Machining introduction

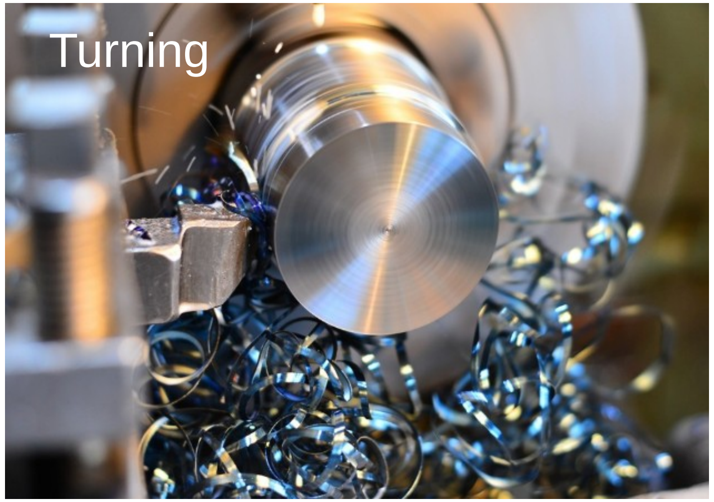
Tony Schmitz

Machining introduction

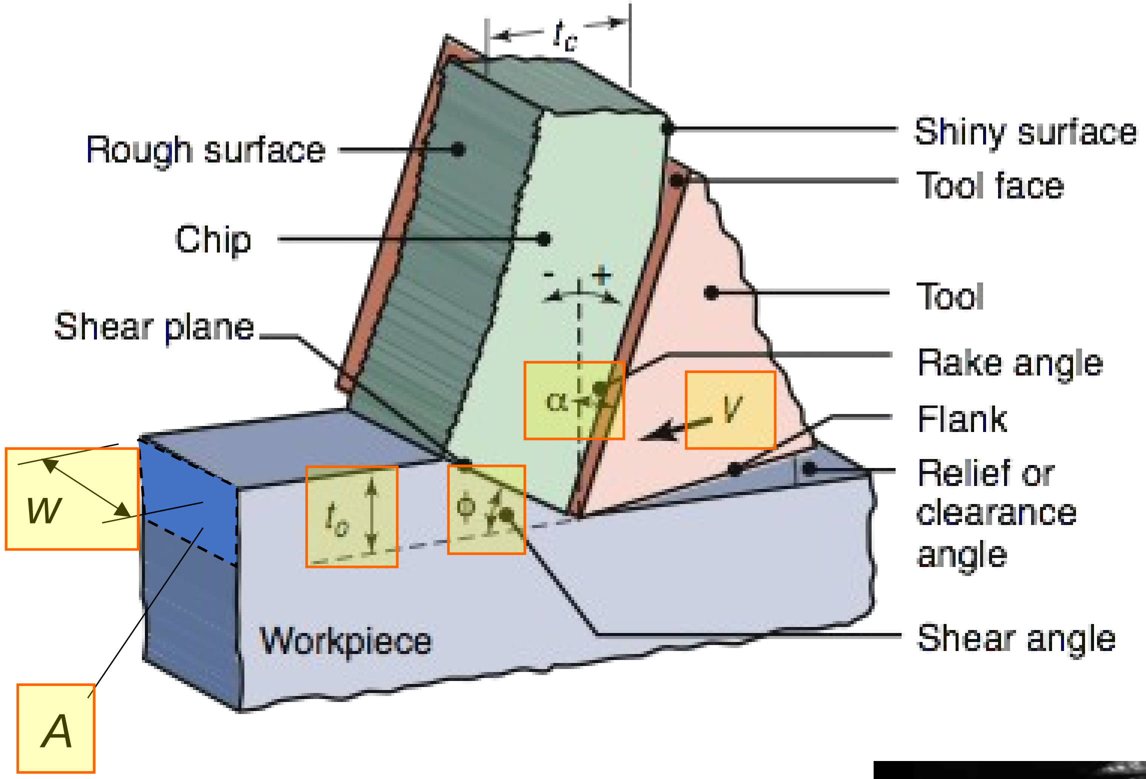
- In machining we remove material using a defined cutting edge to produce parts with the desired dimensions.
- Because machining removes material from the stock, it is a subtractive process.
- This contrasts additive manufacturing, where we deposit material to produce the desired shape.

- Machining can provide
  - good dimensional accuracy
  - internal features
  - sharp edges
  - good surface finish
  - cost benefit for small batch sizes.

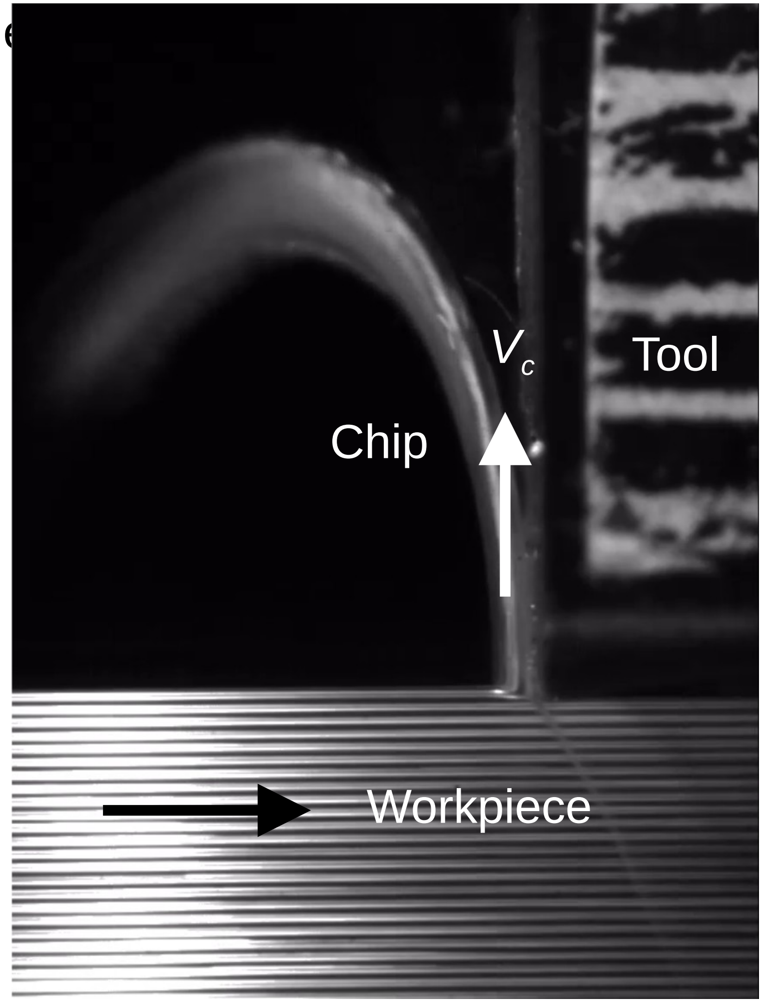
Material is removed in the form of chips that are sheared away by the cutting edge



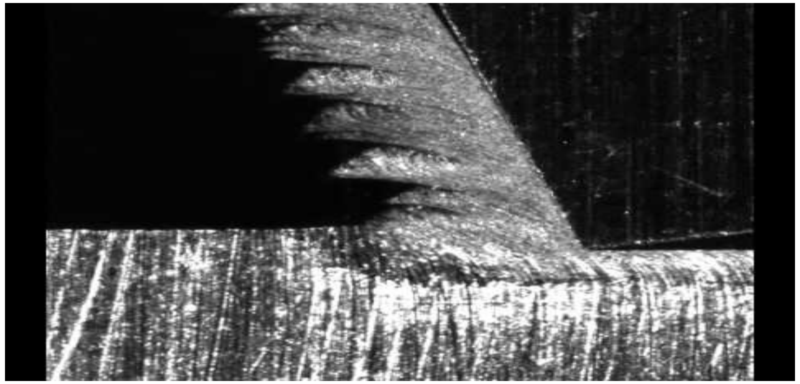
Material is removed in the form of chips that are sheared away by the cutting edge



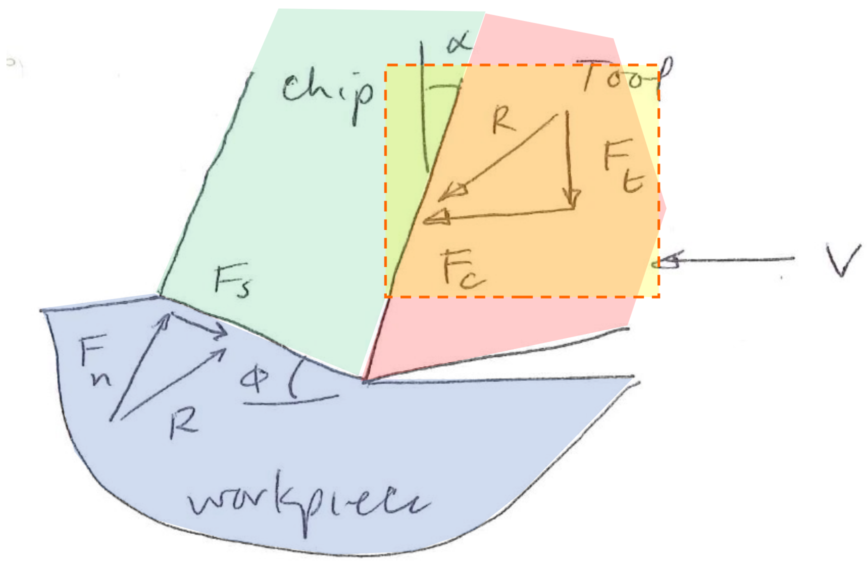
t_0 – commanded chip thickness
w – chip width
A – chip area
V – cutting speed
V_c – chip velocity
α – rake angle
ϕ – shear plane angle



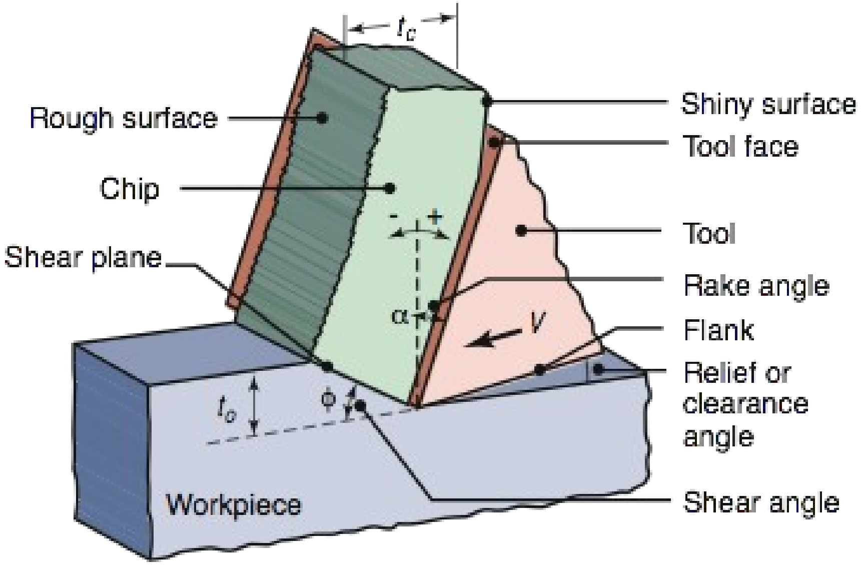
Chip formation video
https://www.youtube.com/watch?v=6QZ98Klssr4

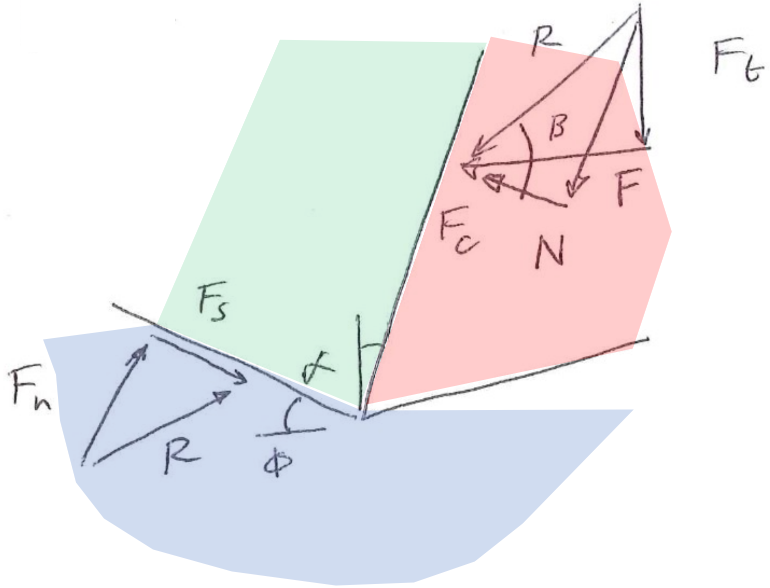


Force is required to shear away the chip

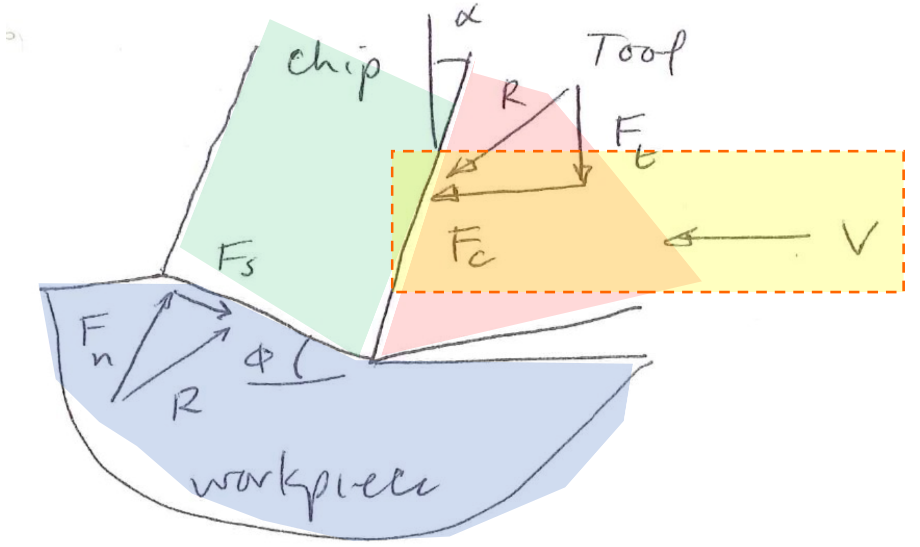


F_c – cutting force acts in cutting speed (*V*) direction
F_t – thrust force is perpendicular to V direction
R – resultant force is the vector sum of F_c and F_t





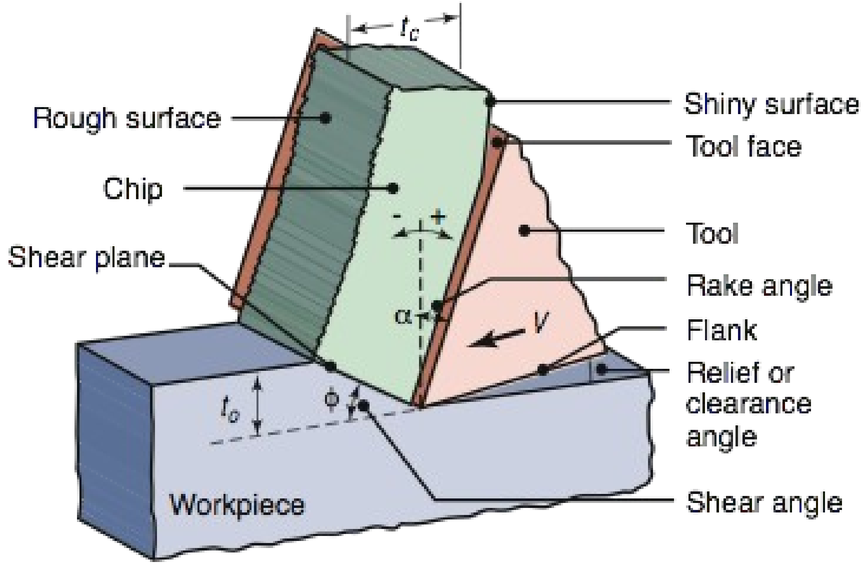
The product of force and velocity is power, *P*.



$P = F_c V$

Power per unit volume is the specific energy, K_s

$K_s = F_c V/(w t_0 V) = F_c/(w t_0)$



MATERIAL	SPECIFIC ENERGY	
	J/mm^3	$hp\text{-}min/in^3$
Al alloys	0.4-1.1	0.15-0.4
Cast irons	1.6-5.5	0.6-2.0
Copper alloys	1.4-3.3	0.5-1.2
High-Temp alloys	3.3-8.5	1.2-3.1
Mg alloys	0.4-0.6	0.15-0.2
Ni alloys	4.9-6.8	1.8-2.5
Refractory alloys	3.8-9.6	1.1-3.5
Stainless steels	3.0-5.2	1.1-1.9
Steels	2.7-9.3	1.0-3.4
Titanium alloys	3.0-4.1	1.1-1.5

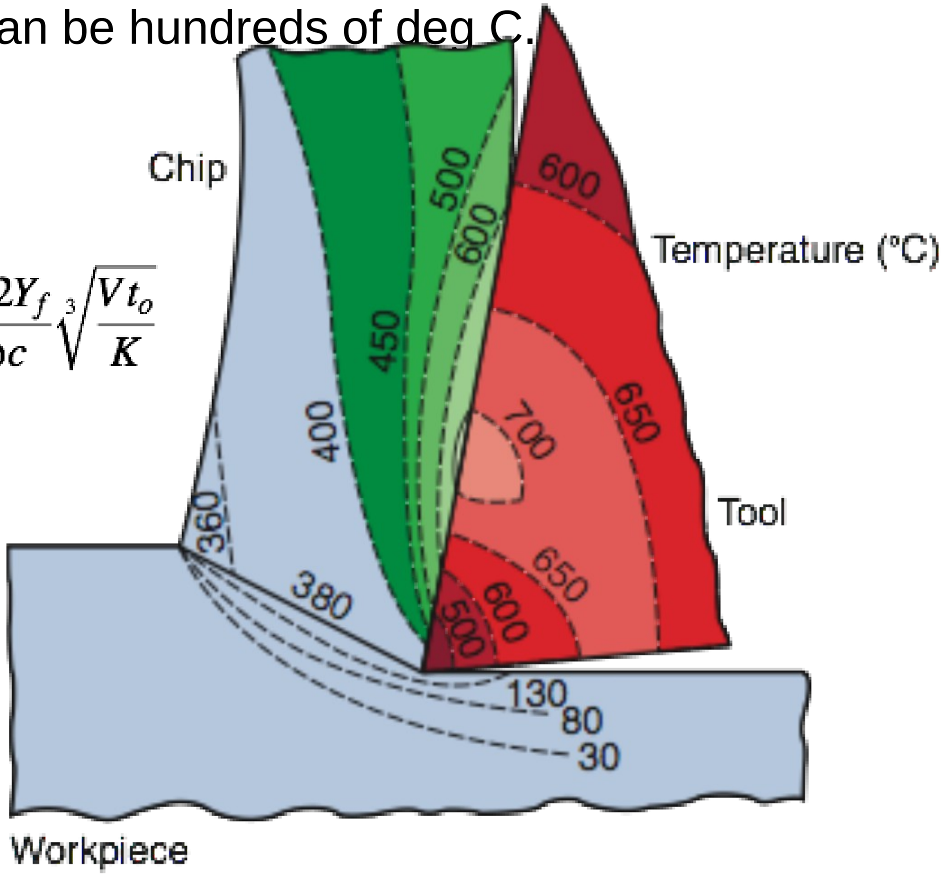
Manufacturing Processes for Engineering Materials, 5th Ed.

