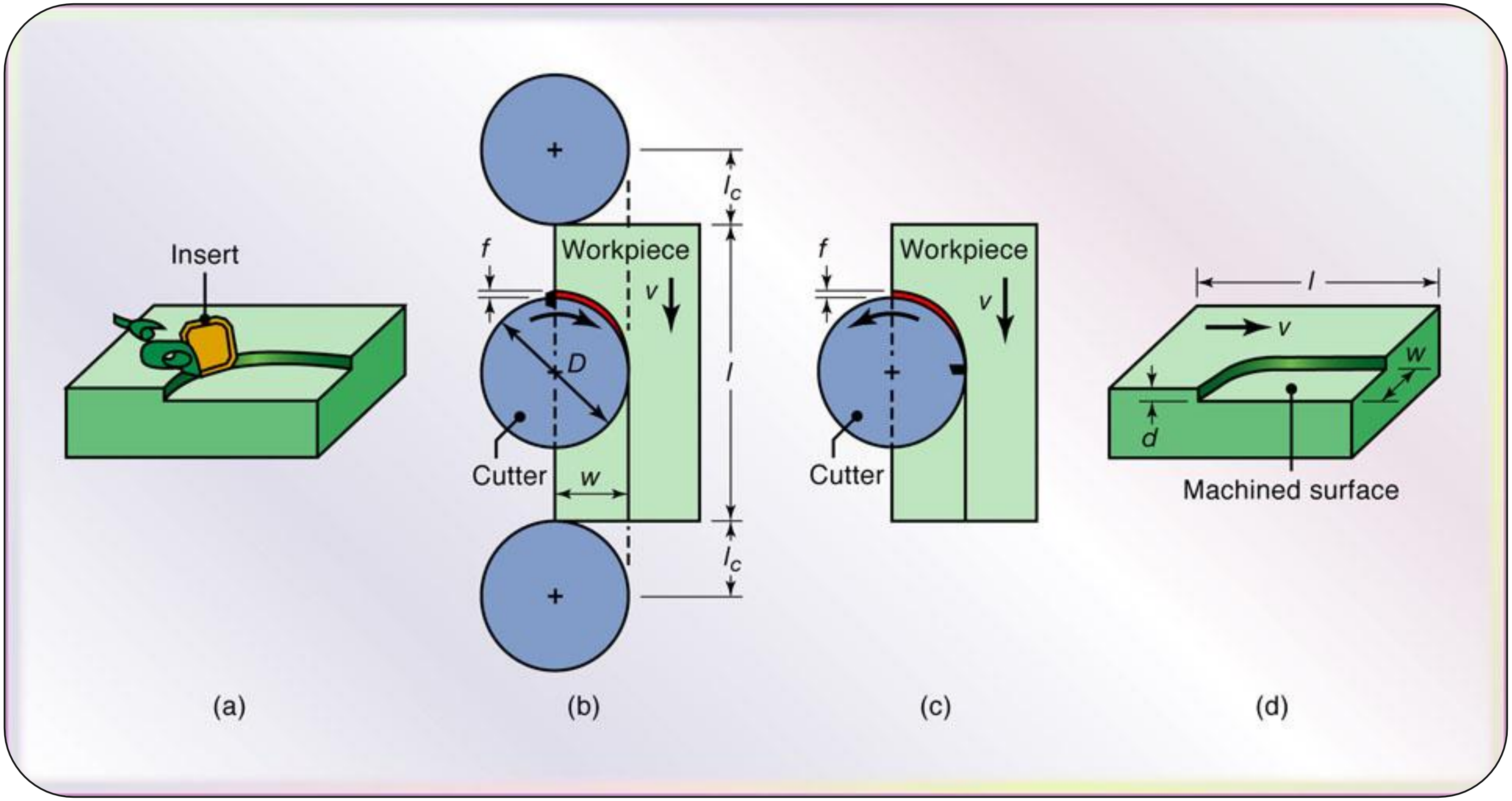


FIGURE 3-17: (a) Illustration showing the difference between conventional milling and climb milling, (b) Slab-milling operation, showing depth of cut, *d*; feed per tooth, *f*; chip depth of cut, *tc* and workpiece speed,(c) Schematic illustration of cutter travel distance, *lc*, to reach full depth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

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

Face Milling

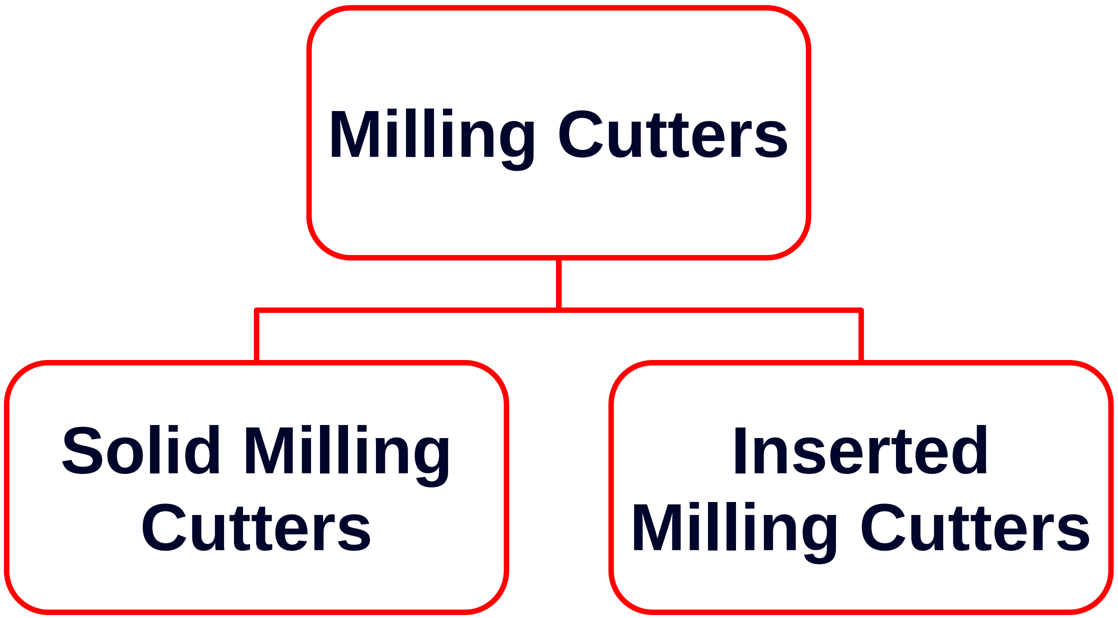


Face-milling operation showing (a) action of an insert in face milling; (b) climb milling; (c) conventional milling; (d) dimensions in face milling. The width of cut, w, is not necessarily the same as the cutter radius(Manufacturing, Engineering & Technology, Fifth Edition, by Serope Kalpakjian and Steven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