Material	Specific energy (N/mm^2)
Aluminum alloys	750
Stainless steels	4100
Titanium alloys	3550
Oriented strand board (OSB)	50
Particle board	25
Medium density fiberboard (MDF)	35
Plywood	35

https://hal.science/hal-04274766v1/document

- The power input to shear away the chips produces heat.
- The temperature increase at the tool-chip interface can be hundreds of deg C.

$$T = \frac{1.2Y_f}{\rho c}\sqrt[3]{\frac{Vt_o}{K}}$$



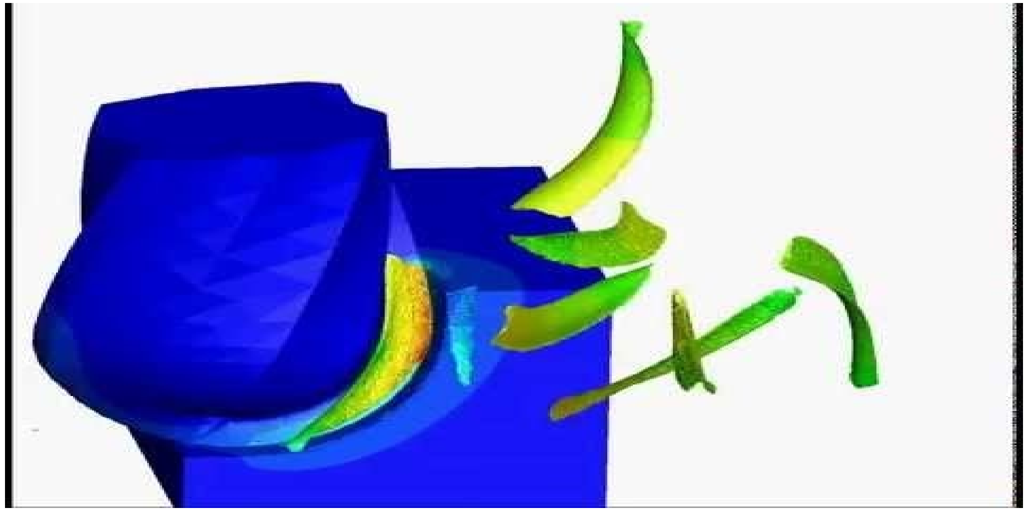
T - mean temp. at tool-chip interface

Y_f - flow stress

ρc - volumetric specific heat

V - cutting speed

t_o - chip thickness



Finite element modeling example for ball end milling

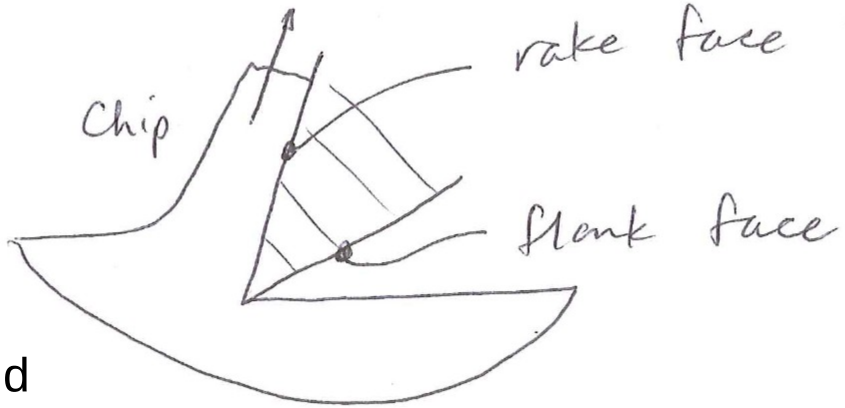
https://www.youtube.com/watch?v=gqfbDo-M6kA

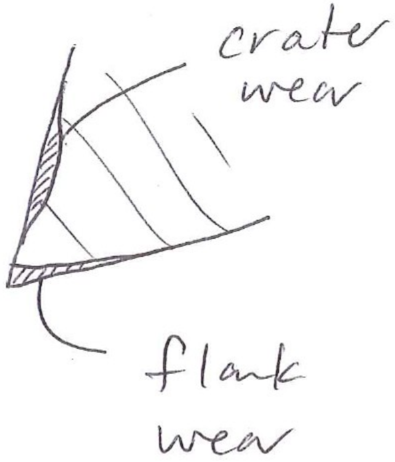
Machining introduction

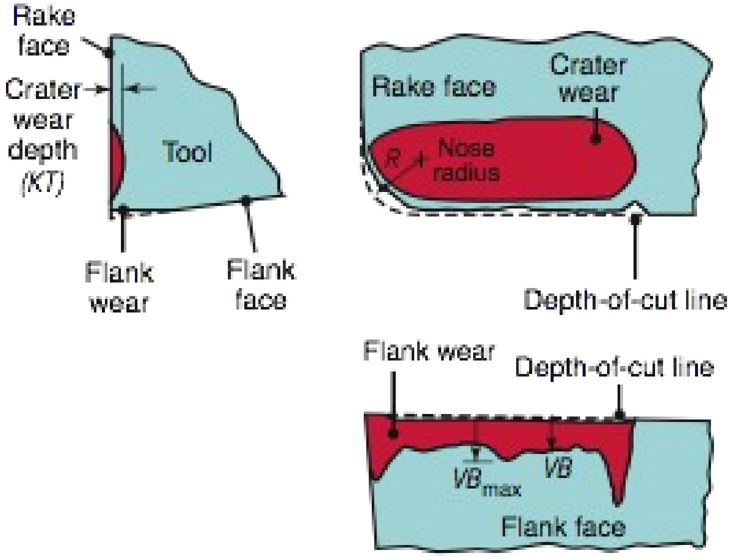
Because cutting tools are subjected to high:
- forces
- temperatures
- cutting speeds

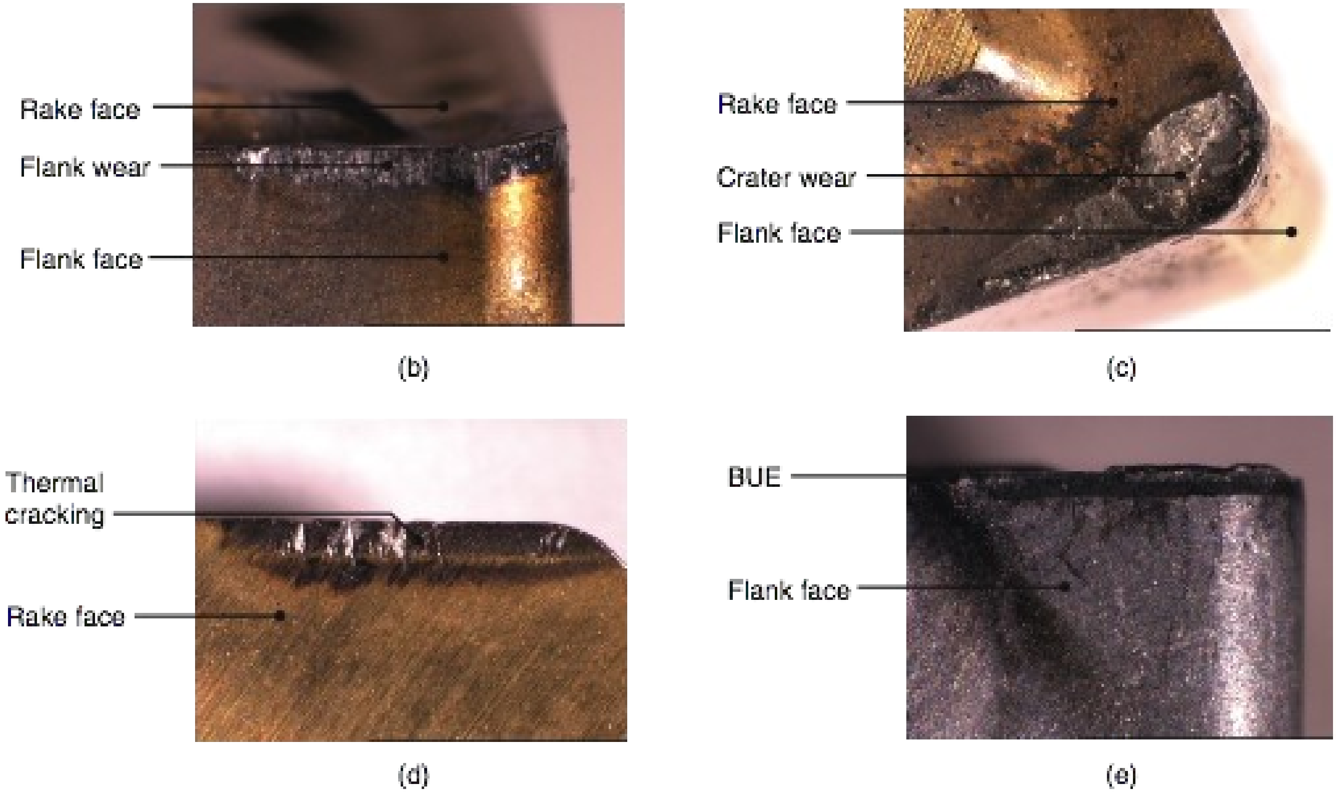
they wear over time.

- Wear features are observed on the rake and flank faces.
- Because temperature increases with cutting speed, tool wear also increases with cutting speed.









The wear rate also depends on:

- tool material
- tool coating
- cutting fluid.

HSS

Carbide

- High-speed steel (HSS) - carbon steel alloyed with molybdenum (M series) and tungsten (T series)

- Carbide - Tungsten carbide (WC) particles bonded together using cobalt.
  - steels, cast irons, abrasive nonferrous parts

  Also Titanium carbide bonded using nickel-molybdenum alloy.

  Carbide tools are available as solid tools or inserts that are clamped to tool holders. The inserts are replaced when they are worn.

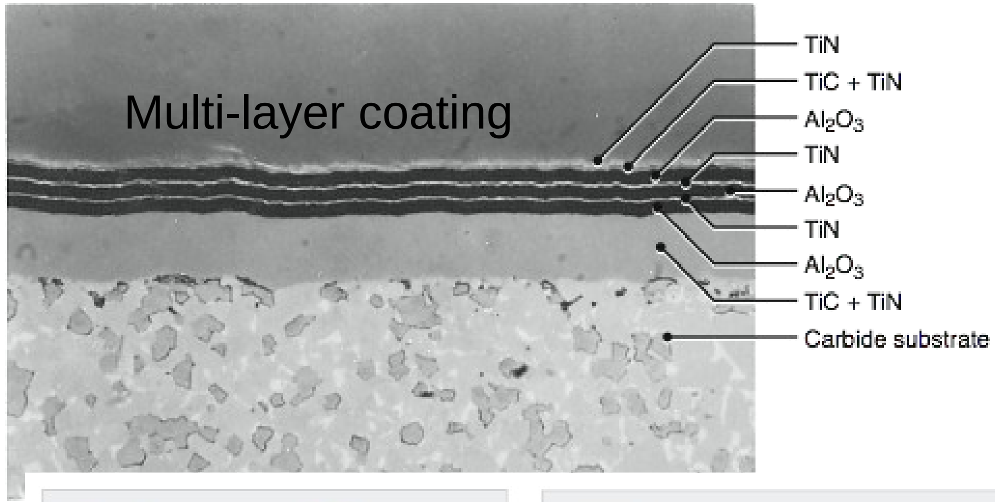
Coatings are applied to reduce friction and chemical reactivity.

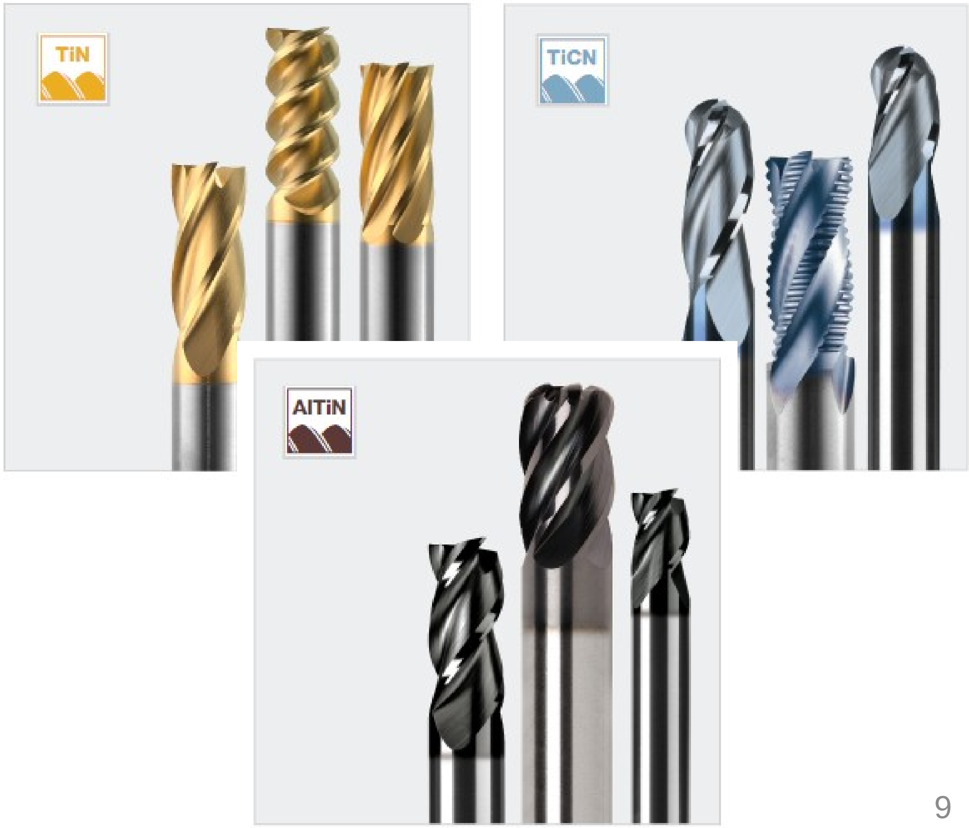
Titanium nitride – (TiN) Gold in color, low friction, high hardness

Titanium carbide – (TiC) Silver/gray in color, flank wear resistance on WC tools

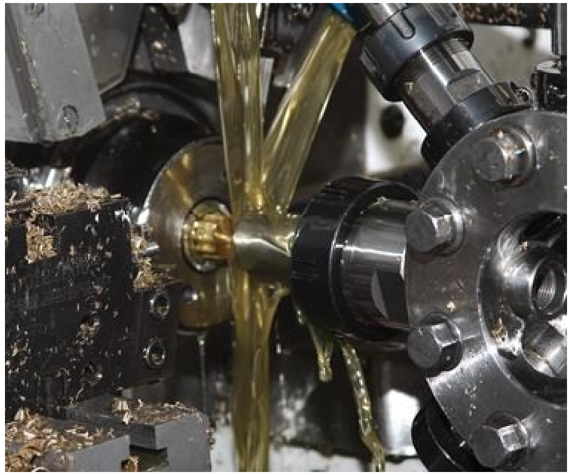
Titanium carbonitride – (TiCN) violet in color, harder and tougher than TiN, used for SSTs

Aluminum oxide – (Al_2O_3) ceramic is chemically inert and offer high wear protection at high temperatures





Cutting fluids are applied to
- cool chip/tool cutting zone and increase tool life (coolant)
- reduce friction and wear (lubricant)
- reduce force
- wash away chips
- protect new surface against corrosion

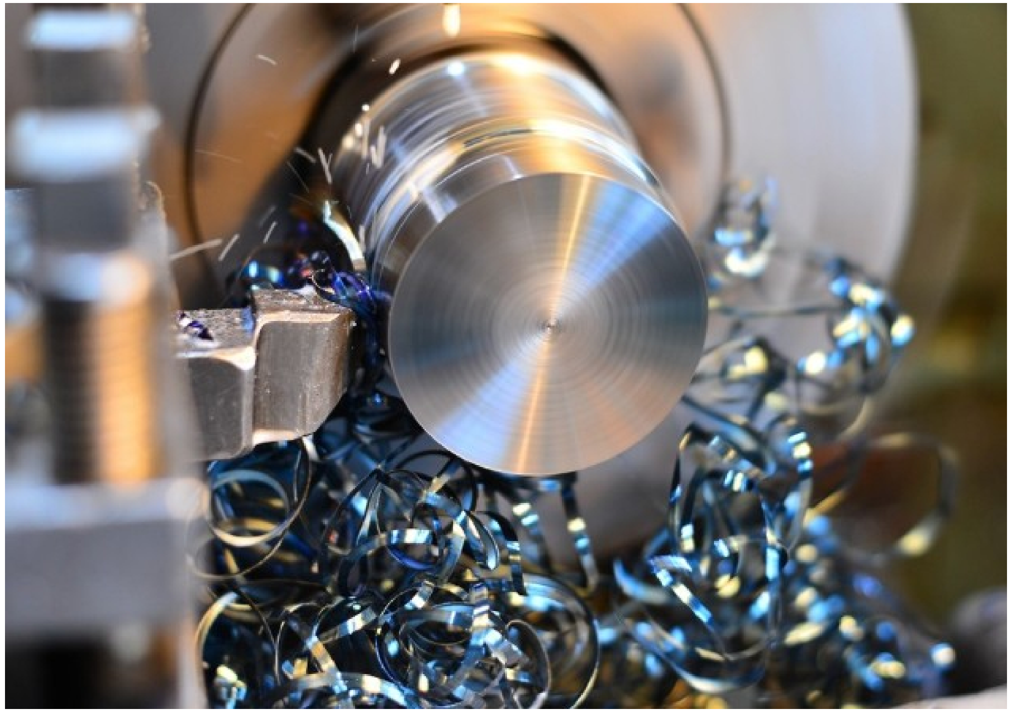


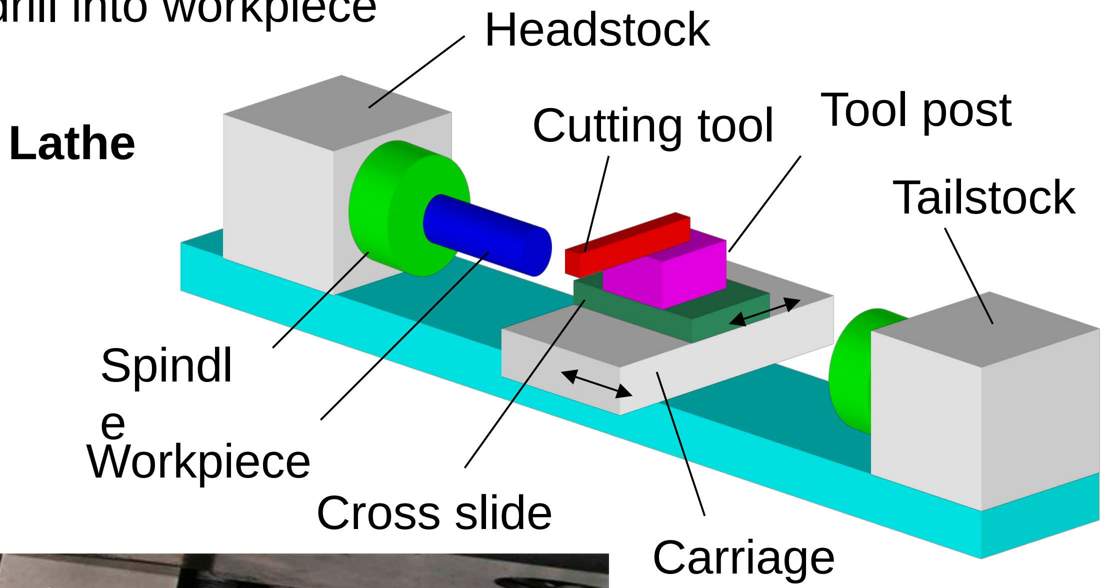
Types of cutting fluids include:
- oils
- emulsions
- semi synthetics
- synthetics

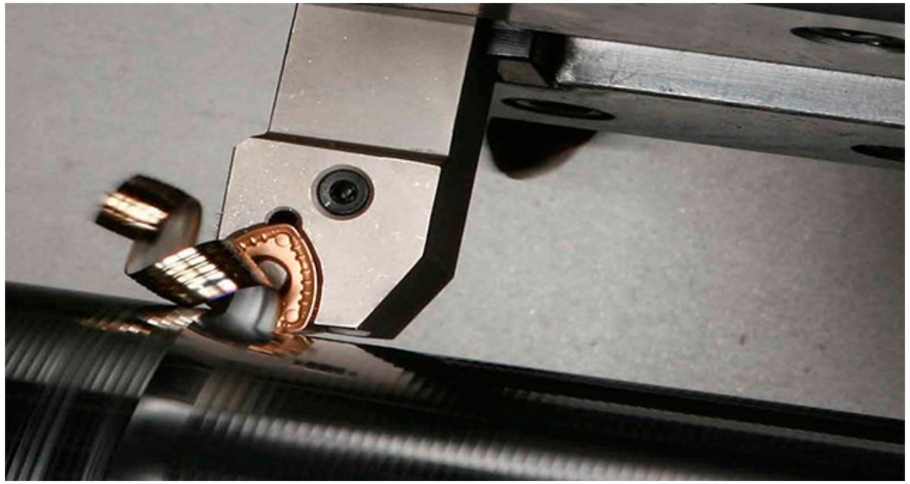
Flood coolant example
https://www.youtube.com/watch?v=rju3ly6nji0

We will introduce the primary machining operations

- **Turning – rotating workpiece, tool moves over workpiece to produce round shapes (lathe)**
- Milling – rotating tool, tool moves over workpiece to produce arbitrary shapes (mill)
- Drilling – lathe: rotating workpiece, drill forced into workpiece to produce hole at part center
- Drilling – mill: rotating drill, produces round holes by forcing drill into workpiece
- Tapping – cut threads inside hole



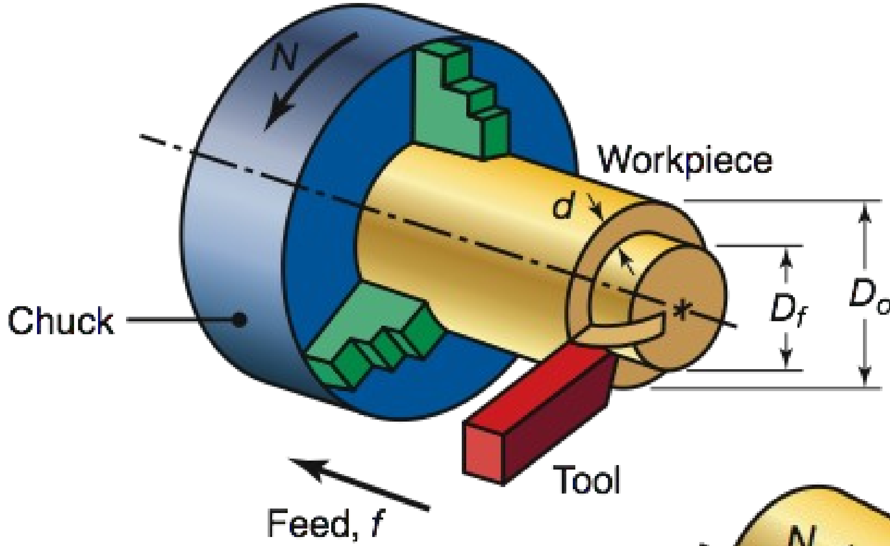




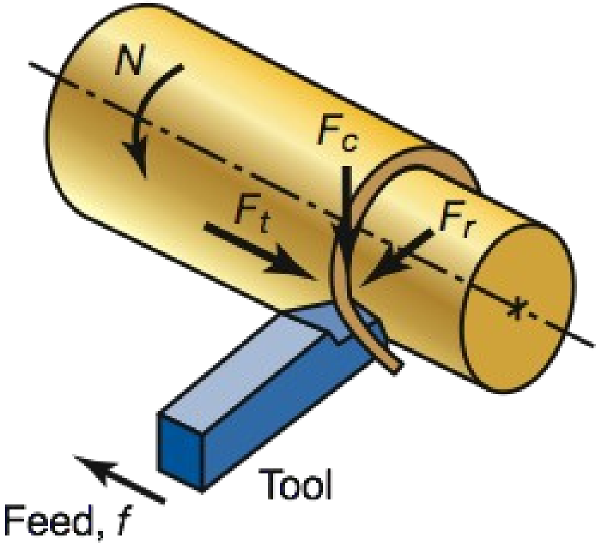
Turning video

https://www.youtube.com/watch?v=8EsAxOnzEms

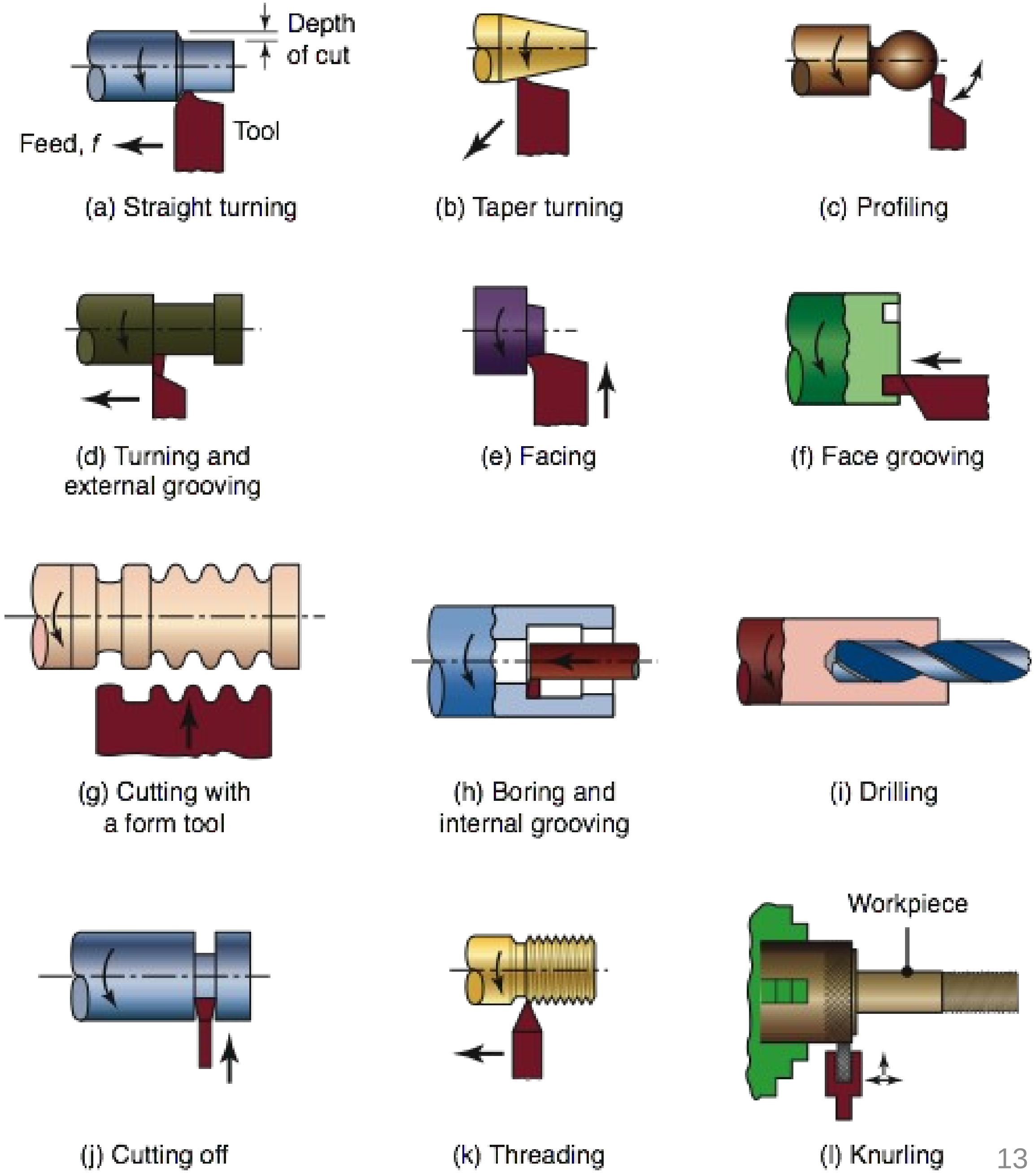
- **Turning – rotating workpiece, tool moves over workpiece to produce round shapes (lathe)**



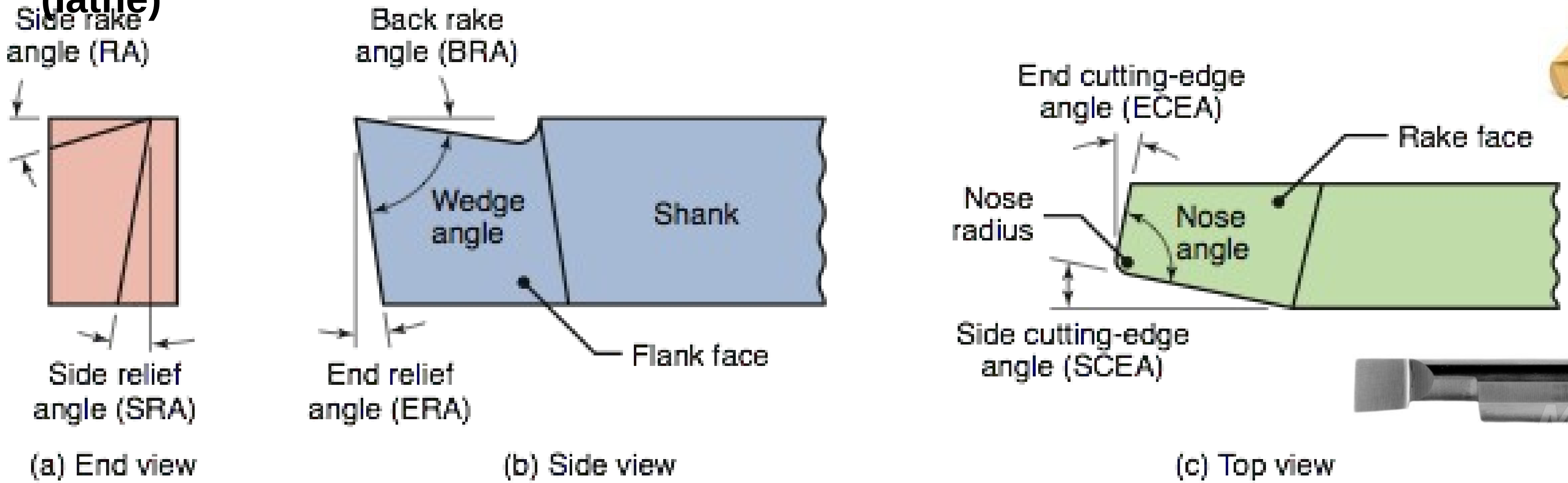
- Depth of cut, $d = (D_o - D_f)/2$
- Feed per revolution, f_r
- Spindle speed, N
- (Average) Diameter, $D = (D_o + D_f)/2$

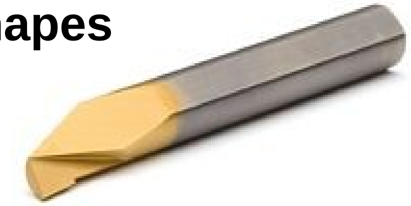


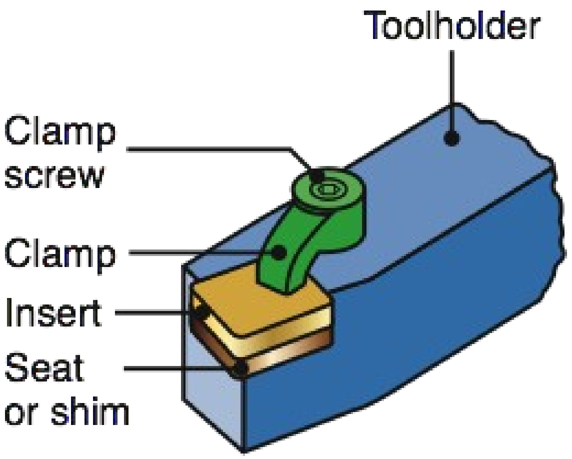
- Cutting speed, $V = \pi DN$
- Feed, $f = f_r N$

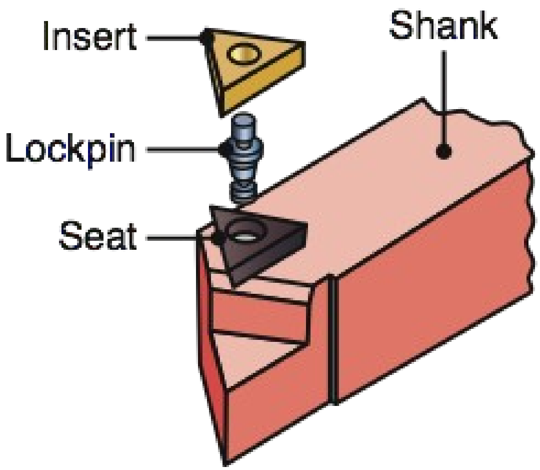


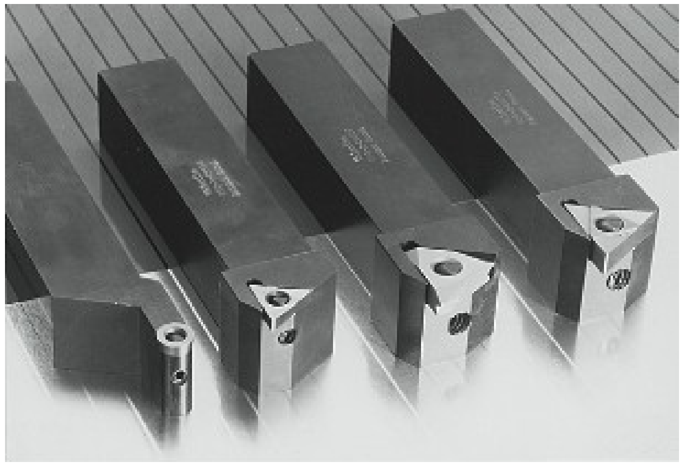
- **Turning – rotating workpiece, tool moves over workpiece to produce round shapes (lathe)**









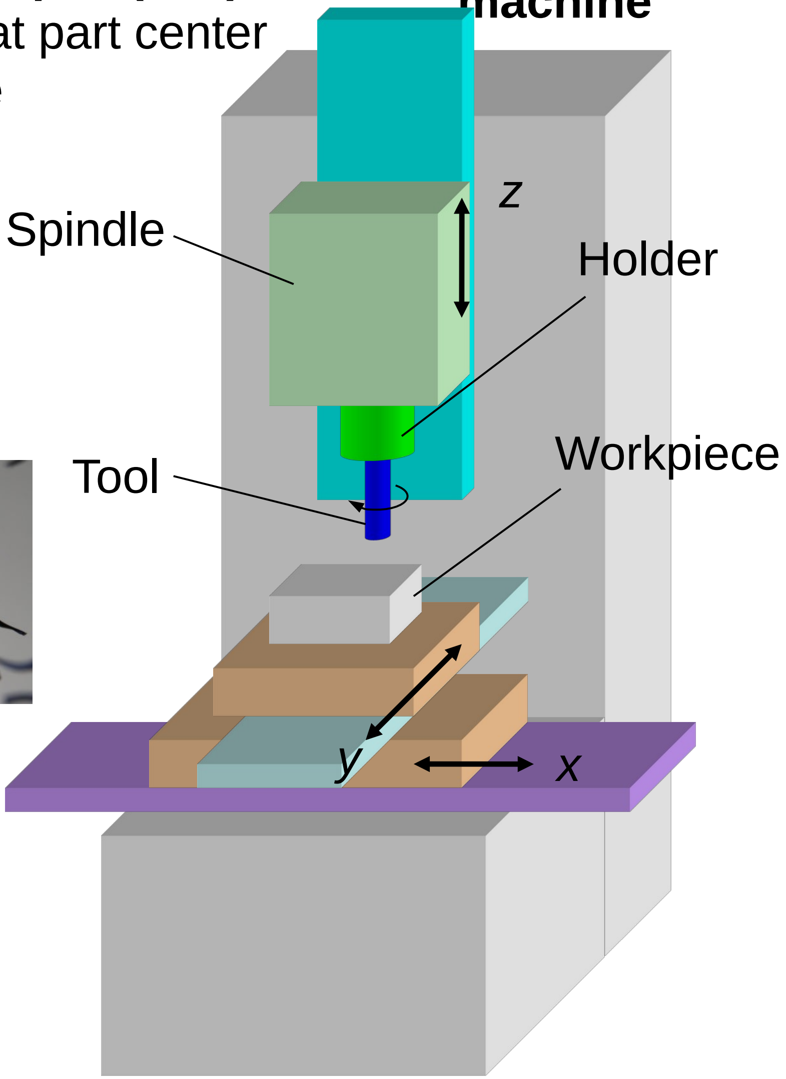


Machining introduction

We will introduce the primary machining operations
- Turning – rotating workpiece, tool moves over workpiece to produce round shapes (lathe)
- **Milling – rotating tool, tool moves over workpiece to produce arbitrary shapes (mill)**
- Drilling – lathe: rotating workpiece, drill forced into workpiece to produce hole at part center
- Drilling – mill: rotating drill, produces round holes by forcing drill into workpiece
- Tapping – cut threads inside hole

Vertical spindle milling machine
- tool is clamped in a holder which is attached to the rotating spindle
- tool-holder-spindle is moved relative to the workpiece using the three orthogonal axes to remove material

3-axis vertical milling machine

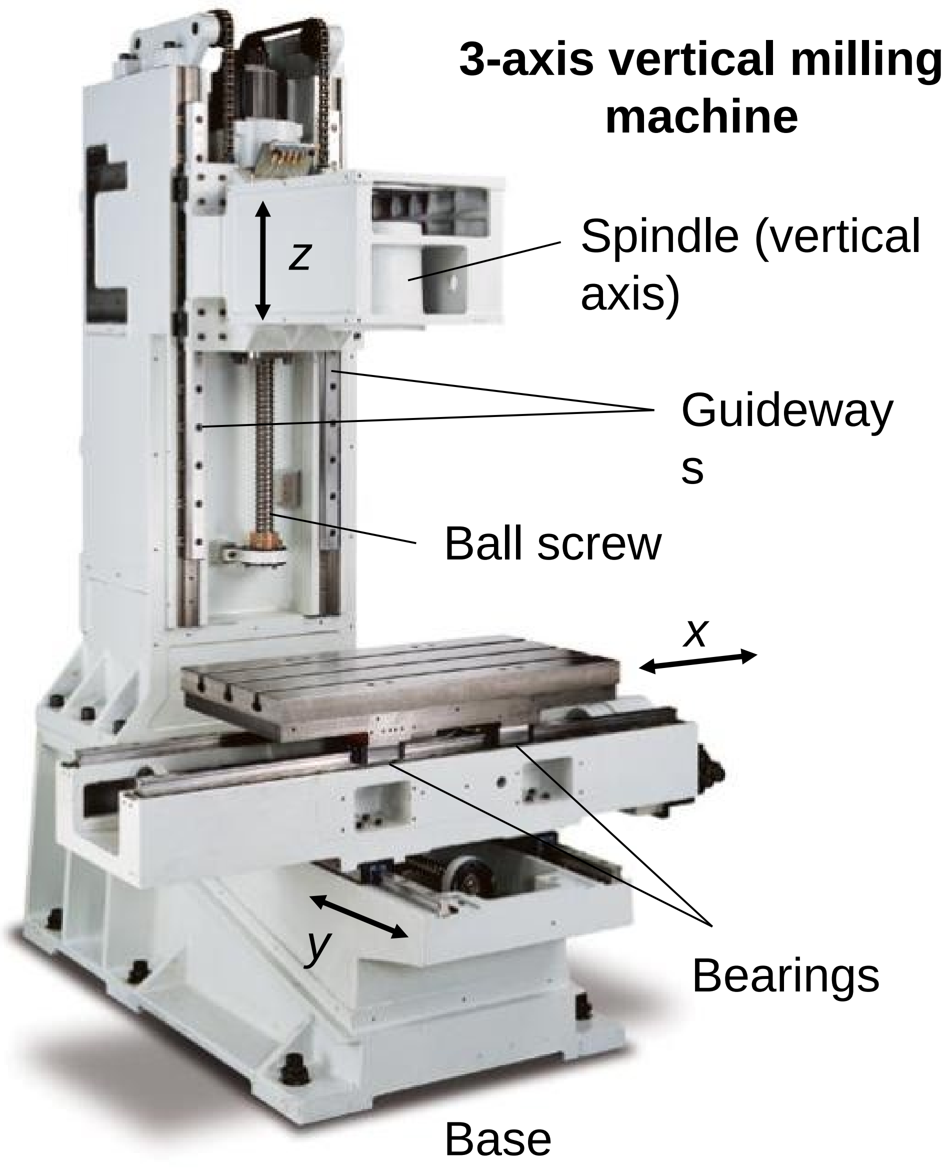


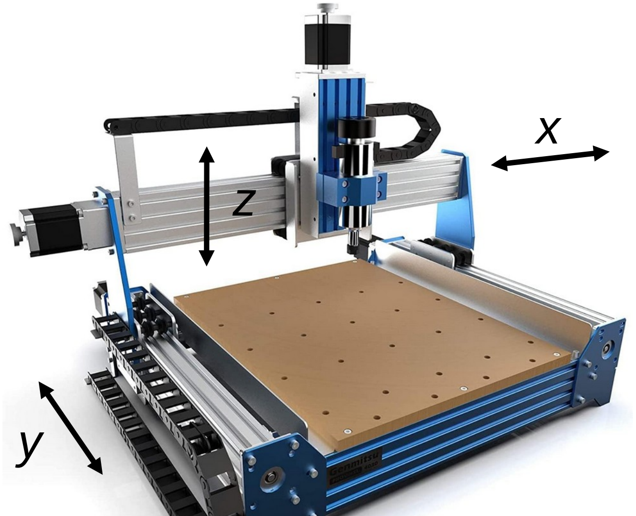
Milling video
https://www.youtube.com/watch?v=AxHexqN0Hr0

Machining Dynamics: Frequency Response to Improved Productivity, 2nd Ed.