Milling Cutters

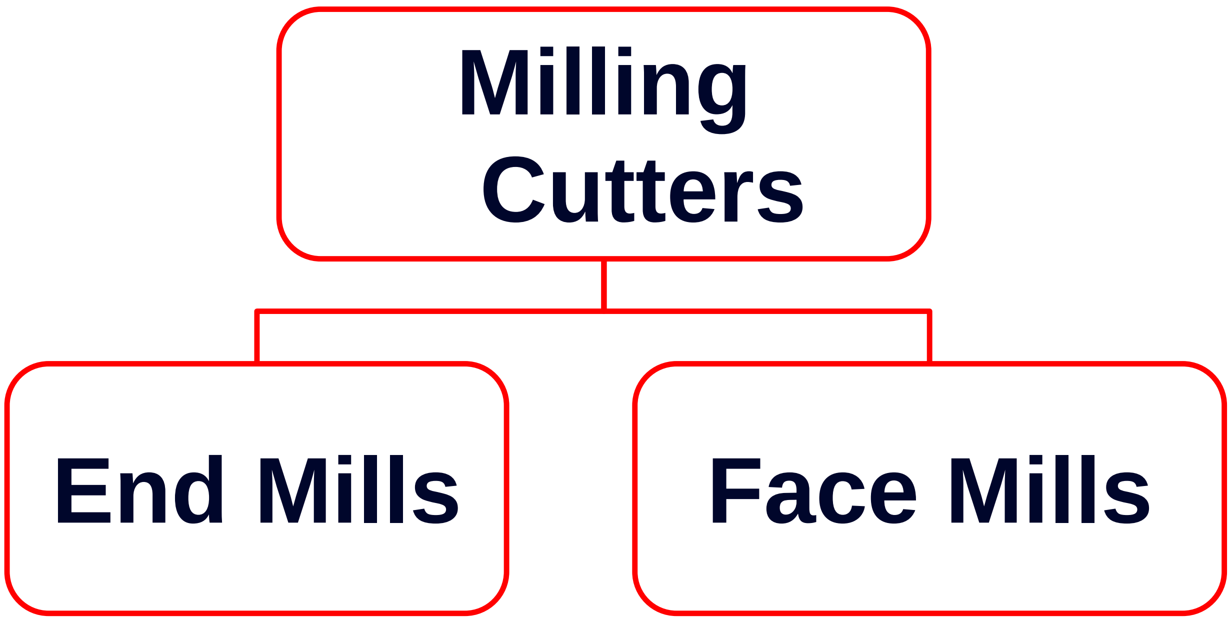
Milling Cutters

- 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:



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

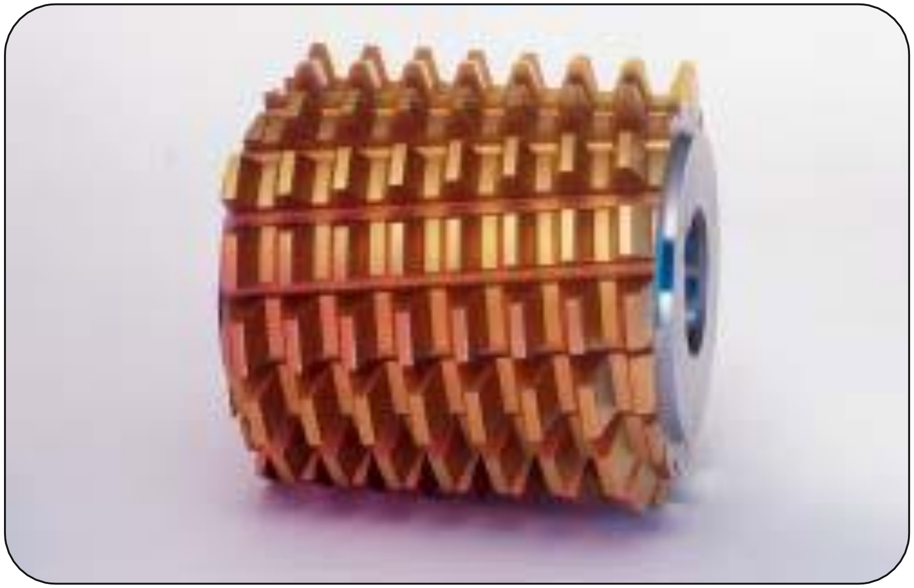


Figure 3-18: Gear hob (Photo Sandvik Coromant) **and thread hob** (Photo Star SU, LLC)

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

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

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

End Mills

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

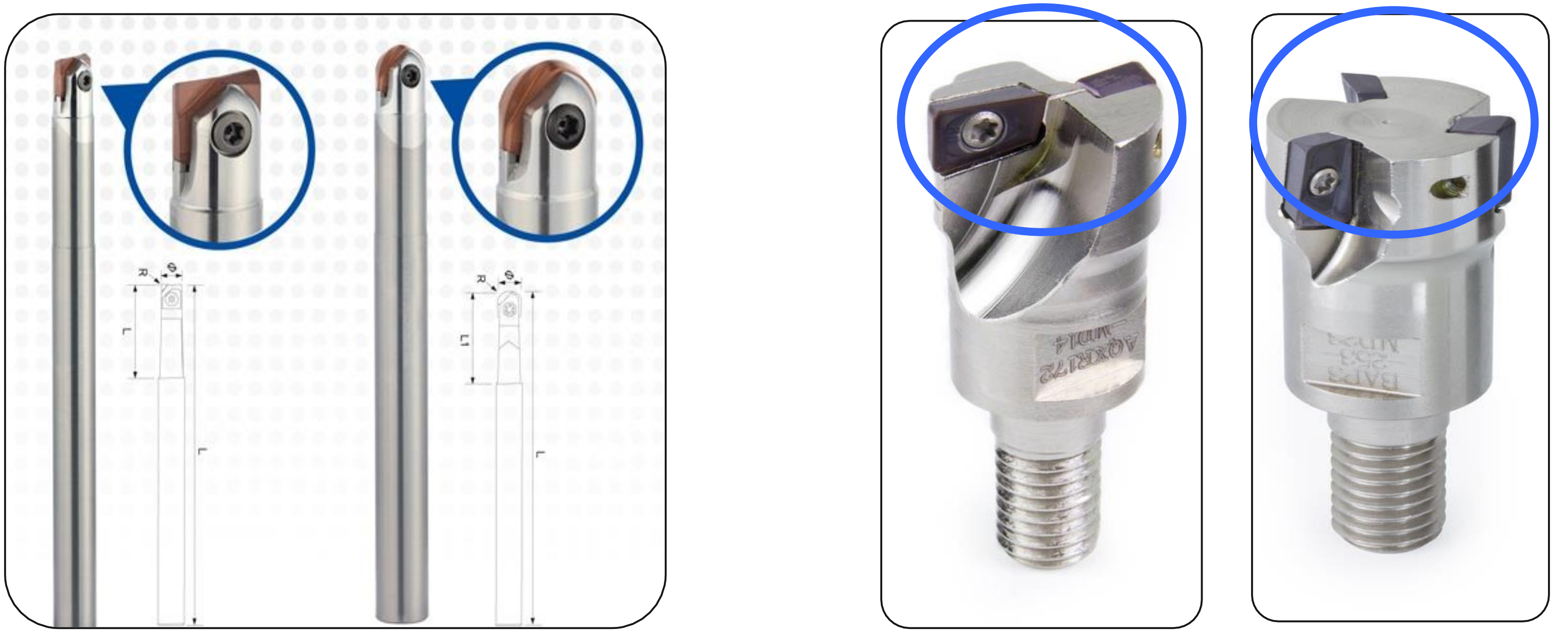
Figure 3-20 :Solid carbide, two-flute, end mill
(Photo MARITool)

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 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



(a) (b) (c)

Figure 3-21:(a) Inserted carbide end mills , (b)and (c) 2 and 3 flute inserted end mills

(Photo Tool Korea Co)

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)*

End Mills

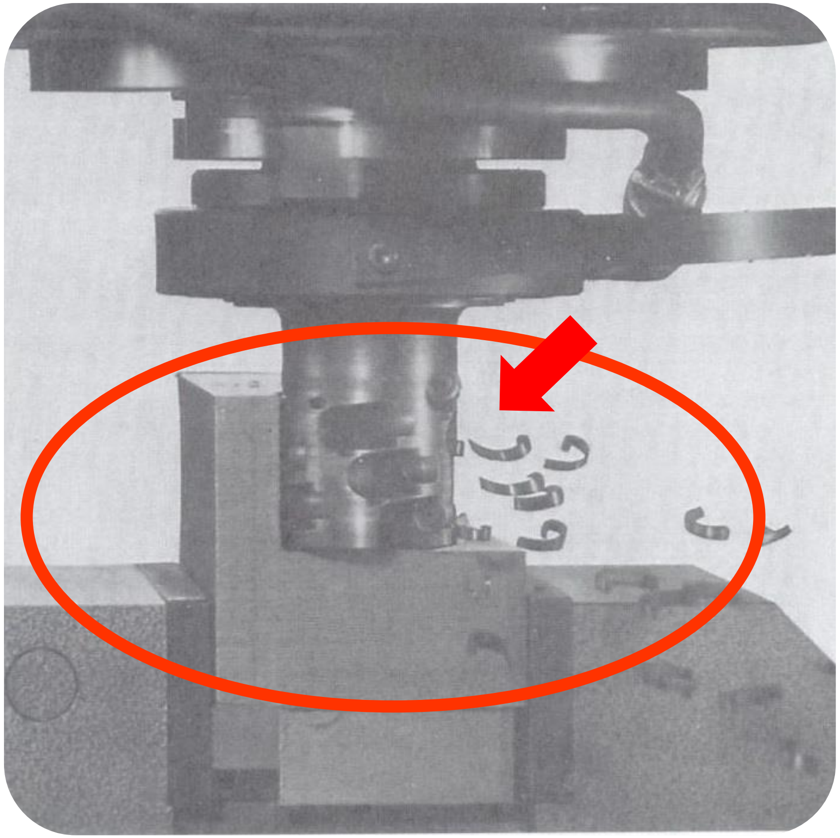


Figure 3-24:"Cyclo Mill" special multi-inserted milling cutter
(Photo GTE Valenite)

- **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

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

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

Face Mills

High number of inserts on the periphery of the cutter

Carbide Inserts

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)*

Face Mills

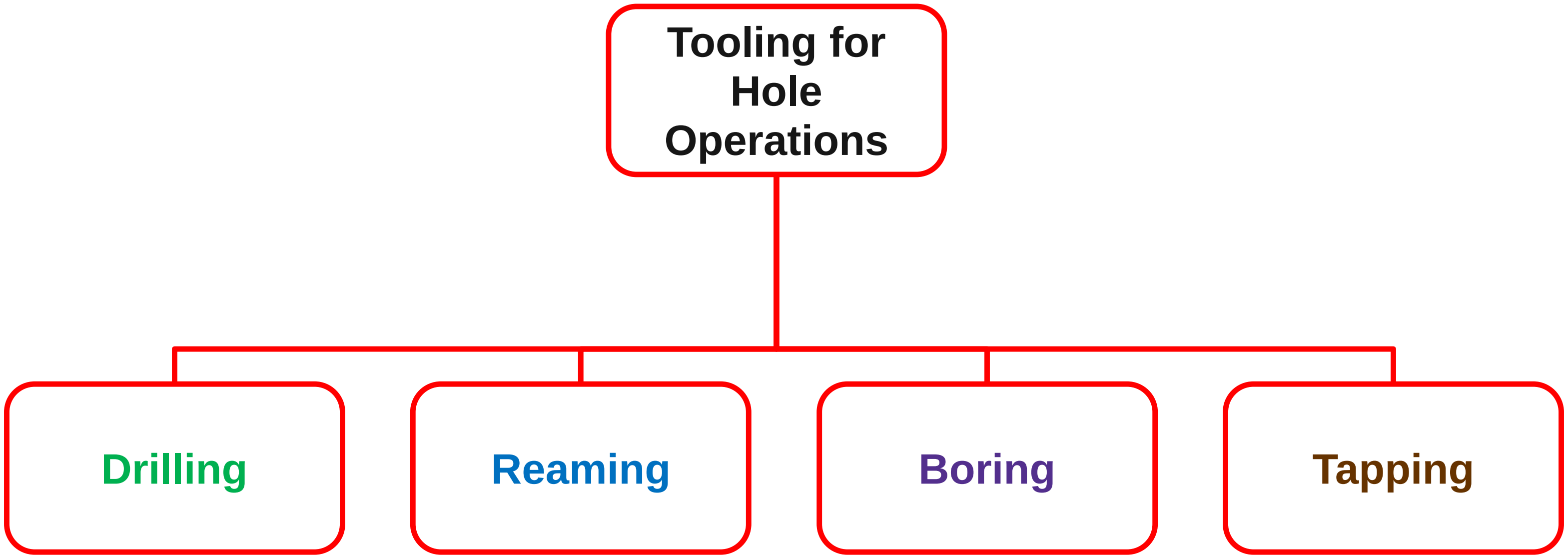
- **Plunge and Profile Cutter** (Fig. 3-27)
- It is designed to plunge into the material first and then begging the cutting path
- The design is a cross between End Mill and Face Mill