Machining introduction

3-axis vertical milling machine

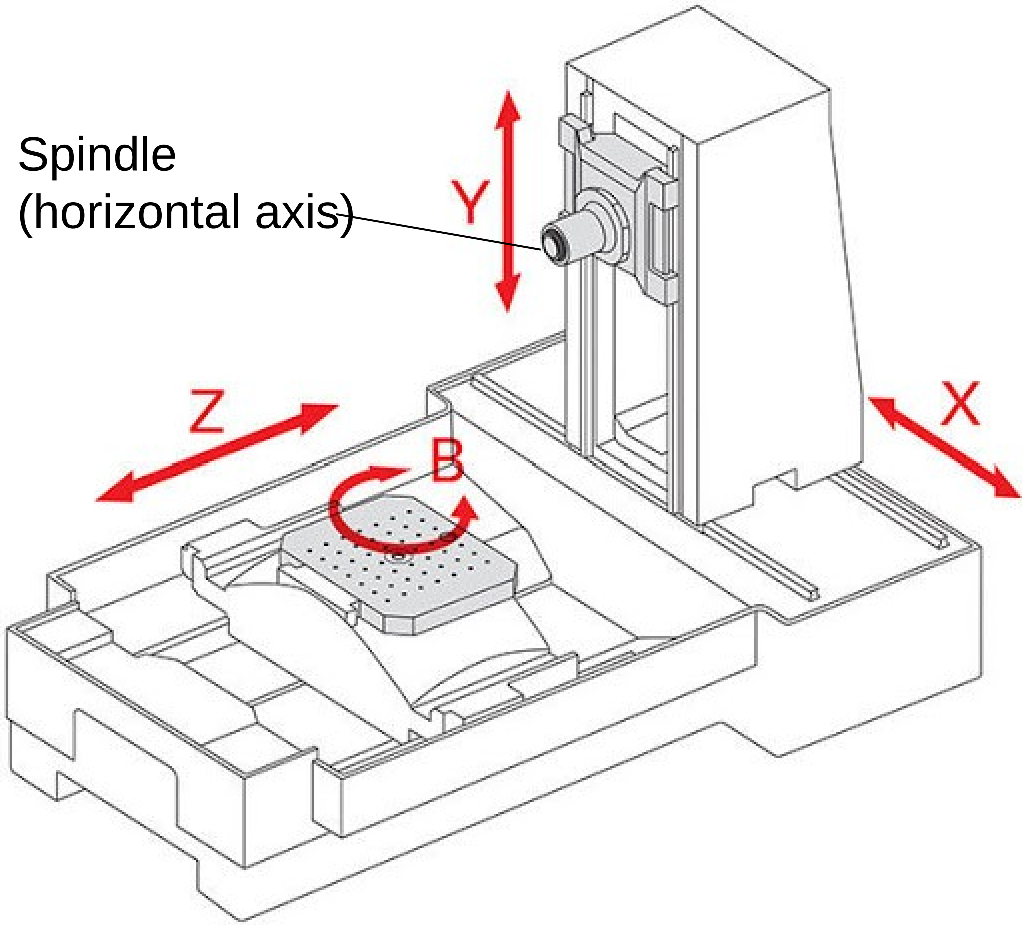




3-axis router

https://www.youtube.com/watch?v=U99asuDT97I

Machining introduction



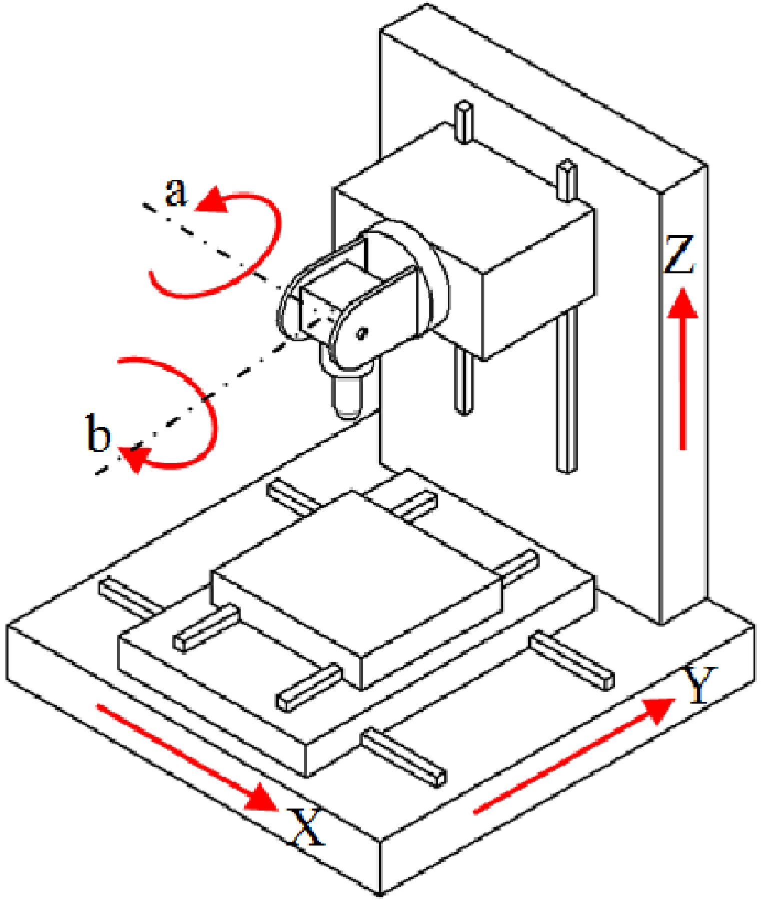
4-axis horizontal milling machine

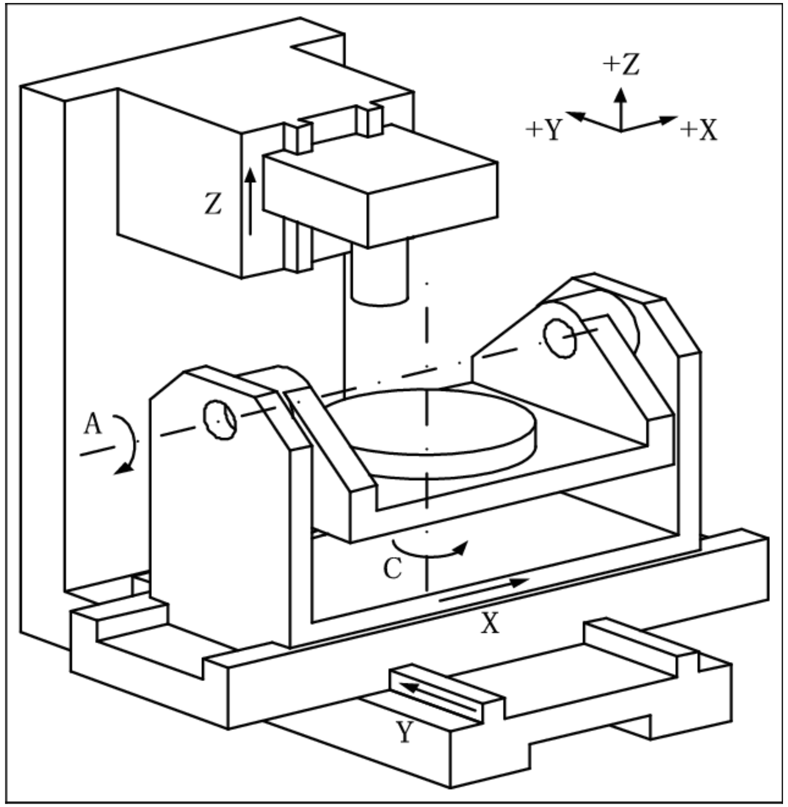
5-axis milling machine

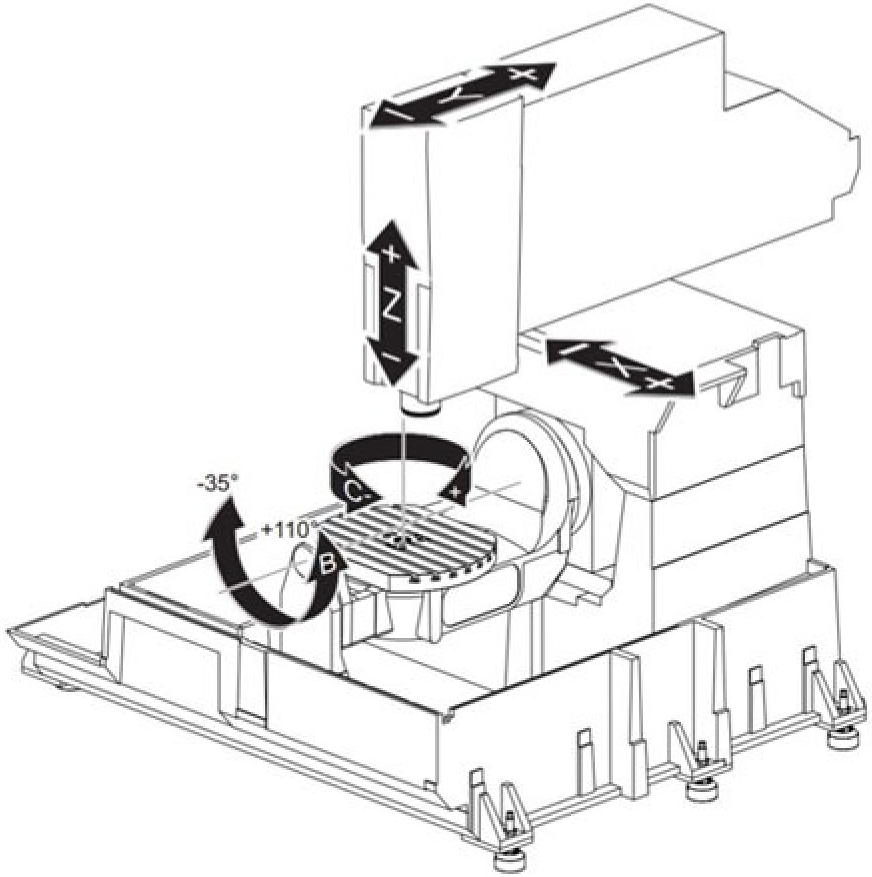
A (rotation about x) – B (rotation about y)

A (rotation about x) – C (rotation about z)

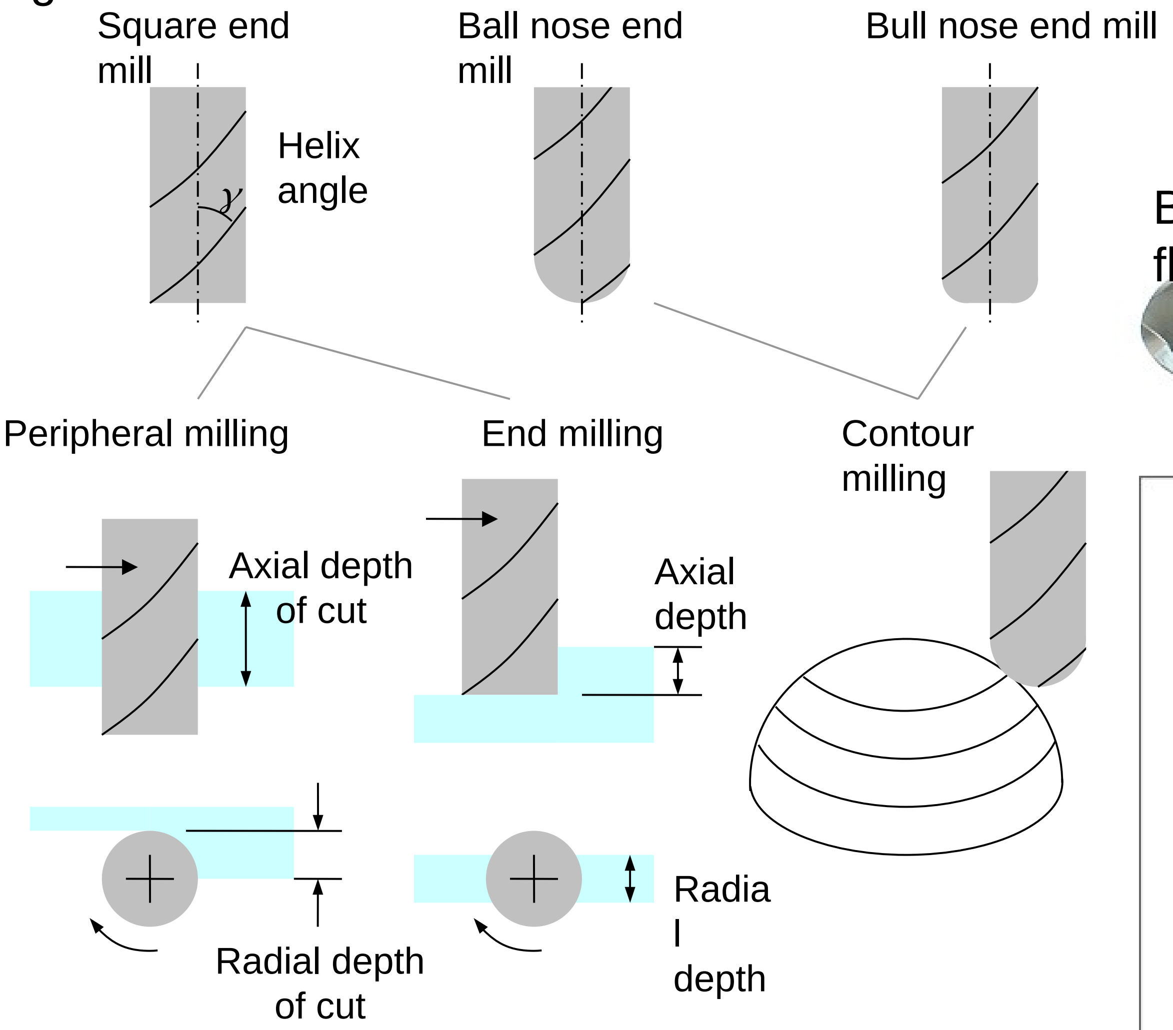
B (rotation about y) – C (rotation about z)

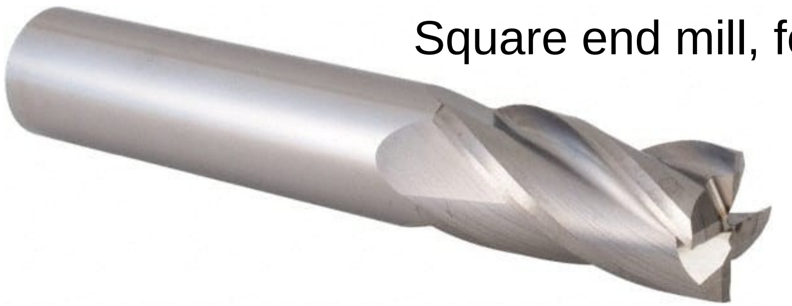






Example milling applications and tool geometries

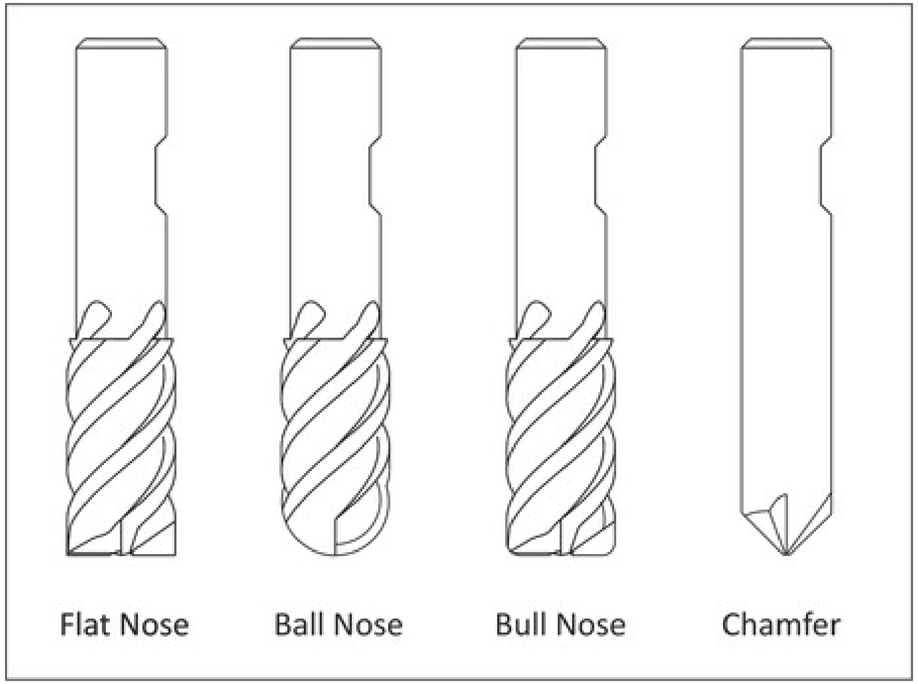




Square end mill, four flutes (teeth)

Ball nose end mill, two flutes

Ball nose end mill, two flutes, indexable



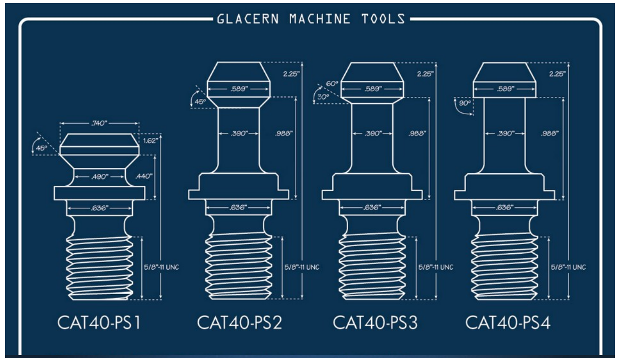
Face mill, indexable

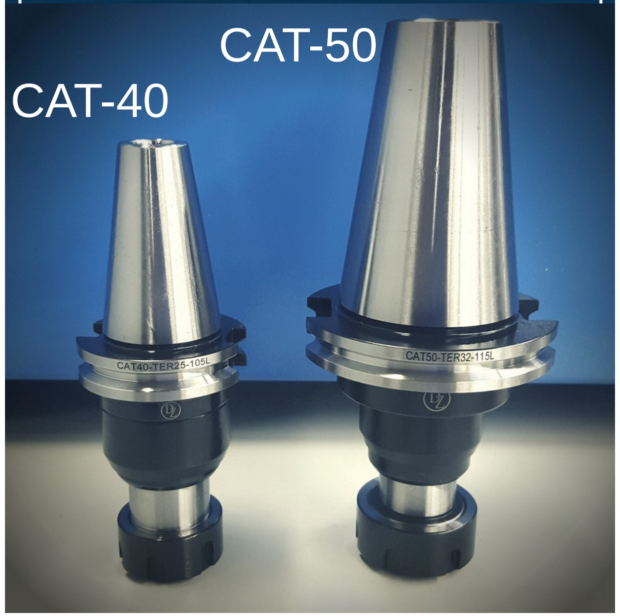
Machining Dynamics: Frequency Response to Improved Productivity, 2nd Ed.

The end mill must be clamped in the holder and the holder in the spindle

- requires two interfaces
  - tool to holder (collet, thermal shrink fit, hydraulic, set screw, …)
  - holder to spindle (CAT40, HSK63A, …)

Retention knobs/pull studs



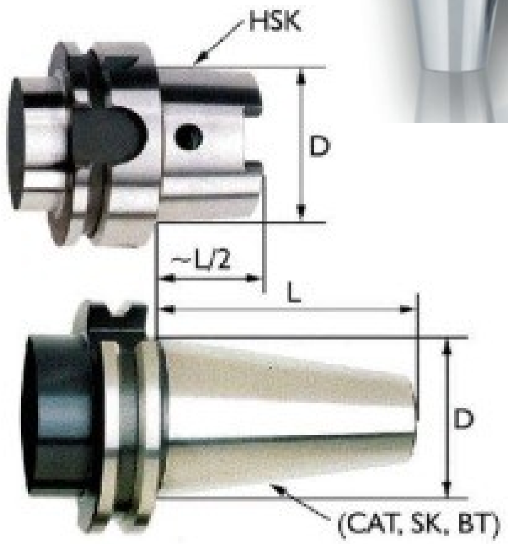


CAT40 spindle connection

ER collet holder

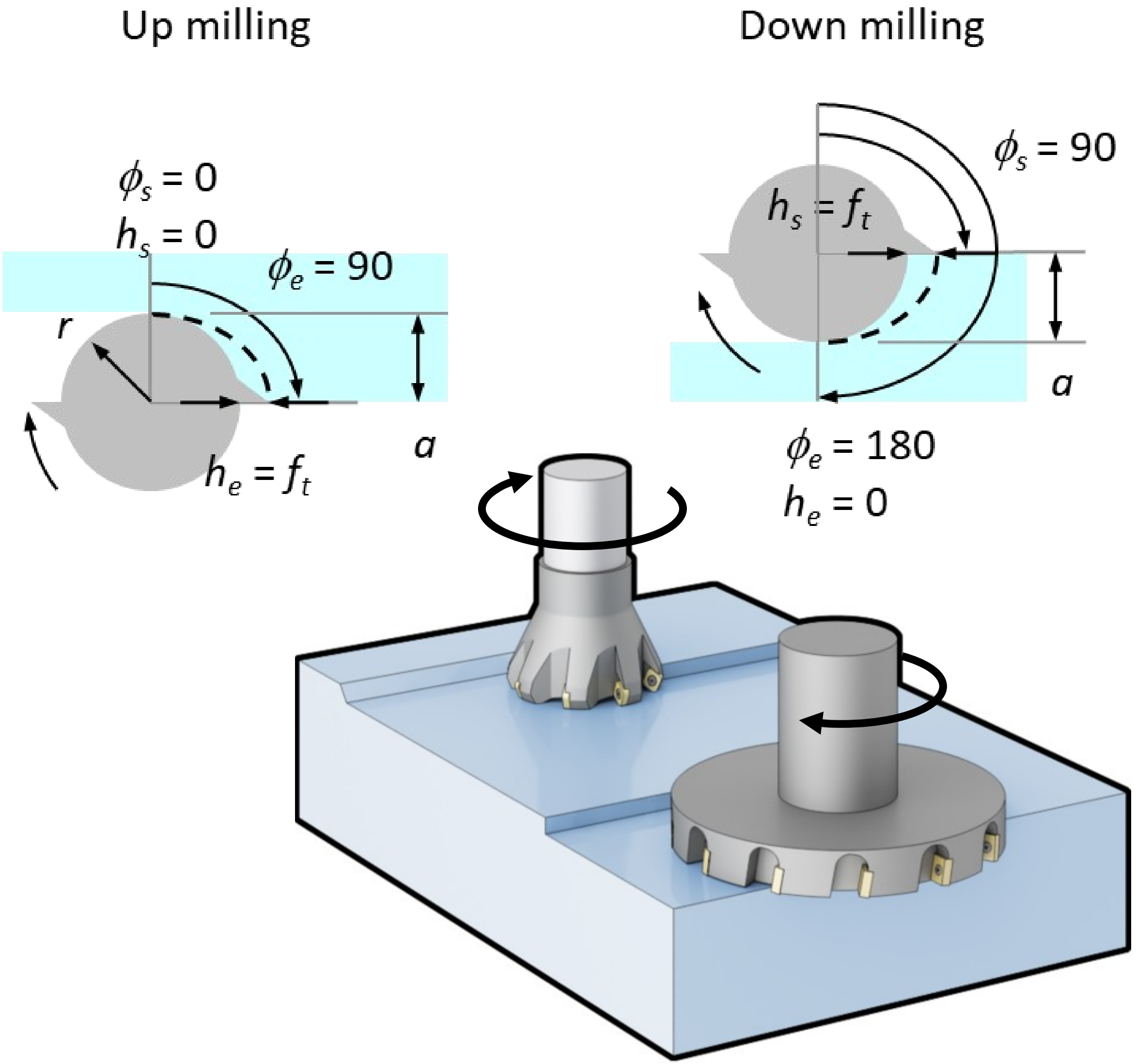
Thermal shrink fit

Weldon/ set screw holder



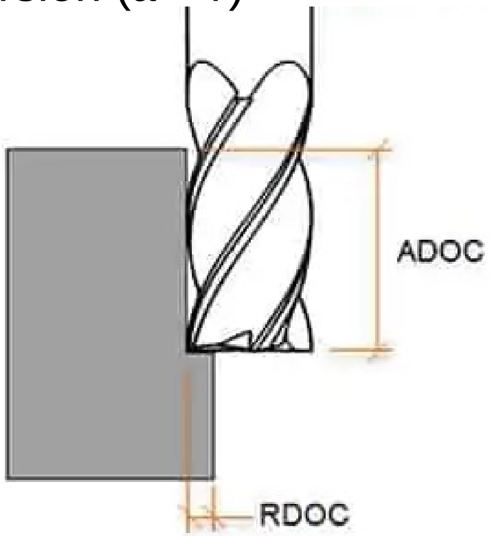
HSK

Chip thickness variation for **up (conventional)** and **down (climb)** milling for 50% radial immersion ($a = r$)

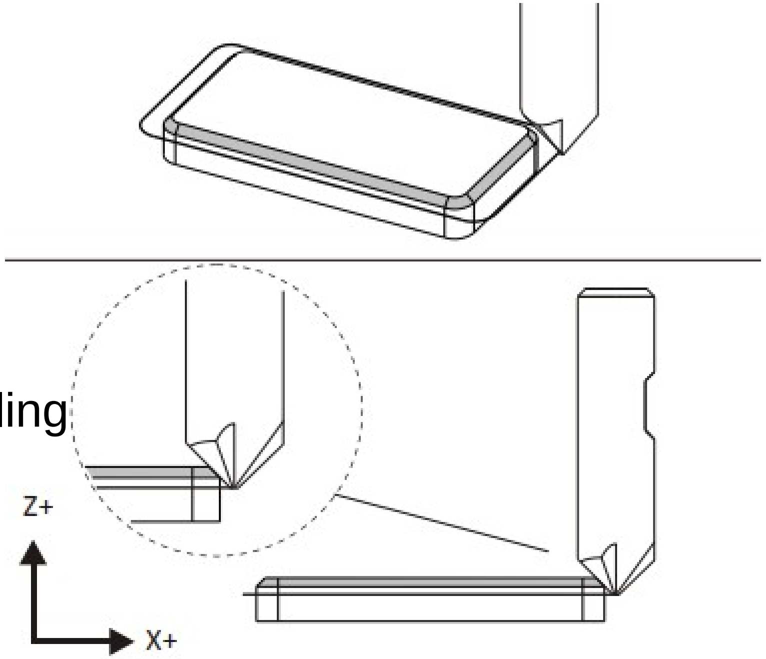


- feed per tooth (chip load), f_t
- spindle speed, N
- diameter, D
- number of teeth, m

- cutting speed, $V = \pi DN$
- feed, $f = f_t mN$



Chamfer milling



[Machining Dynamics: Frequency Response to Improved Productivity](), 2nd Ed.