Figure 3-27:Plunge and profile inserted milling cutter
(Photo Sandvik Coromant)

Tooling for Hole Operations

Tooling for Hole Operations

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

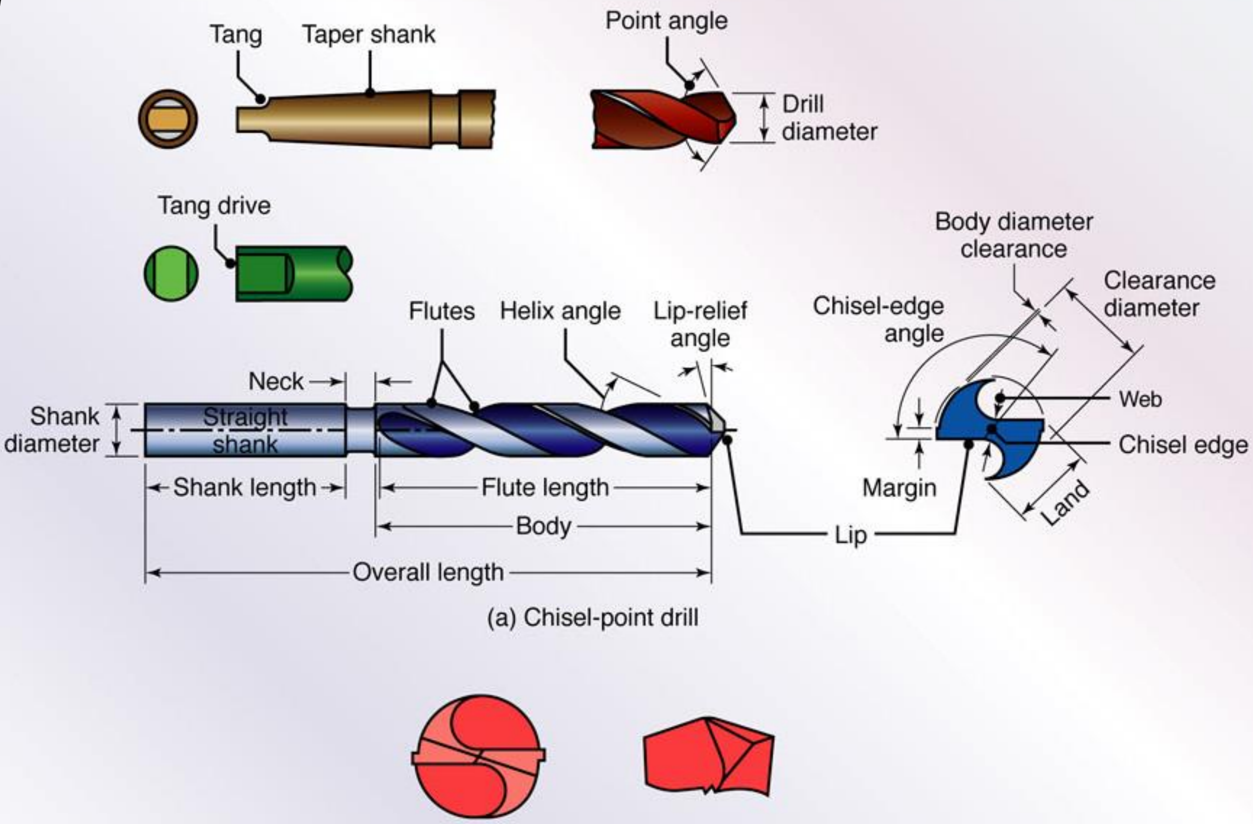


Tooling for Hole Operations

Drilling

- **Drills** are available in different styles for **different materials** (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 it 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