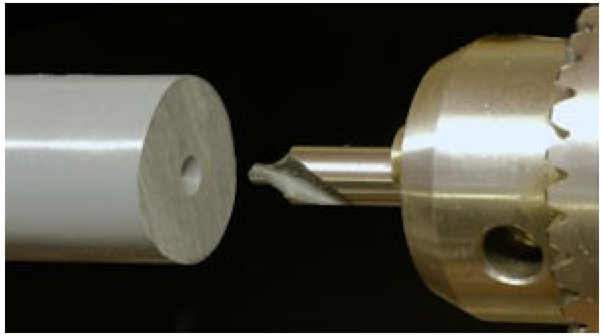
Face milling video

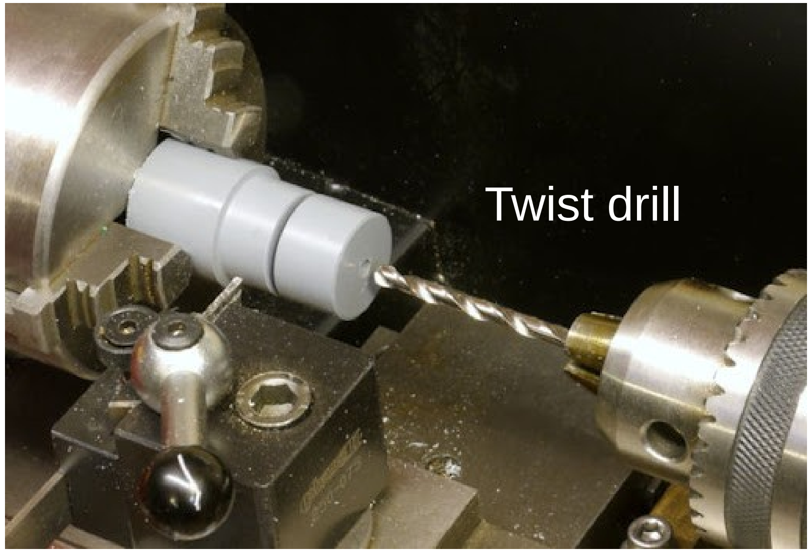
https://www.youtube.com/watch?v=9OsNUi_o6C4

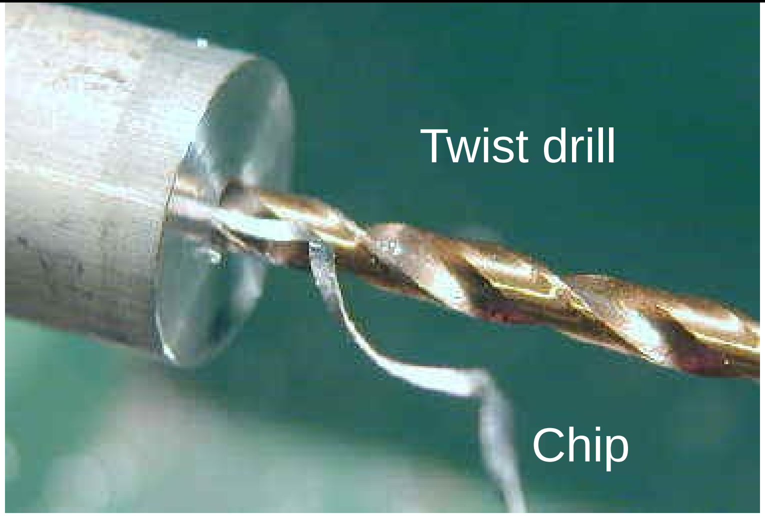
Machining introduction

We will introduce the primary machining operations

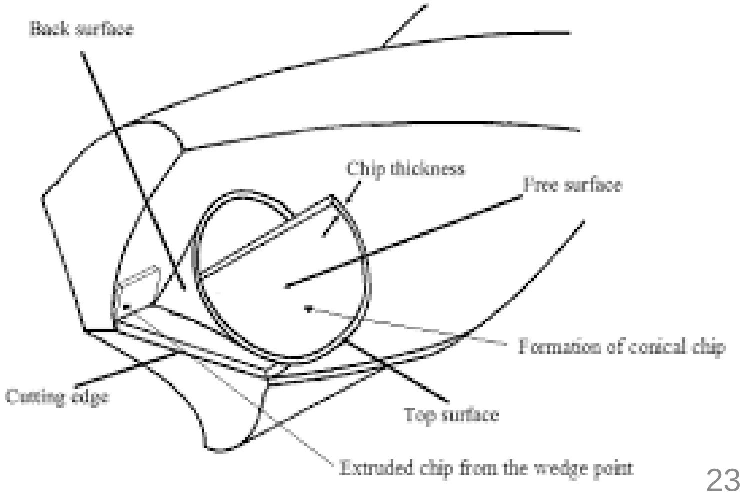
- Turning – rotating workpiece, tool moves over workpiece to produce round shapes (lathe
- Milling – rotating tool, tool moves over workpiece to produce arbitrary shapes (mill)
- **Drilling – lathe: rotating workpiece, drill forced into workpiece to produce hole at p**
- Drilling – mill: rotating drill, produces round holes by forcing drill into workpiece
- Tapping – cut threads inside hole





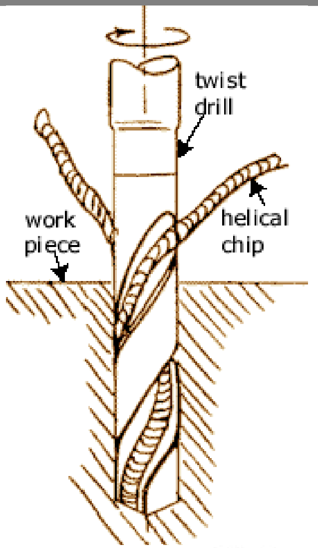


Spot drill (center drill) video
https://www.youtube.com/watch?v=O9uNy76nH8M

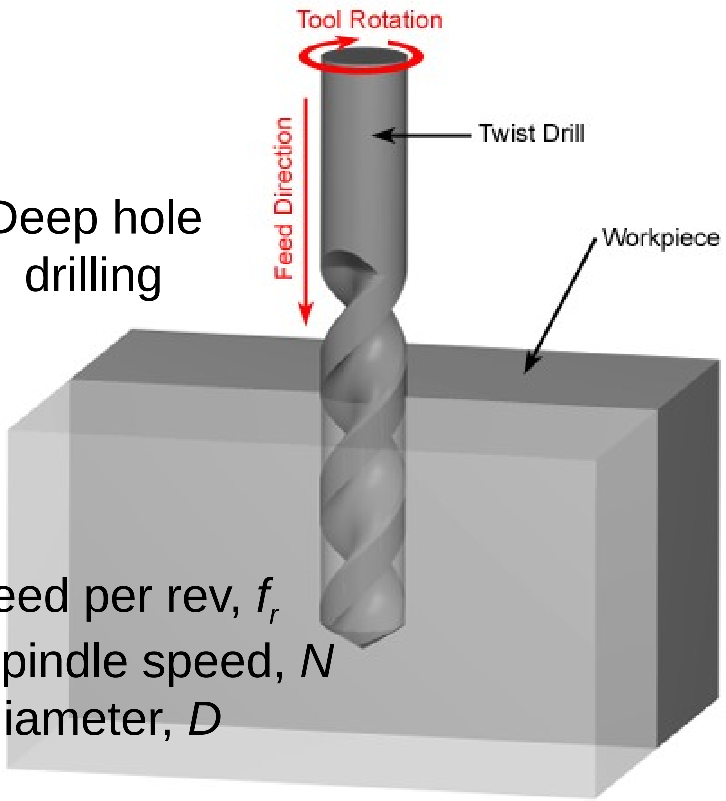


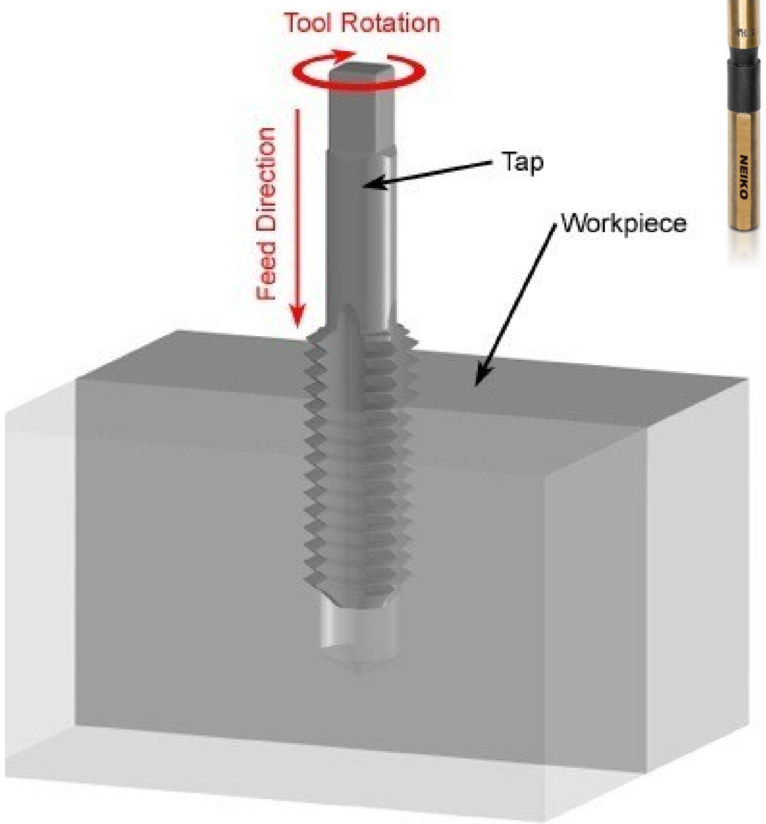
We will introduce the primary machining operations

- Turning – rotating workpiece, tool moves over workpiece to produce round shapes (lathe)
- Milling – rotating tool, tool moves over workpiece to produce arbitrary shapes (mill)
- Drilling – lathe: rotating workpiece, drill forced into workpiece to produce hole at part center
- **Drilling – mill: rotating drill, produces round holes by forcing drill into workpiece**
- **Tapping – cut threads inside hole**

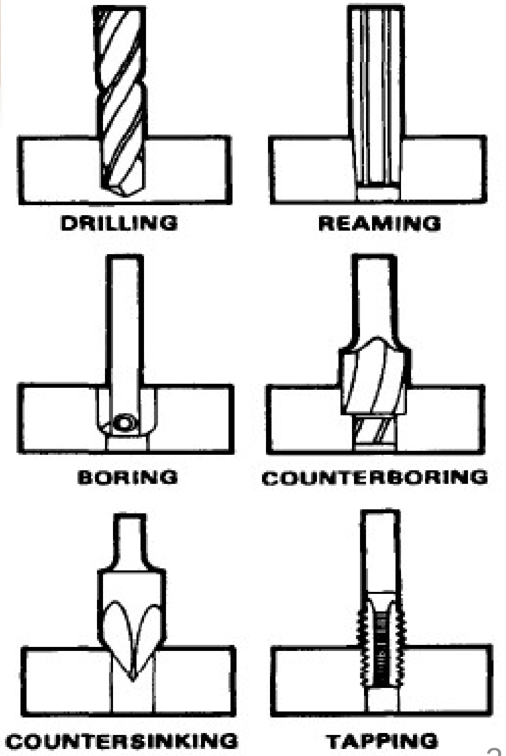


Deep hole drilling





Chamfer drills



- feed per rev, f_r
- spindle speed, N
- diameter, D

- cutting speed, $V = \pi D N$

Drilling and tapping video

https://www.youtube.com/watch?v=om6GQKfoS1g

Machining introduction

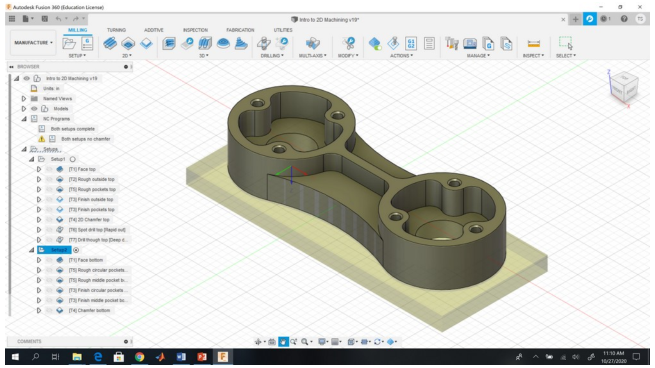
Machining operations

- chip formation by shearing with a sharp cutting edge
  - cutting force, power
  - temperature increase, tool wear
  - tool material, coatings, lubricant/coolant
- turning: straight turning, profiling, facing, drilling
  - parameters: depth of cut, feed per revolution, spindle speed, feed, cutting speed
- drilling: spot drilling, deep hole drilling
  - tools: spot/center drill, twist drill
  - parameters: drill diameter, feed per revolution, spindle speed, feed, cutting speed
- milling: face, peripheral, chamfer, up/down (conventional/climb)
  - milling cutters: face mill, square (flat nose) end mill, bull nose end mill, ball nose end mill
  - parameters: spindle speed, feed per tooth, feed, cutting speed, axial depth (stepdown), radial depth (stepover)

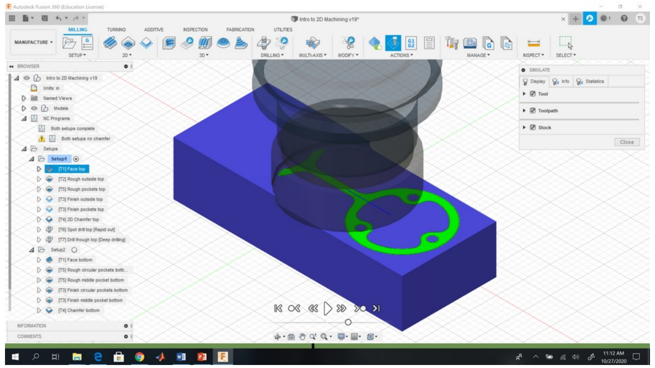
CNC and CAM introduction

Tony Schmitz

Computer numerically controlled (CNC) machining



- the part is designed using computer software to provide a digital model of the desired geometry – computer aided design (CAD)



- the CAD model is used in computer aided manufacturing (CAM) software to generate the instructions, or toolpath, for the CNC machine to produce the part

- the CNC part program, that includes the toolpath and other machine instructions, is uploaded to the CNC controller on the machine tool
- the part is machined and inspected

Computer numerically controlled (CNC) machining

https://www.youtube.com/watch?v=hMK7g_PpCv8

Key CAM software considerations

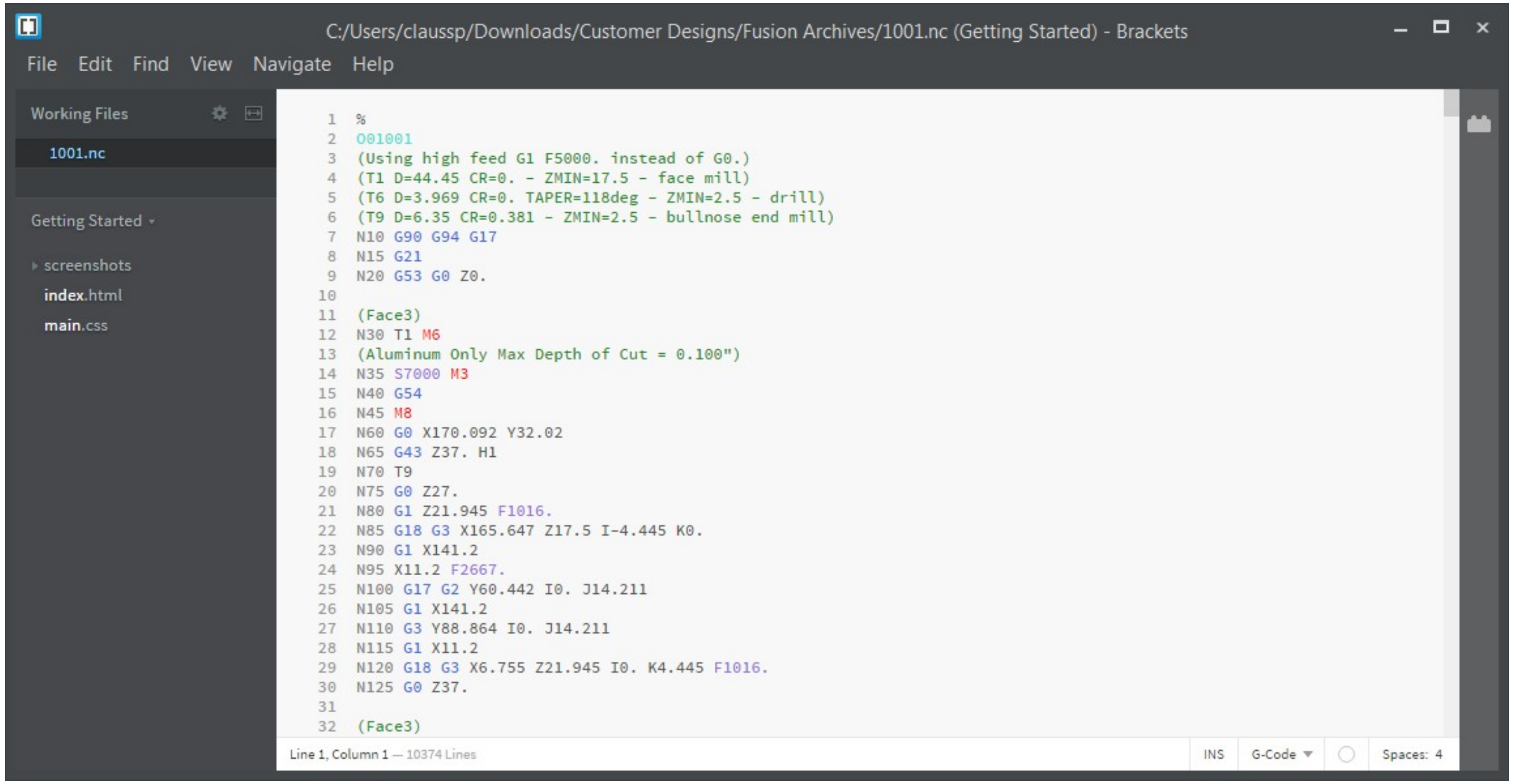
- the instructions are provided using M and G codes – computer code that is interpreted by the machine tool controller

Example **G01** (linear interpolation)**:**
G01 X1 Y1 F20 T01 M03 S500

single line gives the machine a series of instructions to prepare for a milling operation:

G01 – Perform a linear feed move
X1/Y1 – Move to these X and Y coordinates
F20 – Move at a feed rate of 20
T01 – Use Tool 1 to perform the operation
M03 – Turn the spindle on
S500 – Set a spindle speed of 500

The line could be considered a block.
The end of block (EOB) is often marked



The purpose of CAM is to use your part geometry and selected tools (face mill, end mill, drill, etc.) to produce that geometry from the stock model (rectangular block, forging, casting, additively manufactured preform).

CAM output is a part program. The exact format of the program depends on your machine's controller (Fanuc, Siemens, Haas). It must be post-processed for the selected controller ("post the program to machine x").

https://www.autodesk.com/products/fusion-360/blog/cnc-programming-fundamentals-g-code-2020-update/

https://en.wikipedia.org/wiki/G-code

Key CAM software considerations

- the instructions are provided using M and G codes – computer code that is interpreted by the machine tool controller

M codes are machine codes that might differ between CNC machines. These codes control functions on your CNC machine such as coolant and spindle directions. Some of the most common M codes include:

Code	Meaning
M0	Program stop. Press Cycle Start button to continue.
M1	Optional stop. Only executed if Op Stop switch on the CNC control is turned ON.
M2	End of program.
M3	Spindle on Clockwise.
M4	Spindle on Counterclockwise.
M5	Spindle stop.
M6	Change tool.
M8	Coolant on.
M9	Coolant off.
M30	End program and press Cycle Start to run it again.

https://www.autodesk.com/products/fusion-360/blog/cnc-programming-fundamentals-g-code-2020-update/

Key CAM software considerations

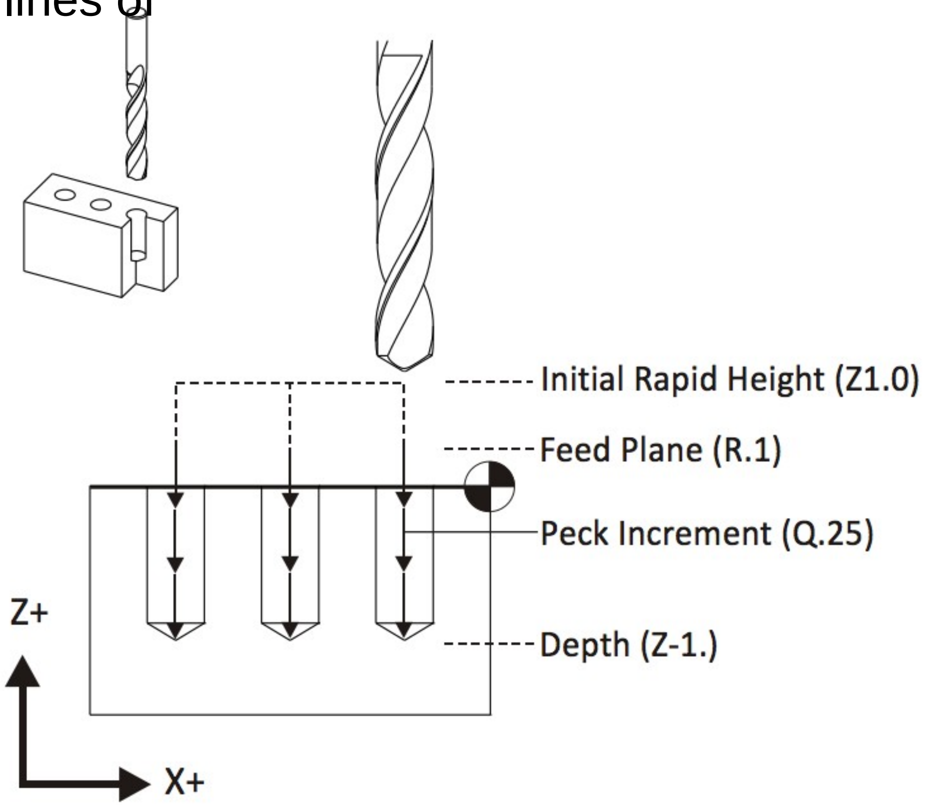
- the instructions are provided using M and G codes – computer code that is interpreted by the machine tool controller

Canned cycles allow you to perform a complicated action in only a few lines of code without typing each individual instruction.

Example **G83**:
Create a hole by peck drilling using only two lines of code. Represents over 20 lines of code.

G98 is a tool return to the initial position.

Canned Cycle	Equivalent Motion: Expanded Code
N70 G98 G83 X1. Y1. Z-1.04 R0.06 Q0.15 P0 F9.	N70 Z0.06
N75 G80	N75 Z0.04
	N80 G01 Z-0.19 F9.
	N85 G00 Z0.06
	N90 Z-0.11
	N95 G01 Z-0.34
	N100 G00 Z0.06
	N105 Z-0.26
	N110 G01 Z-0.49.
	N115 G00 Z0.06
	N120 Z-0.41
	N125 G01 Z-0.64.
	N130 G00 Z0.06
	N135 Z-0.56
	N140 G01 Z-0.79
	N145 G00 Z0.06
	N150 Z-0.71
	N155 G01 Z-0.94.
	N160 G00 Z0.06
	N165 Z-0.86
	N170 G01 Z-1.04.
	N175 G00 Z0.25



https://www.autodesk.com/products/fusion-360/blog/cnc-programming-fundamentals-g-code-2020-update/

CNC and CAM introduction

Typical sequence of activities for a CNC part program

1. Start the CNC program

2. Load the required tool
3. Turn the spindle on

4. Turn the coolant on
5. Move to position above a part

6. Start the machining process

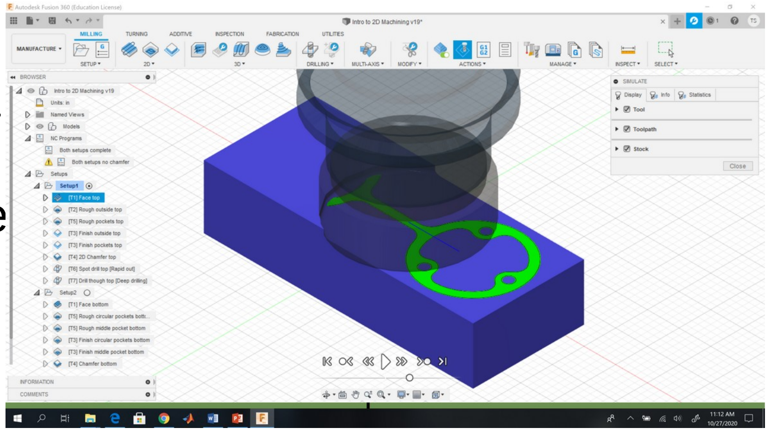
7. Turn the coolant off
8. Turn the spindle off
9. Move away from the part to a safe location

…

10. End the CNC program

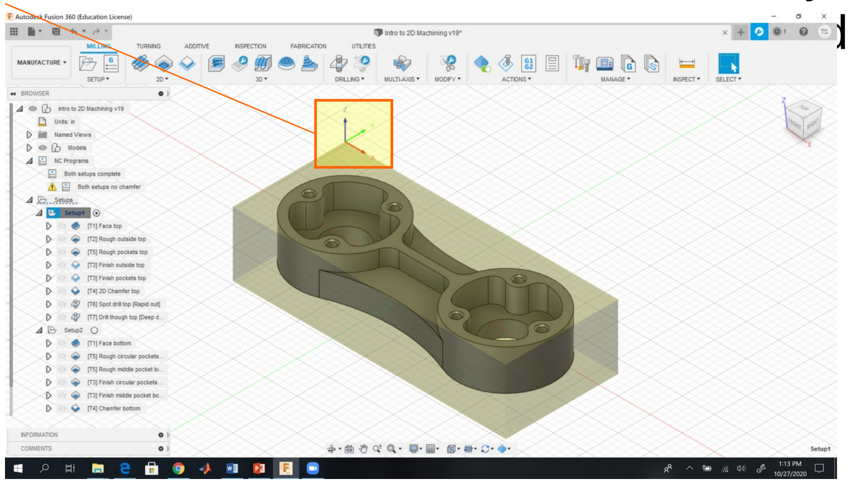
Block	Description	Purpose
%	Start of program.	Start Program
O0001 (PROJECT1)	Program number (Program Name).	
(T1 0.25 END MILL)	Tool description for operator.	
N1 G17 G20 G40 G49 G80 G90	Safety block to ensure machine is in safe mode.	
N2 T1 M6	Load Tool #1.	Change Tool
N3 S9200 M3	Spindle Speed 9200 RPM, On CW.	
N4 G54	Use Fixture Offset #1.	Move To Position
N5 M8	Coolant On.	
N6 G00 X-0.025 Y-0.275	Rapid above part.	
N7 G43 Z1. H1	Rapid to safe plane, use Tool Length Offset #1.	
N8 Z0.1	Rapid to feed plane.	
N9 G01 Z-0.1 F18.	Line move to cutting depth at 18 IPM.	
N10 G41 Y0.1 D1 F36.	CDC Left, Lead in line, Dia. Offset #1, 36 IPM.	Machine Contour
N11 Y2.025	Line move.	
N12 X2.025	Line move.	
N13 Y-0.025	Line move.	
N14 X-0.025	Line move.	
N15 G40 X-0.4	Turn CDC off with lead-out move.	
N16 G00 Z1.	Rapid to safe plane.	
N17 M5	Spindle Off.	Change Tool
N18 M9	Coolant Off.	
(T2 0.25 DRILL)	Tool description for operator.	
N19 T2 M6	Load Tool #2.	
N20 S3820 M3	Spindle Speed 3820 RPM, On CW.	
N21 M8	Coolant On.	Move To Position
N22 X1. Y1.	Rapid above hole.	
N23 G43 Z1. H2	Rapid to safe plane, use Tool Length Offset 2.	
N24 Z0.25	Rapid to feed plane.	
N25 G98 G81 Z-0.325 R0.1 F12.	Drill hole (canned) cycle, Depth Z-.325, F12.	Drill Hole
N26 G80	Cancel drill cycle.	
N27 Z1.	Rapid to safe plane.	
N28 M5	Spindle Off.	End Program
N29 M9	Coolant Off.	
N30 G91 G28 Z0	Return to machine Home position in Z.	
N31 G91 G28 X0 Y0	Return to machine Home position in XY.	
N32 G90	Reset to absolute positioning mode (for safety).	
N33 M30	Reset program to beginning.	
%	End Program.	

https://www.autodesk.com/products/fusion-360/blog/cnc-programming-fundamentals-g-code-2020-update/



Key CAM software considerations

- work holding – the starting material must be clamped on the machine table in a known location
- the starting material is called the stock model in CAM – can be any shape (blue block)
- its location is defined as the Work Coordinate System (WCS) in CAM
- WCS orig del (corner of transparent block)



Example:

G54 – Work Offset

This code is used to define a fixture offset which determines the distance from a machine's internal coordinates to the WCS. You can program multiple offsets if a job requires machining multiple parts at once (G55, G56, etc.)

Work Offset	X	Y	Z
G54	14.2567	6.6597	2.0183
G55	0.0000	0.0000	0.0000
G56	0.0000	0.0000	0.0000
G57	0.0000	0.0000	0.0000
G58	0.0000	0.0000	0.0000
G59	0.0000	0.0000	0.0000

https://www.autodesk.com/products/fusion-360/blog/cnc-programming-fundamentals-g-code-2020-update/

Work holding video

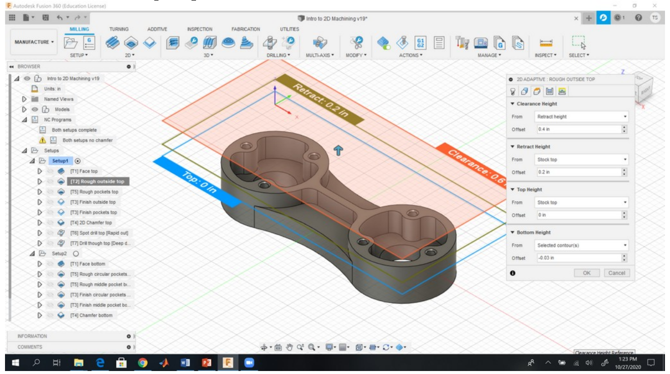
https://www.youtube.com/watch?v=J1VtofzVG24

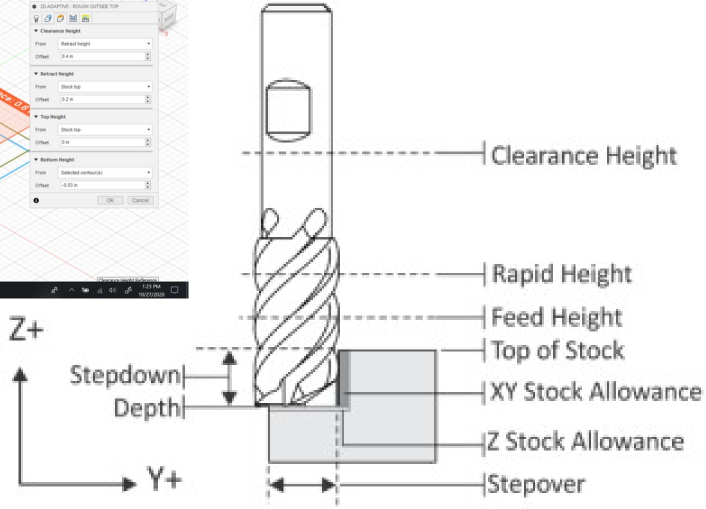
Work locating video

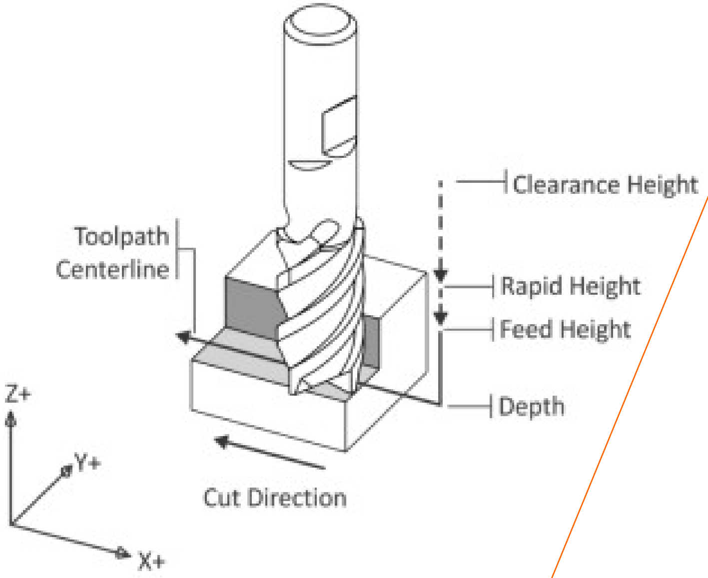
https://www.youtube.com/watch?v=r7-eEj_qq5M

Key CAM software considerations

- process definitions in CAM includes planes for: rapid motions, feed motions, stock top, feature locations in CAD

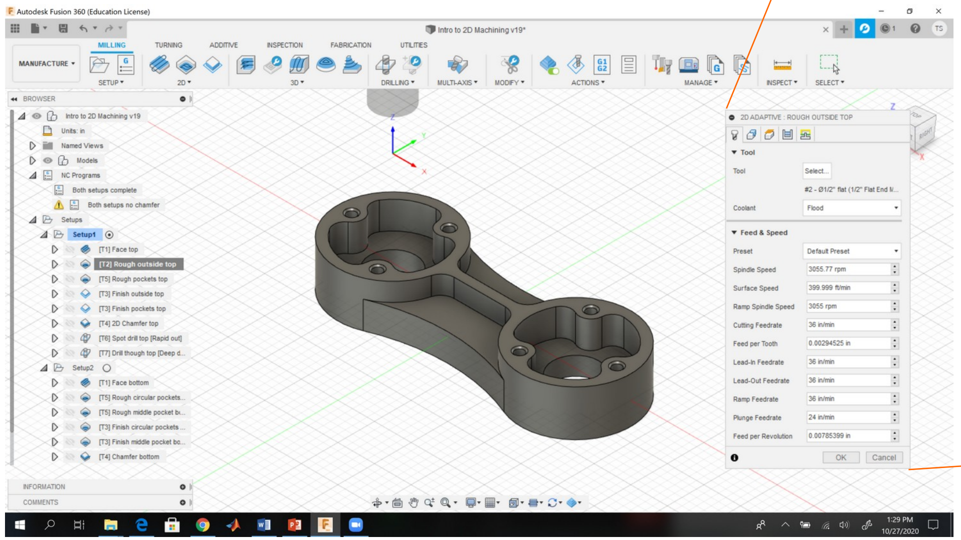


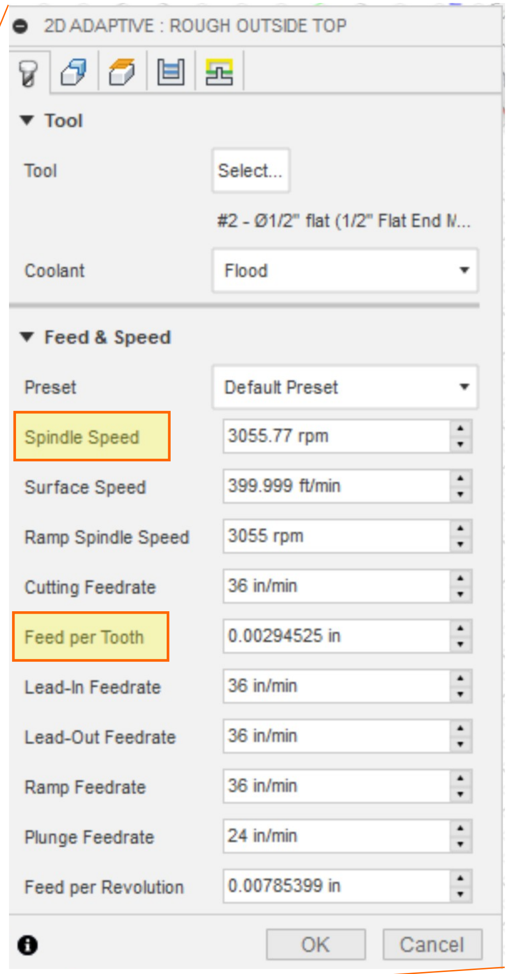




- machining parameters
    - axial depth of cut, stepdown
    - radial depth of cut, stepover
    - spindle speed
    - feed per tooth

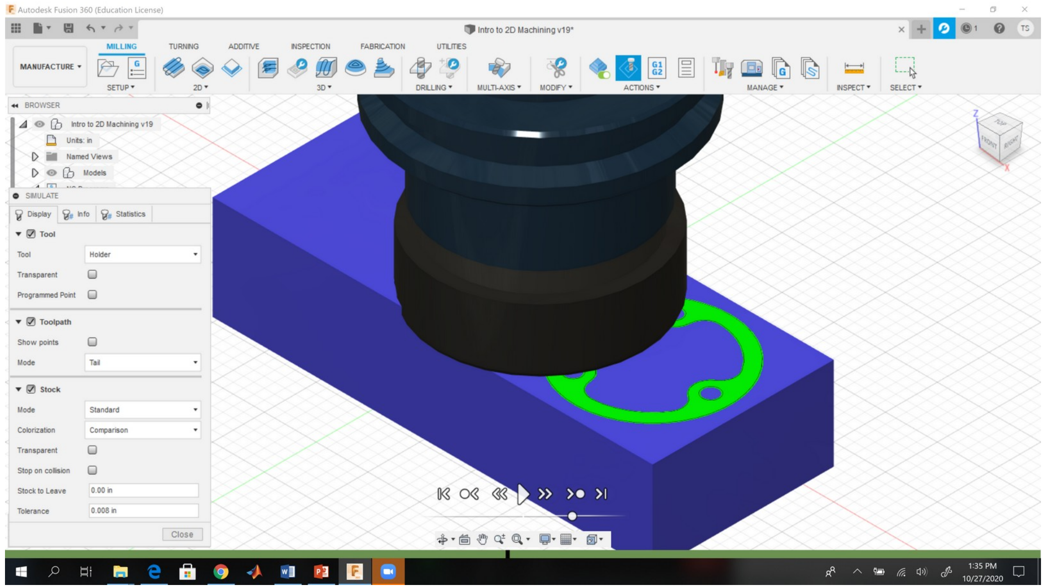
    - feed in motions
    - feed out motions

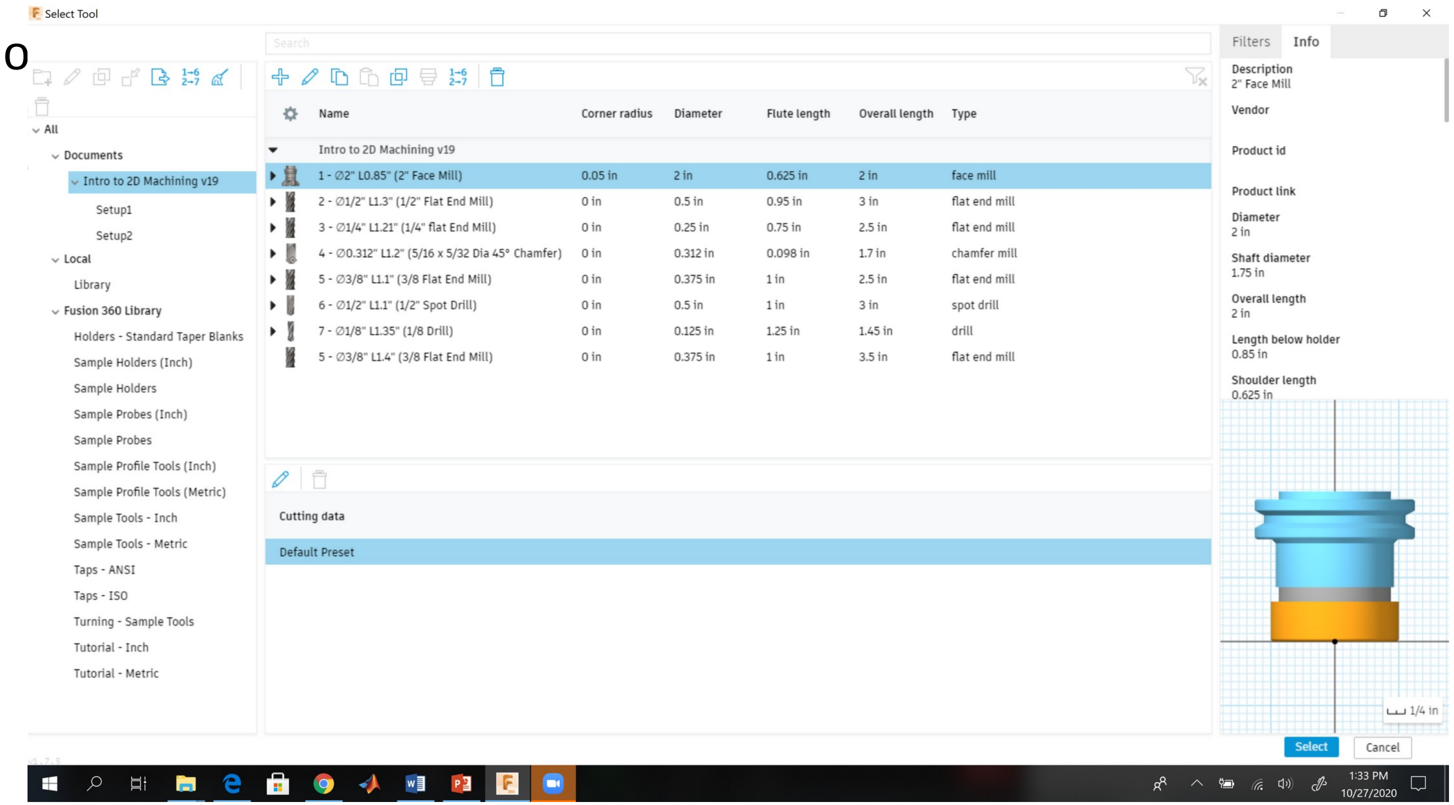




Key CAM software considerations

- process definitions in CAM includes tool selectio

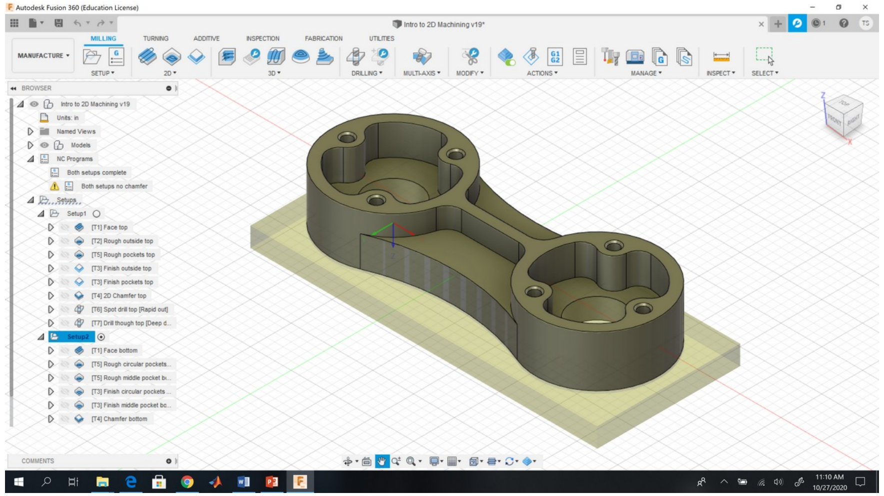


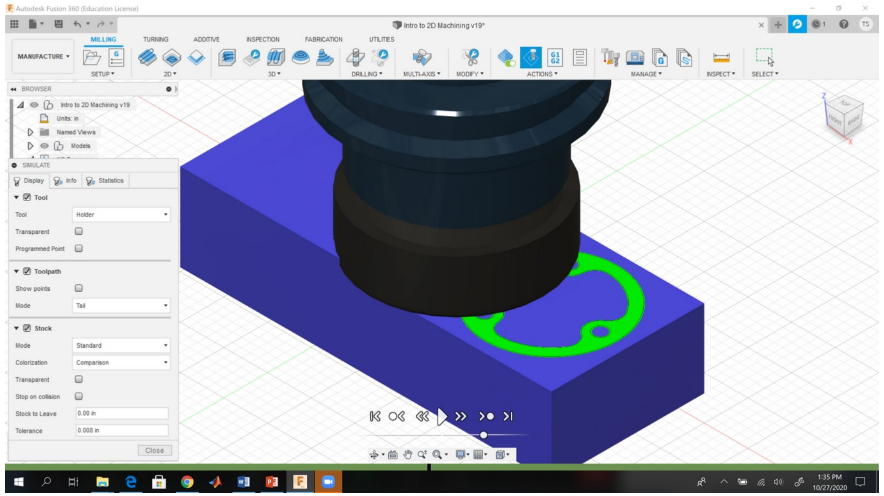


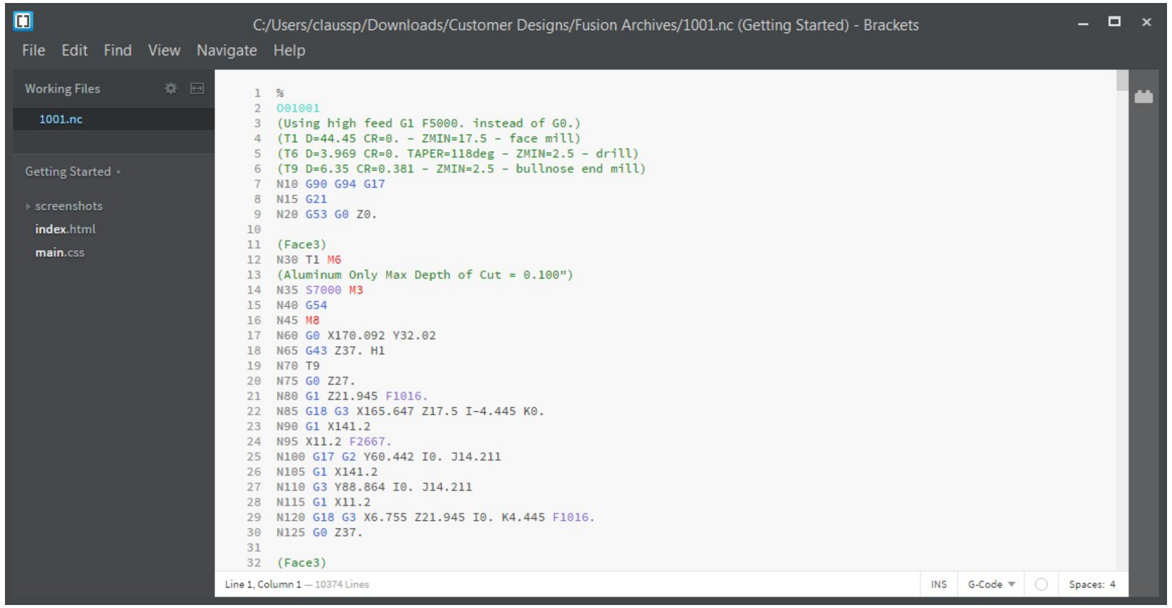
- when the machine-spindle-holder-tool is selected, a **dynamic system** is defined
- the cutting force causes vibration because the tool is not rigid, which affects the machining process
- this dynamic system should be considered when selecting machining parameters