Common Types of Drills



(a) Chisel-point drill

(b) Crankshaft-point drill

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

Various Types of Drills

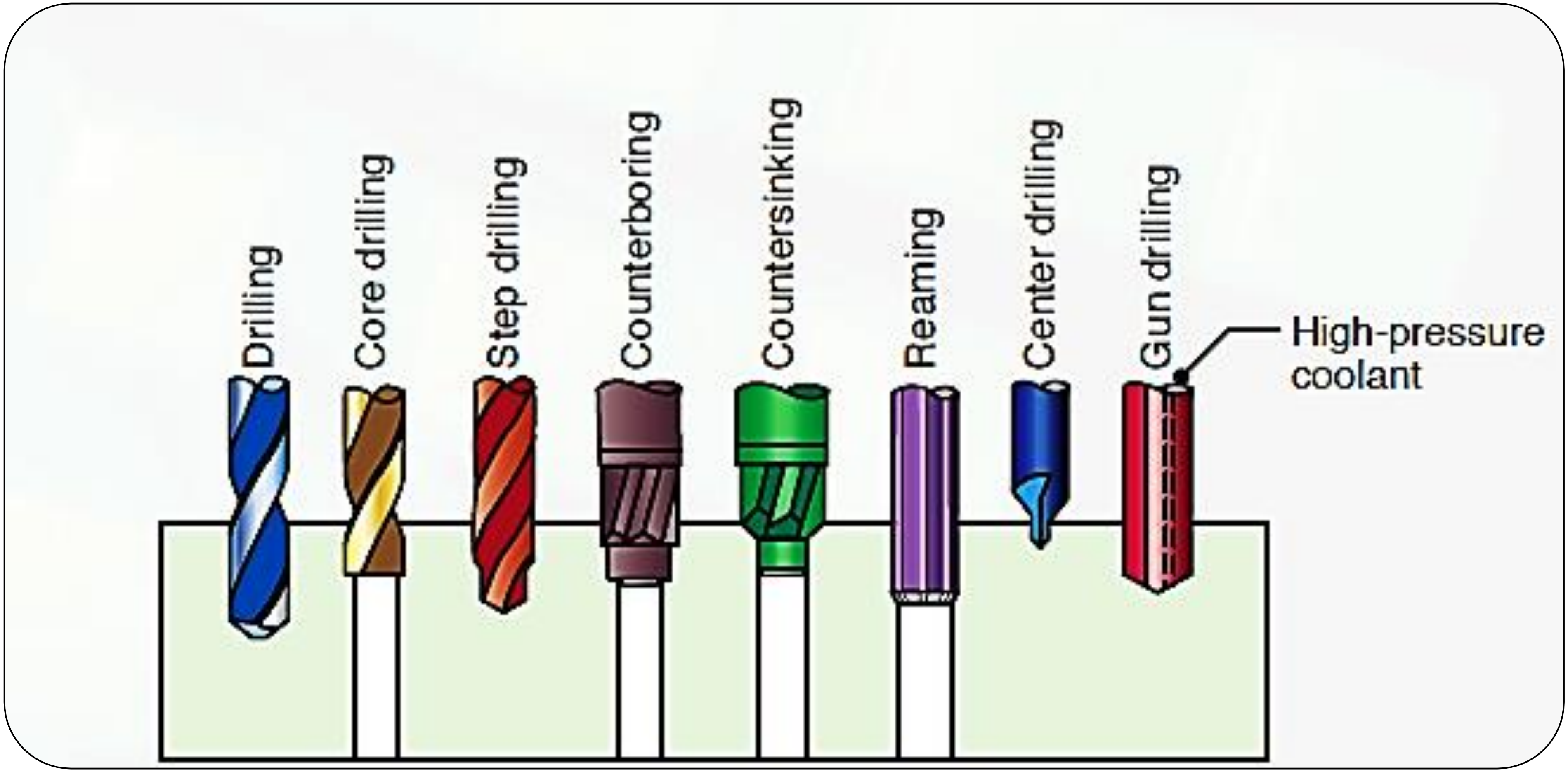


Figure 3-28:Various types of drills and drilling operations

(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

Tooling for Hole Operations

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

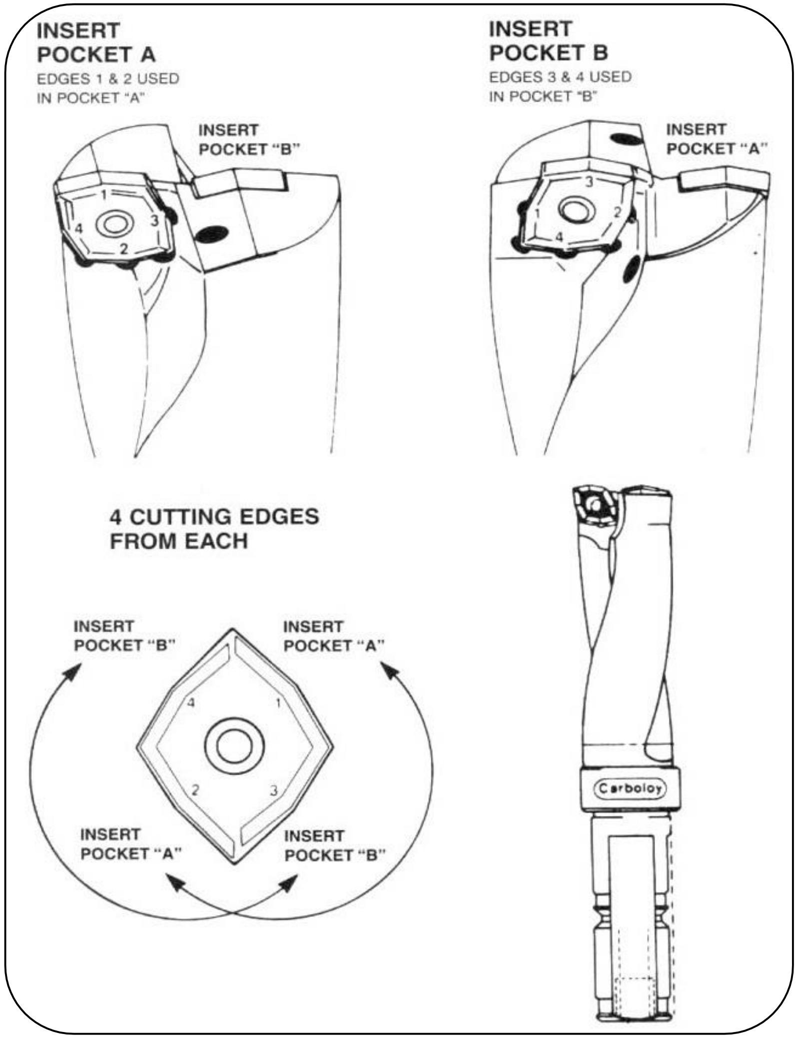


Figure 3-32: (Courtesy Carboloy Inc., A Seco Tools Company)

- **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
- Special drills with Carbide inserts (Fig. 3-32)

LAB Speeds and Feeds in Drilling

Workpiece Material	Surface Speed m/min	Surface Speed ft/min	Feed, mm/rev (in./rev) Drill Diameter 1.5 mm (0.060 in.)	Feed, mm/rev (in./rev) Drill Diameter 12.5 mm (0.5 in.)	Spindle speed (rpm) Drill Diameter 1.5 mm (0.060 in.)	Spindle speed (rpm) Drill Diameter 12.5 mm (0.5 in.)
Aluminum alloys	30-120	100-400	0.025 (0.001)	0.30 (0.012)	6400-25,000	800-3000
Magnesium alloys	45-120	150-400	0.025 (0.001)	0.30 (0.012)	9600-25,000	1100-3000
Copper alloys	15-60	50-200	0.025 (0.001)	0.25 (0.010)	3200-12,000	400-1500
Steels	20-30	60-100	0.025 (0.001)	0.30 (0.012)	4300-6400	500-800
Stainless steels	10-20	40-60	0.025 (0.001)	0.18 (0.007)	2100-4300	250-500
Titanium alloys	6-20	20-60	0.010 (0.0004)	0.15 (0.006)	1300-4300	150-500
Cast irons	20-60	60-200	0.025 (0.001)	0.30 (0.012)	4300-12,000	500-1500
Thermoplastics	30-60	100-200	0.025 (0.001)	0.13 (0.005)	6400-12,000	800-1500
Thermosets	20-60	60-200	0.025 (0.001)	0.10 (0.004)	4300-12,000	500-1500

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

Reaming

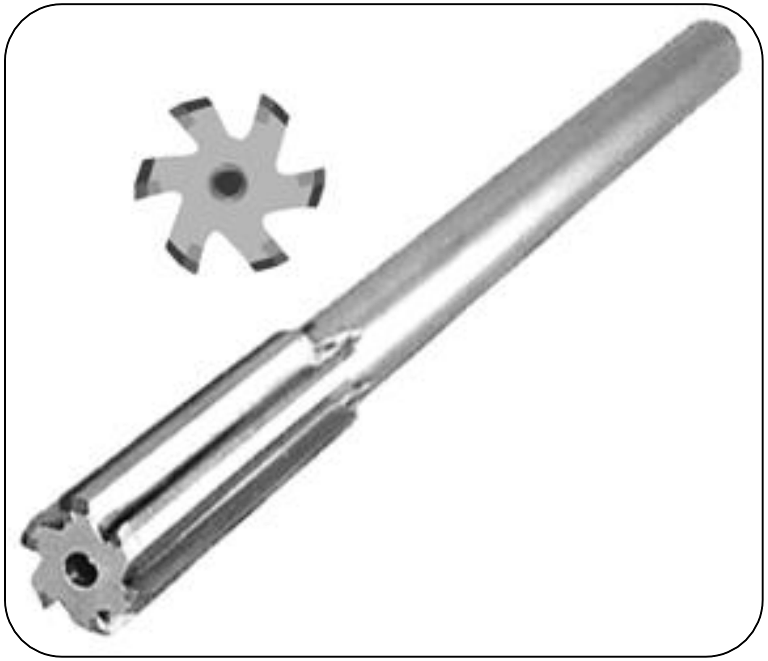


Figure 3-33: Straight flute chucking reamer

(Photo DoALL Manufacturing)

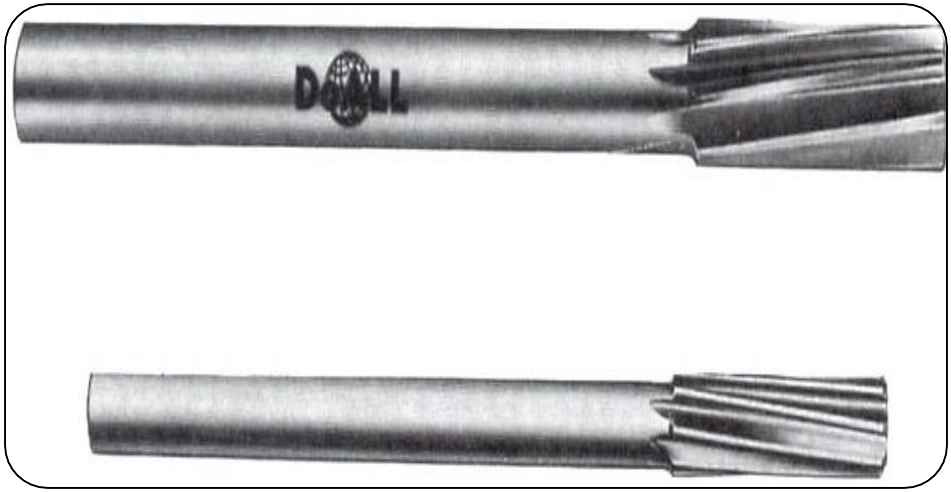


Figure 3-34: Spiral flute chucking reamers

(Photo DoALL Manufacturing)