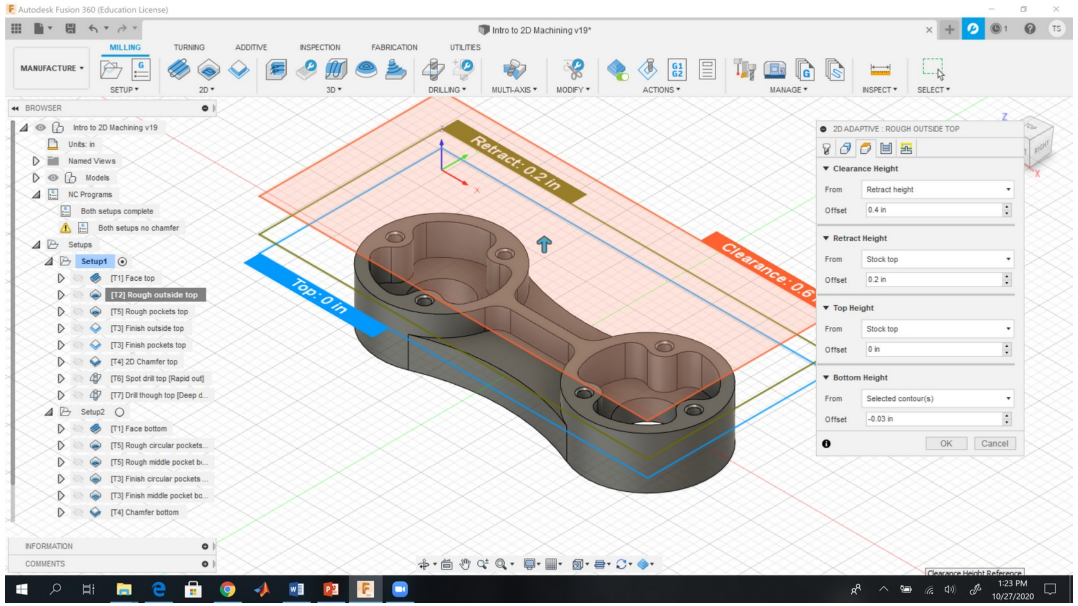
CNC machining

- CAD – digital model
- CAM – toolpaths
  - M/G codes
  - post-processor
- work holding
  - work coordinate system
  - stock model
- **See the CAM example** (extra instructional materials)









Speeds/feeds and workholding

Tony Schmitz

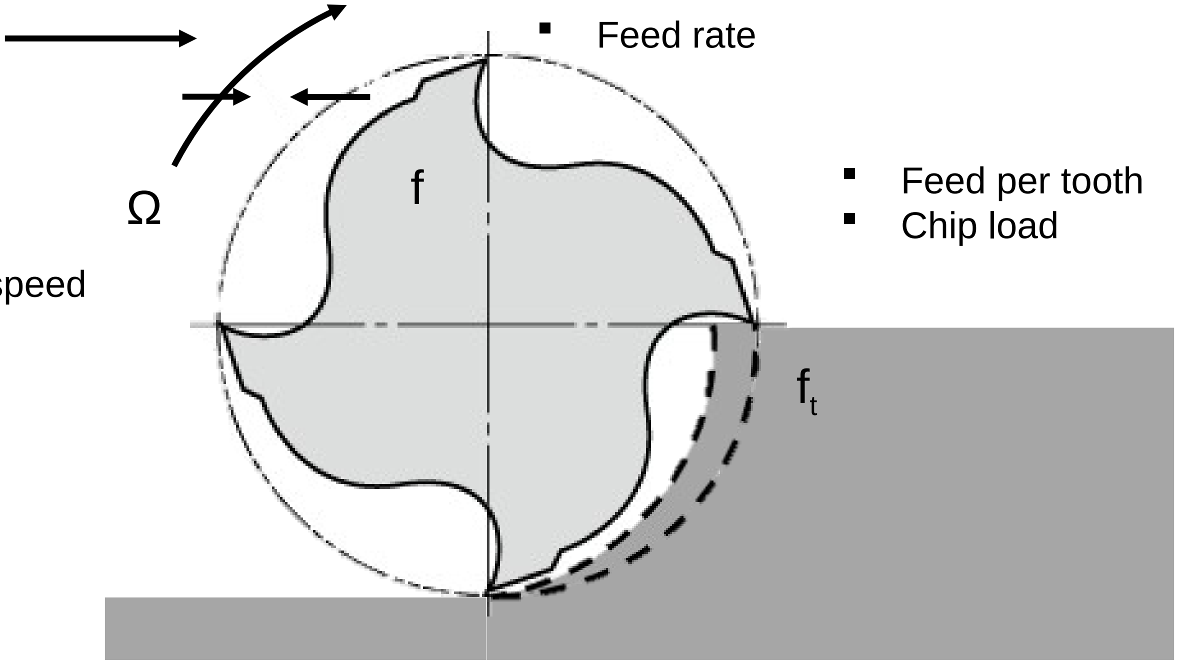
Speeds/feeds for milling

Spindle speed: Ω [rpm] = 12 [in/ft] x V [sfm]/ (π x D [in/rev])

- Ω - spindle speed [rpm]
- V- cutting speed/peripheral velocity [surface feet per minute]
- D - tool diameter [in]
- Pay attention to units if converting to metric

Feed rate: f [ipm] = Ω [rpm] x f_t [ipt] x N [teeth]

- f - linear feed rate [inches per minute]
- Ω - spindle speed [rpm]
- f_t - feed per tooth [inches per tooth]
- N - number of teeth



- Feed rate
- Feed per tooth
- Chip load
- Spindle speed

Spindle speed: Ω [rpm] = 12 [in/ft] x V [sfm]/ (π x D [in/rev])

- Ω - spindle speed [rpm]
- **V - cutting speed/peripheral velocity [surface feet per minute]**
- D - tool diameter [in]
- Pay attention to units if converting to metric

Feed rate: f [ipm] = Ω[rpm] x f_t [ipt] x N [teeth]

- f - linear feed rate [inches per minute]
- Ω - spindle speed [rpm]
- **f_t - feed per tooth [inches per tooth]**
- N - number of teeth

Material	Hardness Bhn	SFM	Feed Per Tooth					
			1/8"dia	¼"	3/8"	½"	¾"	1"
Aluminum 2011,2024,6061,7075	--	**800**	.0016	.0026	.004	.0055	.007	.009
Free Machining 12L14, 1215,11L17	80-160	**150**	.0006	.0015	.0022	.004	.005	.0055
Low Carbon Steel 1008, 1018, 1020	*	*						
Medium Carbon Steel 4140, 8620	*	*						
Tool Steel, H.S.S A2,D2,M2,M42	*	*						
Cast Iron	120-175	**200**	.0005	.0014	.002	.0035	.005	.0052
Stainless Steel 300 Series	--	**130**	.0005	.0013	.0019	.0033	.004	.0044
Stainless Steel 400 Series	*	*						
Stainless Steel 15-5, 17-4	*	*						
Titanium 6AL-4V	*	*						
Inconel 625, 718	*	*						
Copper, Bronze	120-160	**450**	.0009	.0018	.0026	.0045	.006	.0065
Brass (360 half hard)	--	**700**	.0013	.0022	.0038	.005	.007	.009

CAUTION: Recommended ranges are typically for highest material removal with a stiff setup and adequate lubrication/chip evacuation.

DOES NOT guarantee

- Stable vibration in every setup/machine
- Optimal tool life

Speeds/feeds for milling

Spindle speed: Ω[rpm] = 12 [in/ft] x V [sfm]/ (π x D [in/rev])

Feed rate: f [ipm] = Ω[rpm] x f_t [ipt] x N [teeth]

Example:
- Aluminum workpiece (**800 sfm**)
- ½" diameter carbide end mill (**0.0055"** per tooth)
  - Ω = **6115** rpm
- 3 teeth
  - f = **100** ipm

Example:
- 300 series stainless steel workpiece (**130** sfm)
- ½" diameter carbide end mill (**0.0033"** per tooth)
  - Ω = **994** rpm
- 3 teeth
  - f = **10** ipm

Material	Hardness Bhn	SFM	Feed Per Tooth 1/8"dia	¼"	3/8"	½"	¾"	1"
Aluminum 2011,2024,6061,7075	--	**800**	.0016	.0026	.004	.0055	.007	.009
Free Machining 12L14, 1215,11L17	80-160	**150**	.0006	.0015	.0022	.004	.005	.0055
Low Carbon Steel 1008, 1018, 1020	*	*						
Medium Carbon Steel 4140, 8620	*	*						
Tool Steel, H.S.S A2,D2,M2,M42	*	*						
Cast Iron	120-175	**200**	.0005	.0014	.002	.0035	.005	.0052
Stainless Steel 300 Series	--	**130**	.0005	.0013	.0019	.0033	.004	.0044
Stainless Steel 400 Series	*	*						
Stainless Steel 15-5, 17-4	*	*						
Titanium 6AL-4V	*	*						
Inconel 625, 718	*	*						
Copper, Bronze	120-160	**450**	.0009	.0018	.0026	.0045	.006	.0065
Brass (360 half hard)	--	**700**	.0013	.0022	.0038	.005	.007	.009

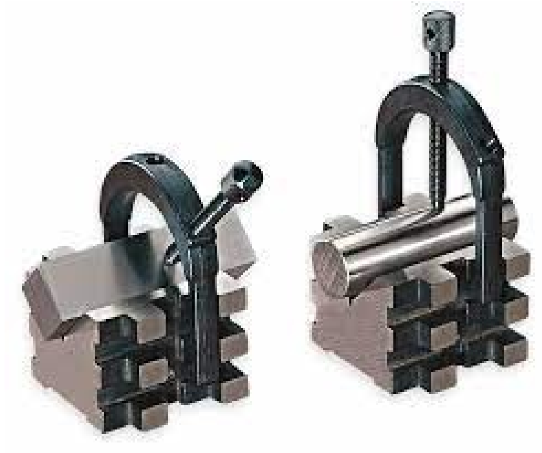
Online calculators

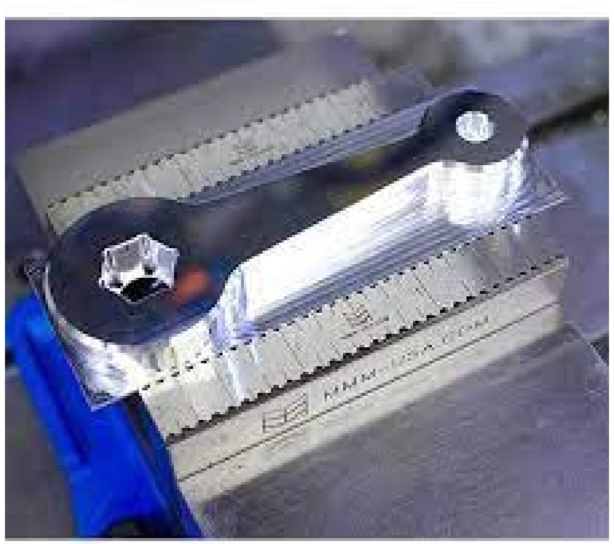
https://www.machiningdoctor.com/calculators/chip-load-calculator/

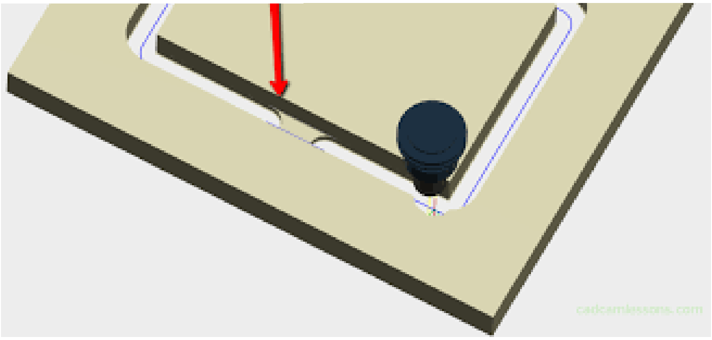
https://idcwoodcraft.com/pages/chipload-calculator?srsltid=AfmBOopi6apWNDgXHI1z6qHo004TvP9APusqQFymbNpEkTrsdQauIq3Z

Workholding, or fixturing, is a method for securing a workpiece during a machining operation

- vise
- toe clamp
- vee block
- vacuum fixturing
- tab/picture frame



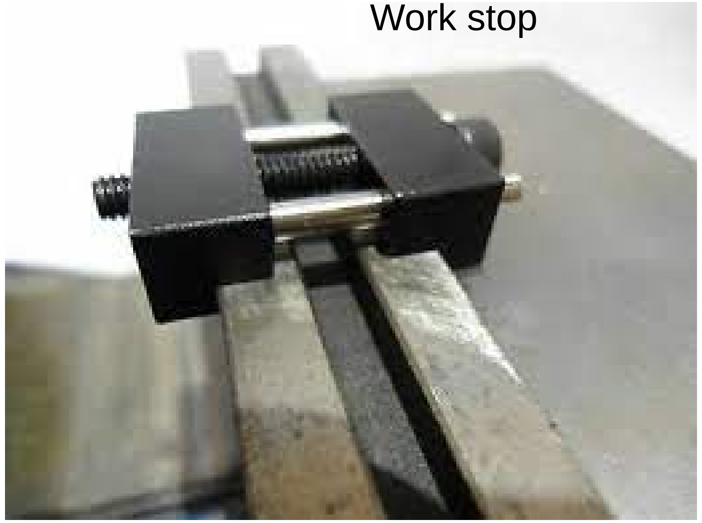


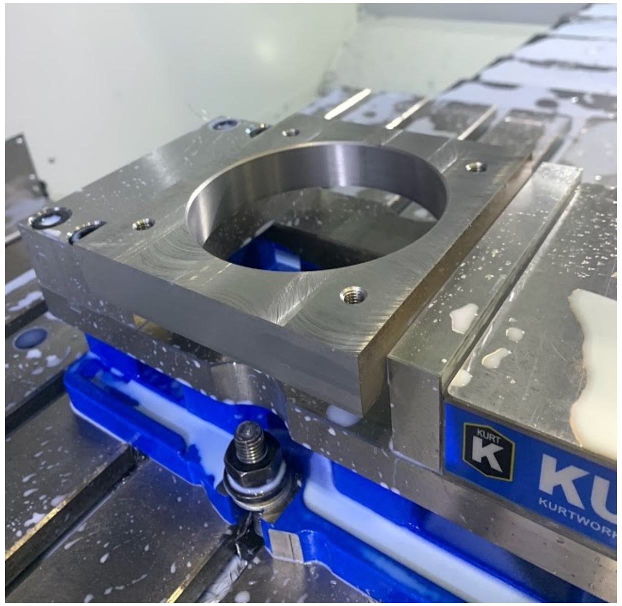


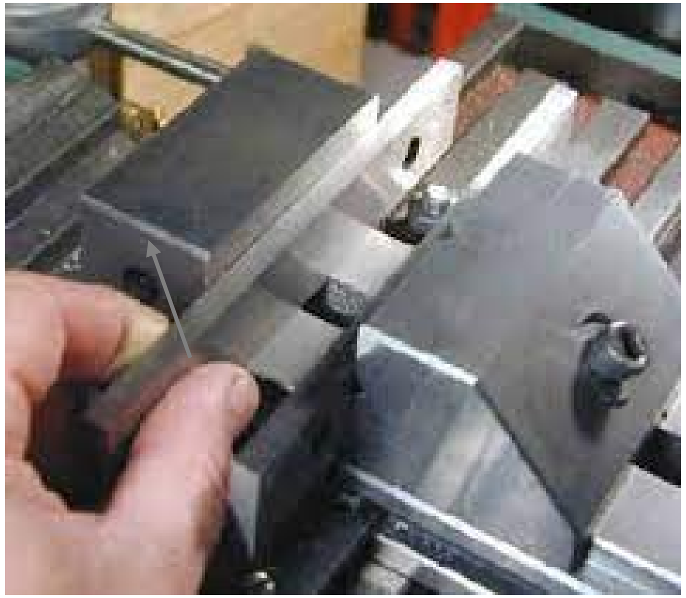
Vise

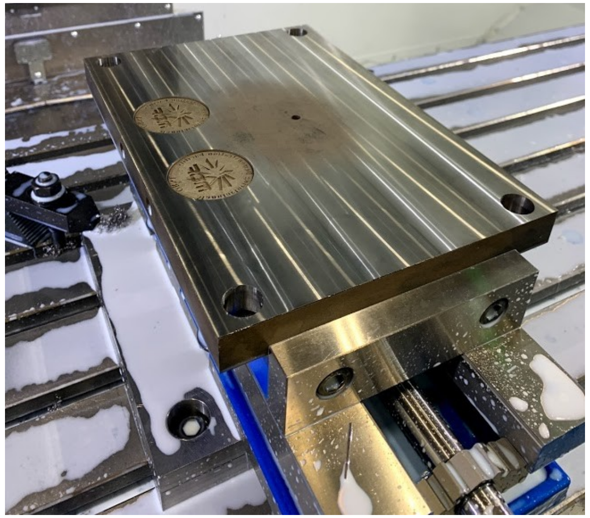
- most common workholding method for milling
- prismatic parts
- frequently used with parallels
- traditionally hard, flat steel jaws are used
- work stops.

Work stop









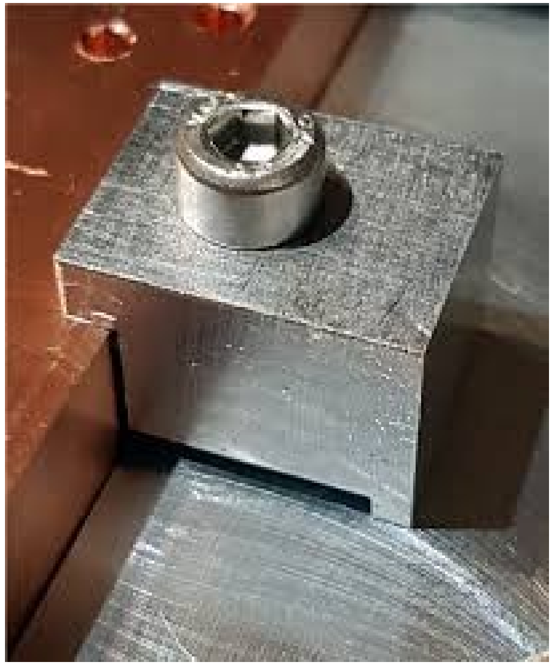
Parallels change height of part from table

Soft jaws can be used to hold parts with complex geometries.

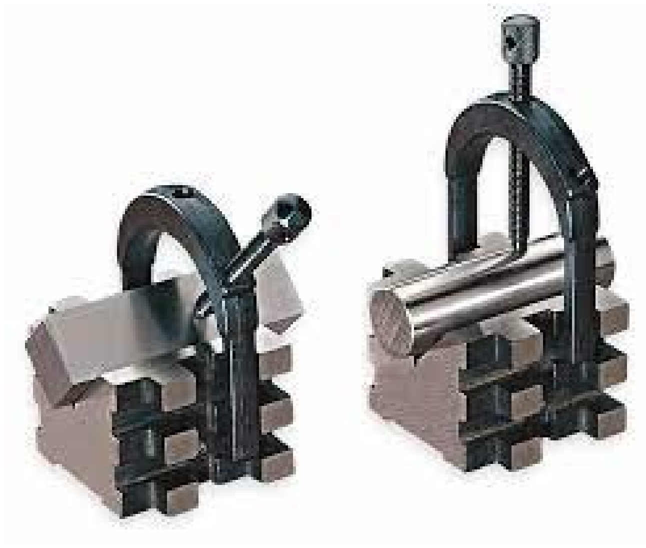
Soft jaws

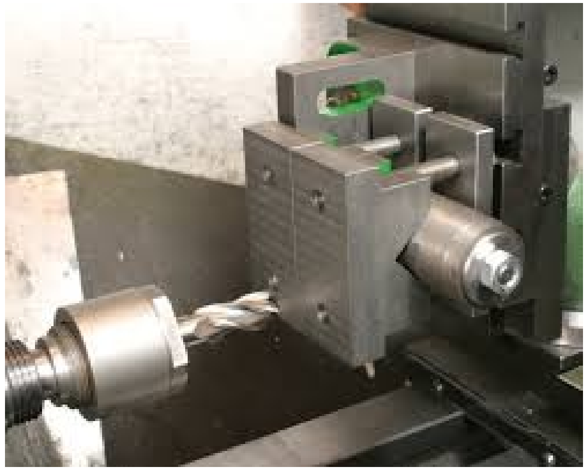
Extended size jaws can be used to clamp longer or taller parts.

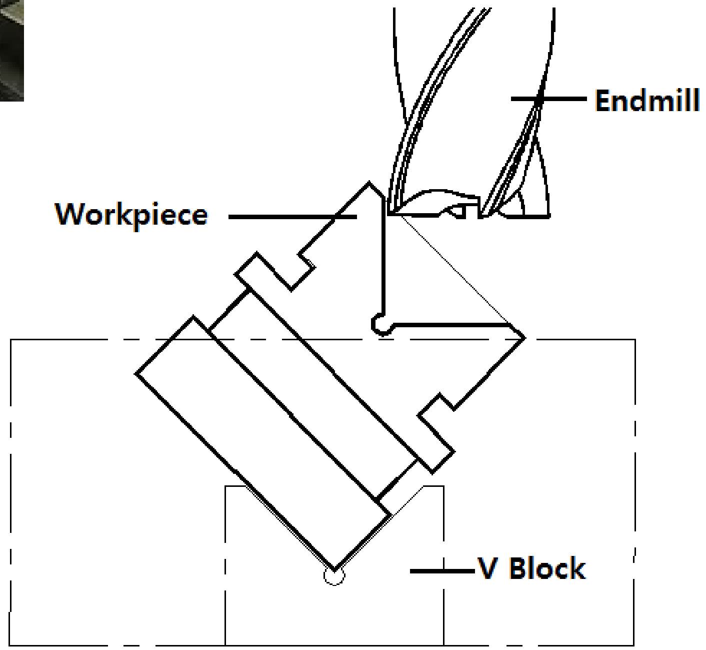
Toe clamps can be used to clamp a variety of workpiece geometries to a table or other standoff.



Vee blocks can be used to clamp round components or produce an angle on a prismatic component.

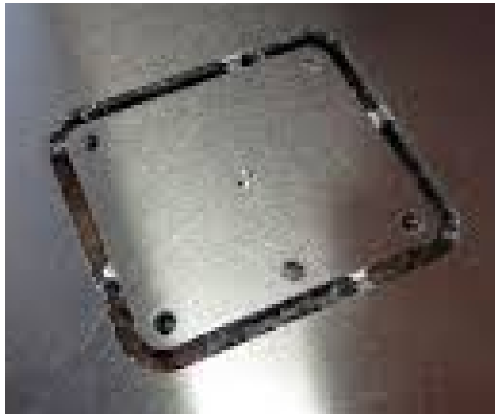


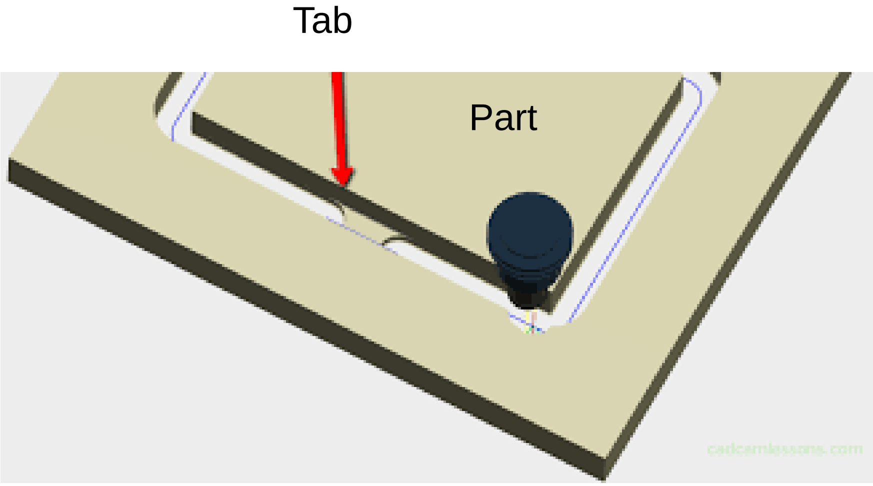


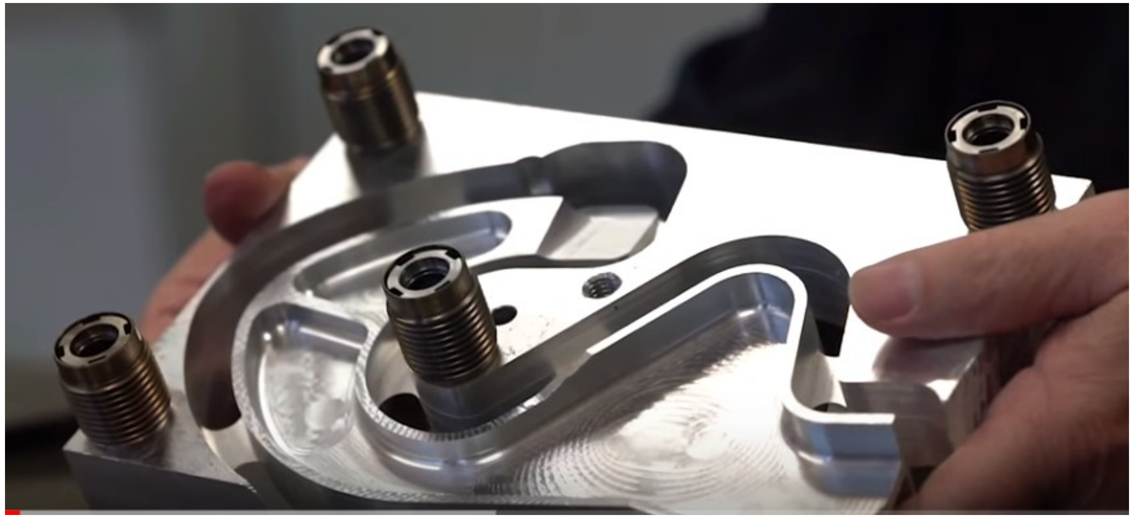


Vacuum plates can be used to hold workpieces down with low profile hardware. Often used to hold plates flat for machining.

Tabs/picture frame can be used to hold small or thin parts within a larger piece of stock. The stock acts as fixture for oddly shaped parts. Tabs are removed by secondary process.

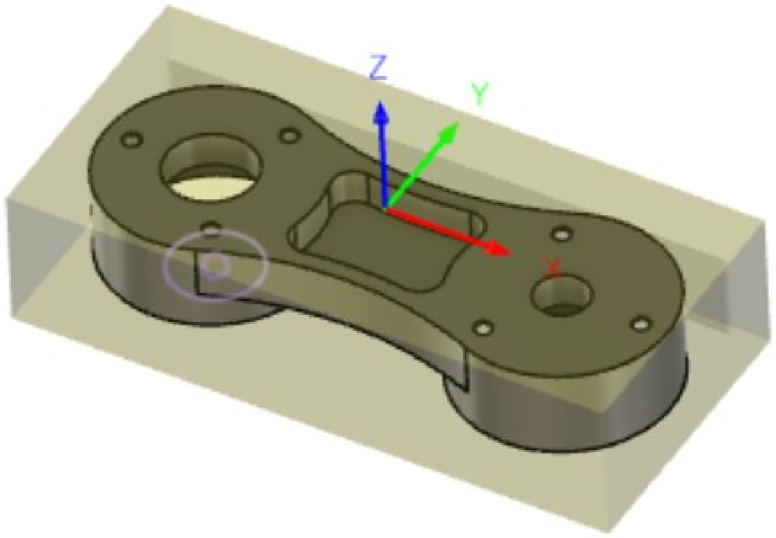






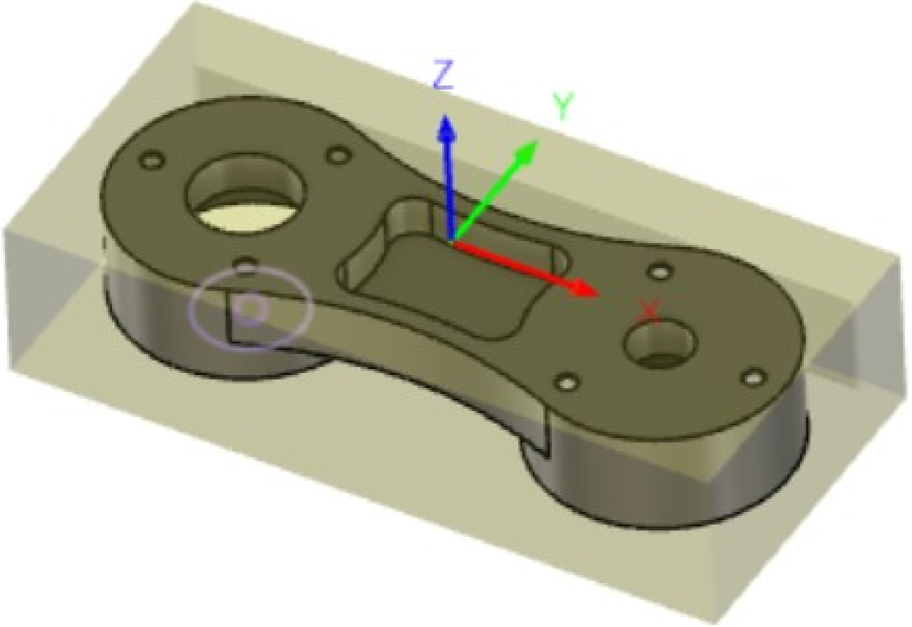
The **work coordinate system (WCS)** defines the position and orientation of the XYZ axes of the part program.
- This is the coordinate system in which we program our CAM operations.

How do we select the location of our WCS?
- meet drawing requirements
- features which are easy to find for setup on the machine
- examples are an edge, corner, hole, bisection between planes, or bolt pattern.

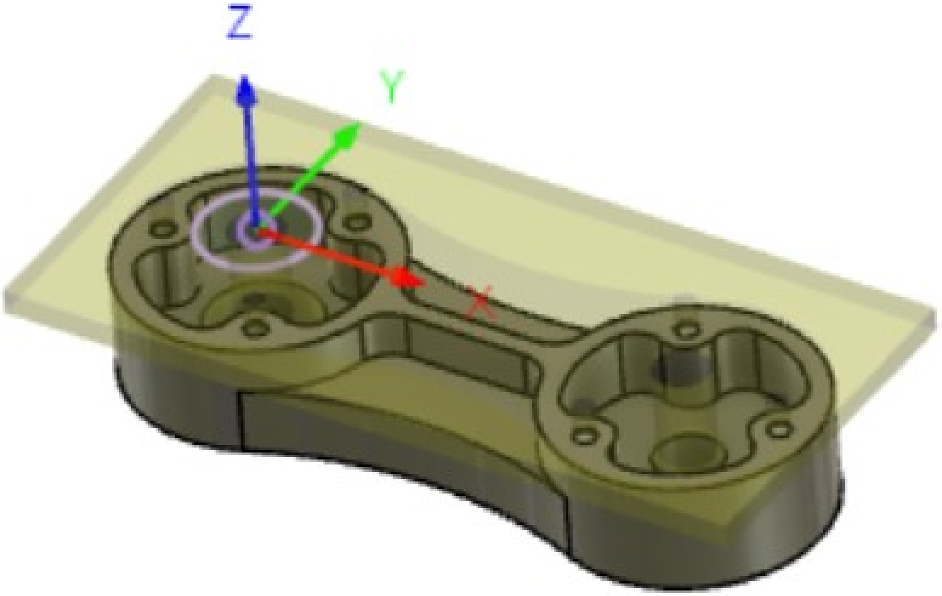


- When machining multiple sides of a part we need to transfer a known feature.

OP10



OP20

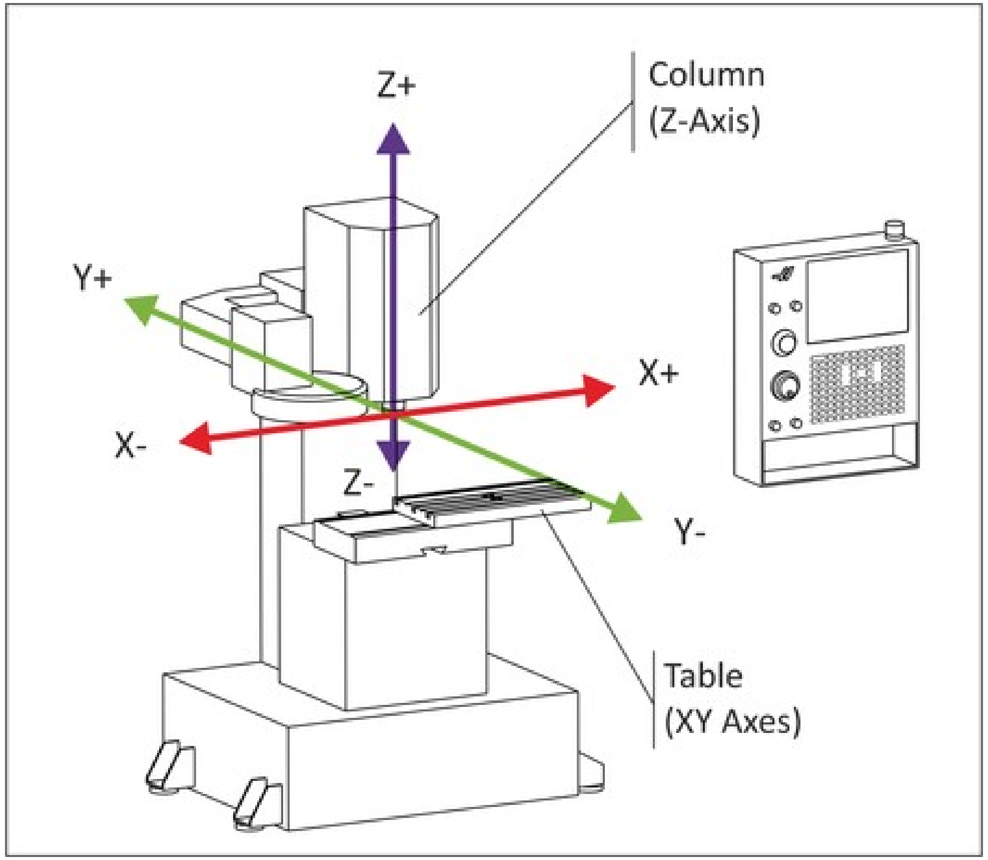


- Can use the hole location from OP10 to re-find the part when we turn it over from OP20.

Every machine has an associated coordinate system (**machine coordinate system**).

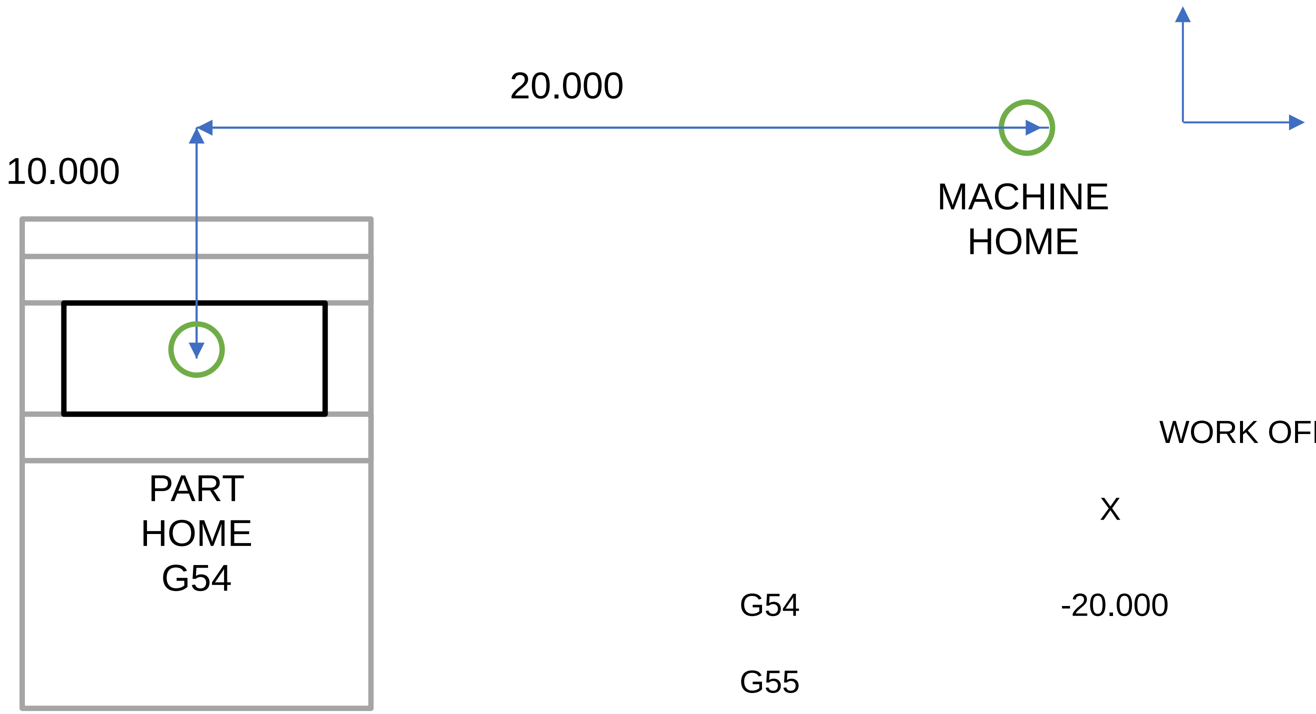
Typically, the origin of the machine coordinates (machine home) is at the XYZ travel limits.

- The movement in the coordinate system is always based on how your tool moves, not the table.
- This coordinate system may not be intuitive to the operator.
- We need to translate between machine coordinates and work coordinates.



When the table moves to the left, the tool "moves" to the right.

- We translate between machine coordinates and work coordinates using **work offsets.**
- Using a probe or edge-finder, the workpiece origin can be found in machine coordinates.
- A work offset is used to measure the difference between the WCS and the machine home.



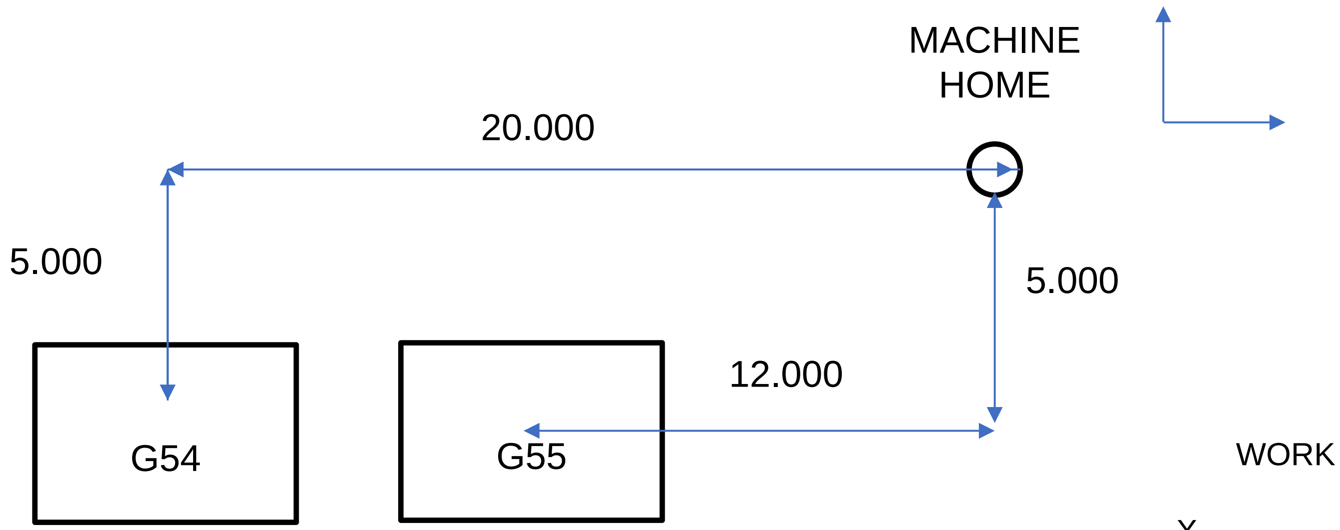
Y

X

WORK OFFSETS

	X	Y	Z
G54	-20.000	-10.000	-12.000
G55			
G56			
G57			
G58			
G59			

Multiple WCSs can be used
- to reference different operations or features
- to reference multiple parts.



Y

X

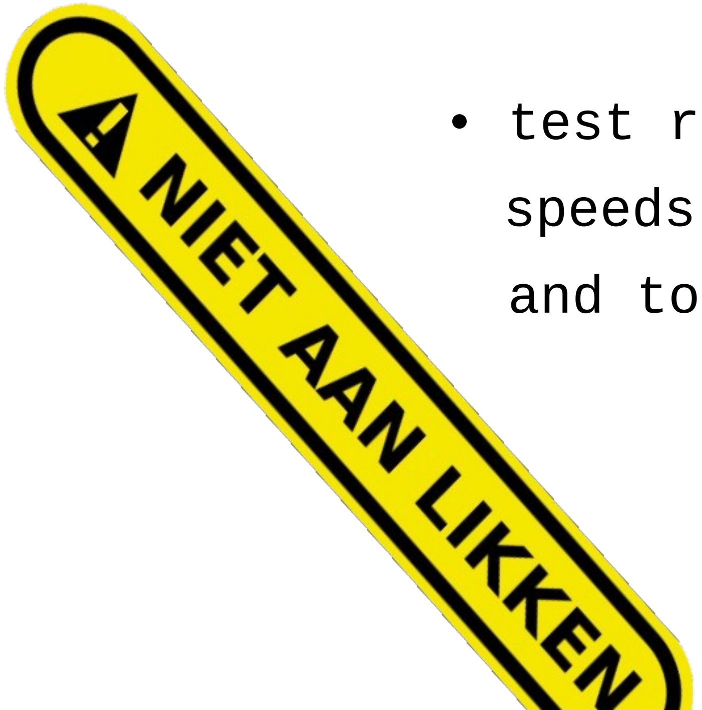
WORK OFFSETS

	X	Y	Z
G54	-20.000	-5.000	-1.000
G55	-12.000	-5.000	-1.000
G56			
G57			
G58			
G59			

group assignment

- do your lab's safety training

- test runout, alignment, fixturing, speeds, feeds, materials, and toolpaths for your machine



assignment

individual assignment

- make (design+mill+assemble) something big (~meter-scale)
- extra credit: don't use fasteners or glue
- extra credit:
  - include curved surfaces
  - use three-axis toolpaths

assignment