Reaming

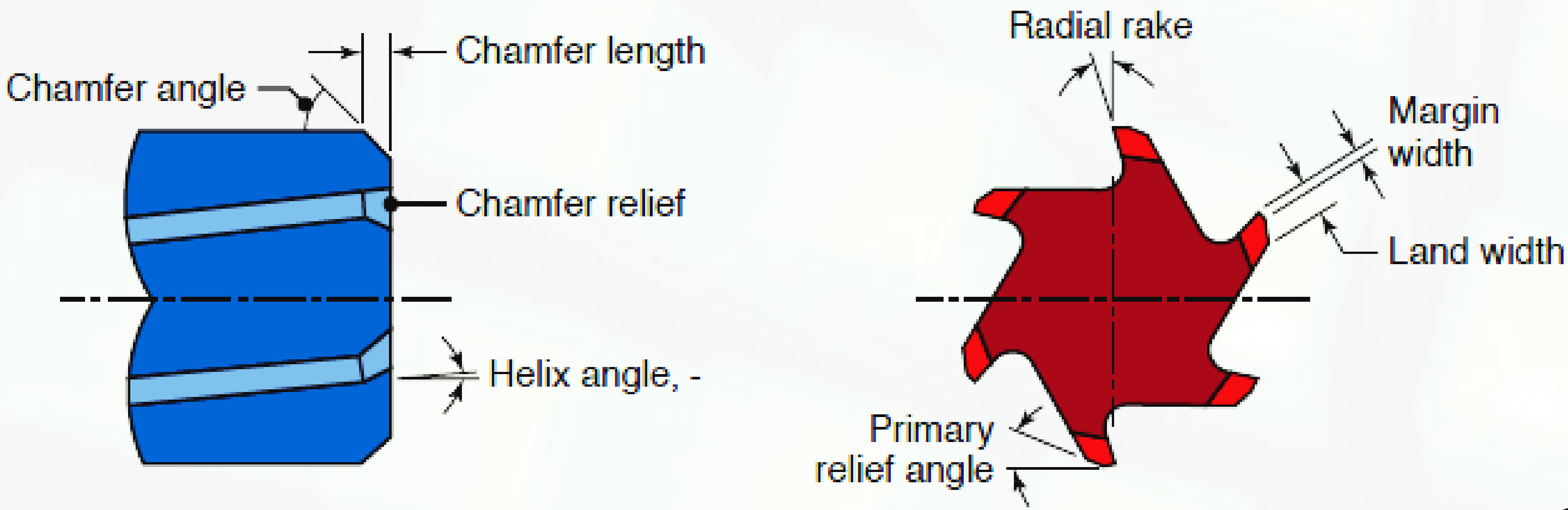


Figure 3-35: Terminology for a spiral fluted reamer

(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**

Tooling for Hole Operations

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

Tooling for Hole Operations

Tapping

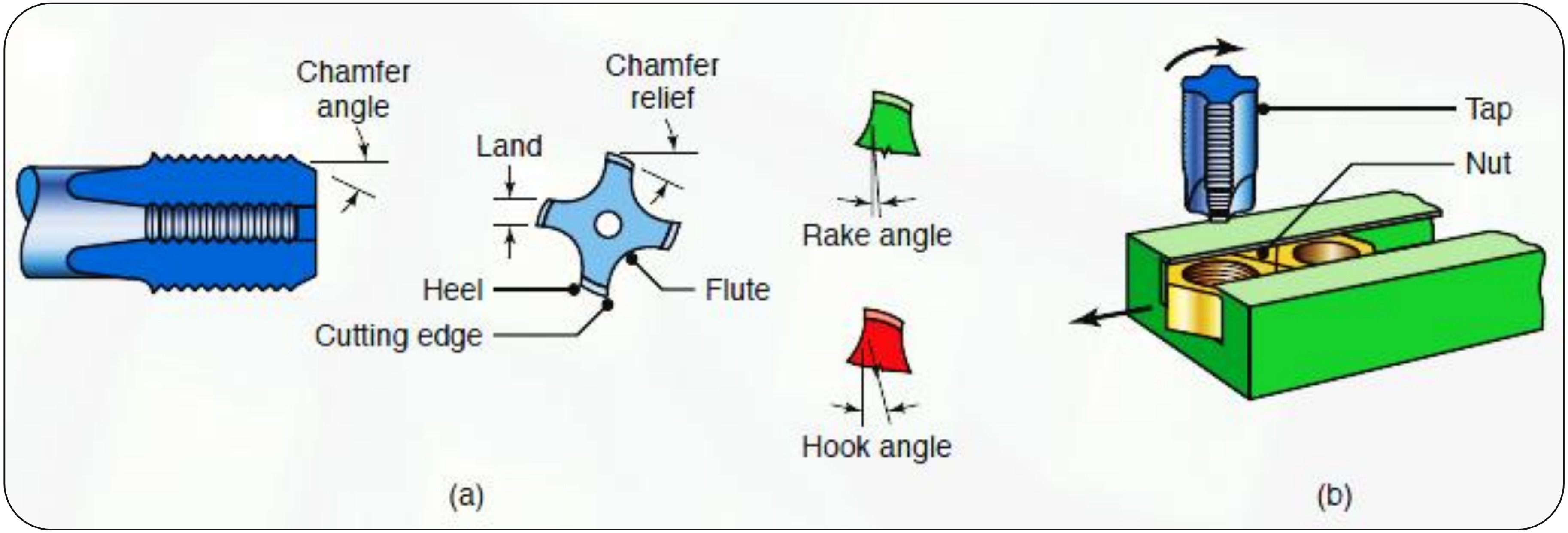


Figure 3-38: (a)Terminology for a tap;(b) illustration of tapping of steel nuts in high production

(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 spent 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

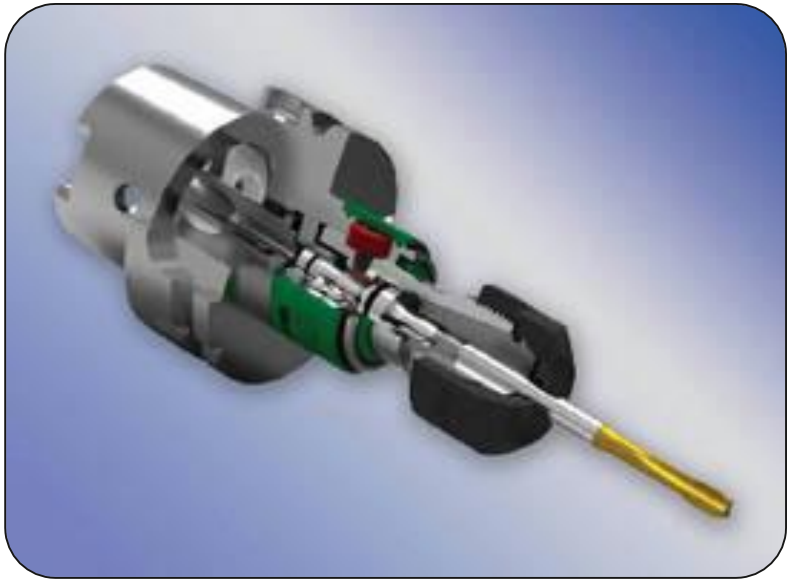


Figure 3-39 :Special inserted tooling for use with NC. From left to right:

- **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)*

Special Inserted Cutters

Figure 3-40 :Indexable Inserted end mill suitable for multi-functional milling

(Photo BIG Kaiser Precision Tooling Inc.)

Special Inserted Cutters

	CoroBore® 820	DuoBore™			Heavy duty	
Boring range (inch)	1.378–12.047	.984–10.630	.984–3.976	3.898–5.906	5.906–11.811	9.843–21.653
Boring depth	4 x D_{5m}	4 x D_{5m}	6 x D_c	23.622 inch	4 x D_{5m}	15.748 inch
Hole tolerance	IT9	IT9	IT9	IT9	IT9	IT9
Material	P M K N S H	P M K N S H	P M K N S H	P M K N S H	P M K N S H	P M K N S H
Number of cutting edges	3	2	2	2	2	2
Insert types	T-Max P CoroTurn® 107	T-Max P CoroTurn® 107	CoroTurn® 107	CoroTurn® 107	T-Max P	T-Max P CoroTurn® 107
Power requirement	Medium, high	(Low), medium	(Low), medium	(Low), medium	Medium, high	Medium, high
Lead angle	6º (15º), 0º, -5º	15º, 6º, 0º	15º, 0º	15º, 0º	15º, 0º, -5º	15º, 0º, -5º

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

Special Inserted Cutters

	Fine boring head	CoroBore® 825 – Fine boring tools				CoroBore® 825 – Damped fine boring tools	
							
Boring range (inch)	.118–1.654	.748–6.953	5.906–12.779	9.843–22.898	9.843–38.646	.906–6.953	5.906–12.779
Boring depth	5 x D_c	4 x D_{5m}	4 x D_{5m}	15.748 inch	15.748 inch	6 x D_c	6 x D_{5m}
Hole tolerance	IT6	IT6	IT6	IT6	IT6	IT6	IT6
Material	P M K / N S H	P M K / N S H	P M K / N S H	P M K / N S H	P M K / N S H	P M K / N S H	P M K / N S H
Lead angle	0°, -1°, -2°	-2°	-2°	-2°	-2°	-2°	-2°

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

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:

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

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

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

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

Table 1 Cutting Speeds for common materials

Speed and Feeds

- **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

Drilling Feeds

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

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):

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

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

$$ipm = 2666 \times 0.005 \rightarrow ipm = 13,33$$

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 eeep 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:

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

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

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

(3.82 is derived from 12 divided (π))

The number 12 is used to convert the inch value of the part diameter into feet

Remember, we measure our parts in inches but use feet in cutting speed calculations.

Speed and Feed Example

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

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

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

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

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

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

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

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

$$CT = 0{,}00302$$

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

Speed and Feed Example

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

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

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

$$f = 0,010$$

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

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

$$F = 87.32\ inches/min$$

Conclusion:

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

LAB

Calculating Feed Rates

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

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

Where :

F_m = Milling feed rate expressed in inches per minute

f_t = Feed in inches / tooth

n_t = number of teeth on the tool

N = Spindle speed in revolutions per minute(rpm)

(Oberg, E. & Jones F. D. & Horton, H. L. & Ryffell, H. H. (2000). Machinery's Handbook, 26th ed., New York, NY: Industrial Press Inc.

Kibbe, R.R., Neely, J.E., Meyer, R.O., & White, W.T. (2002). Machine tool practices. Upper Saddle River, NJ: Prentice Hall.)

Recommended Tool Feed

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

Table 3 : Tool Feed

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

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

Table 3 : Tool Feed

f_t=0.005 in. / tooth

Feed Rate Calculation Example

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

$$n_t = 2$$

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

$$\mathrm{N} = rpm = \frac{CS \times 4}{D} \rightarrow \mathrm{N} = \frac{250 \times 4}{0.5} \rightarrow \mathrm{N} = 2000 rpm$$

Feed Rate Calculation Example

LAB

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

$$F_{\mathrm{m}} = f_t \times n_t \times \mathrm{N} \rightarrow F_{\mathrm{m}} = 0.005 \times 2 \times 2000$$

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

Feed Rate Calculation Example 2

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

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

$$ft = 0.005\ in./tooth$$

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

$$n_t = 4$$

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

$$\mathrm{N} = rpm = \frac{CS \times 4}{D} \rightarrow \mathrm{N} = \frac{250 \times 4}{0.75} \rightarrow \mathrm{N} = 1333.33rpm$$

Feed Rate Calculation Example 2

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

$$F_{\mathrm{m}} = f_t \times n_t \times \mathrm{N} \rightarrow F_{\mathrm{m}} = 0.005 \times 4 \times 1333.33$$

$$\rightarrow F_{\mathrm{m}} = 26.67\ in/min.$$

Summary 1/2

- **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

Summary 2/2

- **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

1. Ann Mazakas, Manager of Technical Communications, DP Technology, ‘’The Art of Automation’’, courses materials of ESPTRIT World Conference on CNC
2. Chryssolouris G., «Manufacturing Systems: Theory and Practice», 2nd Edition, 2006, Springer-Verlag
3. http://toolkorea.en.ec21.com/
4. http://www.alliedmaxcut.com/
5. http://www.bigkaiser.com/
6. http://www.doallind.com/
7. http://www.dptechnology.com/
8. http://www.emuge.com/technical/pdf/Toolholding/zp10024_gb.pdf
9. http://www.fioramachinery.com.au/
10. http://www.iscar.com/

References

11. http://www.iscar.com/
12. http://www.kennametal.com/
13. http://www.maritool.com/
14. http://www.sandvik.coromant.com/
15. http://www.star-su.com/
16. http://www.travers.com/ttc-production
17. http://www.use-enco.com/
18. Kalpakjian S., «Manufacturing Engineering and Technology», 2nd Edition, 1992, Addison-Wesley Publishing company
19. Kalpakjian,Schmid,<<Manufacturing Processes for Engineering Materials>>, 5th ed. 2008
20. Kibbe, R.R. & Neely, J.E. & Meyer, R.O. & White, W.T. (2002). Machine Tool Practices, Upper Saddle River, NJ: Prentice Hall.
21. Mattson M., “CNC Programming, Principles and Applications”, Delmar, 2002
22. Ming X G, Yan J Q, Wang X H, Li S N, Lu W F, Peng Q J and Ma Y S (2008) Collaborative process planning and manufacturing in product lifecycle management. Computers in Industry 59(2-3):154-166

References

23. Moriwaki T., “Multi-Functional Machine Tool”, CIRP Annals Manufacturing Technology, Vol. 57/2, 2008, pp. 736-749
24. Newman S T, Nassehi A, Xu X W, Rosso Jr. R S U, Wang L, Yusof Y, Ali L, Liu R, Zheng L Y, Kumar S, Vichare P and Dhokia V (2008) Strategic advantages of interoperability for global manufacturing using CNC technology. Robotics and Computer-Integrated Manufacturing 24(6):699-708
25. Oberg, E. & Jones F. D. & Horton, H. L. & Ryffell, H. H. (2000). Machinery’s Handbook, 26th ed., New York, NY: Industrial Press Inc.
26. Kibbe, R.R., Neely, J.E., Meyer, R.O., & White, W.T. (2002). Machine tool practices. Upper Saddle River, NJ: Prentice Hall.
27. Rehg J A and Kraebber H W (2005) Computer Integrated Manufacturing, 3rd ed. Prentice-Hall, Upper Saddle River, New Jersey
28. Seams W., “Computer Numerical Control, Concepts & Programming”, 4th Edition, Delmar, 2002
29. Zeid I (1991) CAD/CAM Theory and Practice, International ed. (Computer Science Series). MacGraw-Hill, New York
30. Γ. Χρυσολούρης, «Συστήματα Παραγωγής Θεωρία και Πράξη» Μέρος I και II, 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 I και II”», 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 – II. Τεχνολογία κατεργασιών κοπής των μετάλλων», 1998, Εκδόσεις Ζήτη
35. Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,Κ.Σαλωνίτης