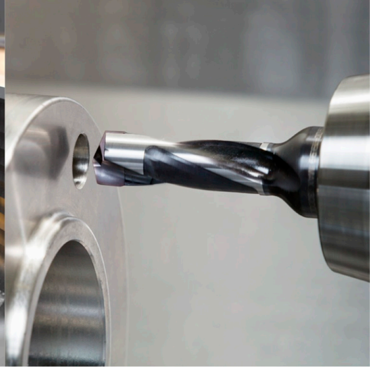
PART 1-2

SANDVIK Coromant

Training Handbook

METAL CUTTING TECHNOLOGY





Metal Cutting Technology training handbook

This handbook will serve as your main source of information throughout the Sandvik Coromant metal cutting training and may also be used as reference in your future endeavors.

Train with us

Deepen and broaden your knowledge with our training programs. We offer basic to advanced training at our Centers worldwide, allowing you to combine theory with practice using state of the art equipment and machines.

Visit sandvik.coromant.com to find activity schedules and register.


Content

Turning

Turning generates cylindrical and rounded forms with a single-point tool. In most cases the tool is stationary with the workpiece rotating.

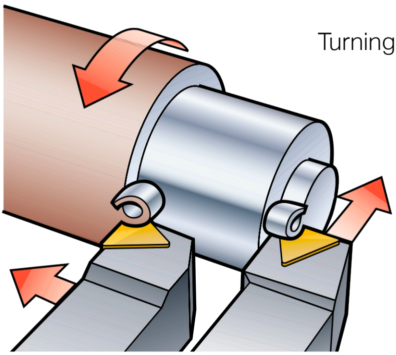
A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

General turning operations

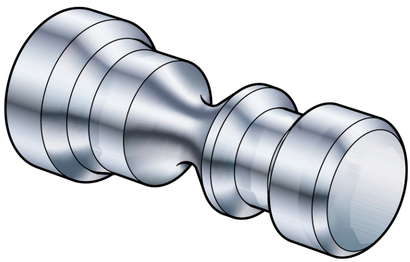
Turning is the combination of two movements – rotation of the workpiece and feed movement of the tool.

The feed movement of the tool can be along the axis of the workpiece, which means the diameter of the part will be turned down to a smaller size. Alternatively, the tool can be fed towards the center (facing off) at the end of the part.

Often feeds are combinations of these two directions, resulting in tapered or radius surfaces.



Turning and facing as axial and radial tool movements.

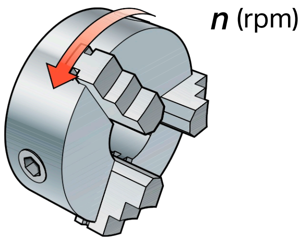


Three common turning operations:

- Longitudinal turning
- Facing
- Profiling

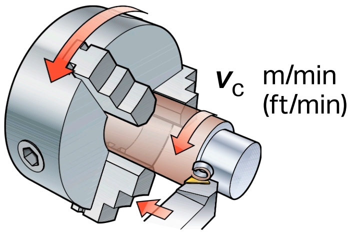
Definitions of terms

Spindle speed

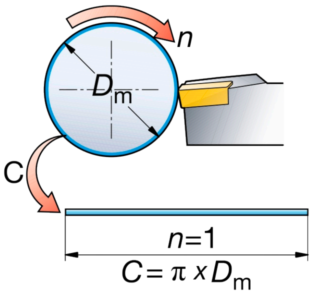


The spindle speed rpm (revolution per minute) is the rotation of the chuck and workpiece.

Cutting speed



The cutting speed is the surface speed, m/min (ft/min), at which the tool moves along the work-piece in feet (meters) per minute.



Definition of cutting speed

The definition of cutting speed (v_c) as the result of the diameter, pi (π) and the spindle speed (n) in the revolutions per minute (rpm). The circumference (C) is the distance the cutting edge moves in a revolution.

v_c = cutting speed, m/min (ft/min)
D_m = machined diameter, mm (inch)
n = spindle speed, (rpm)
C = Circumference, $\pi \times D_m$ mm (inch)
π (pi) = 3.14

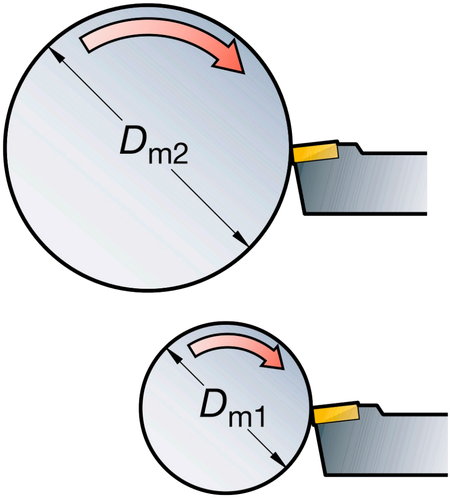
Metric

$$v_c = \frac{\pi \times D_m \times n}{1000} \text{ m/min}$$

Inch

$$v_c = \frac{\pi \times D_m \times n}{12} \text{ ft/min}$$

Calculation of the circumference (C)



- Circumference = π x diameter
- π (pi) = 3.14

Example:

D_{m2} = 100 mm (3.937 inch)
C = 3.14 x 100
= 314 mm

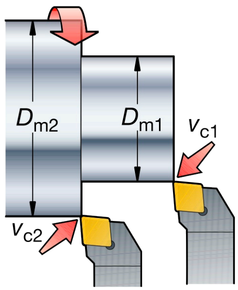
C = 3.14 x 3.937
= 12.362 inch

D_{m1} = 50 mm (1.969 inch)
C = 3.14 x 50
= 157 mm

C = 3.14 x 1.969
= 6.183 inch

Example of cutting speed calculation

The cutting speed differs depending on the workpiece diameter.



Given:

Spindle speed, n = 2000 rpm

Diameter, D_{m1} = 50 mm (1.969 inch)

Diameter, D_{m2} = 80 mm (3.150 inch)

Metric

$$v_c = \frac{\pi \times D_m \times n}{1000} \text{ m/min}$$

$$v_{c1} = \frac{3.14 \times 50 \times 2000}{1000} = 314 \text{ m/min}$$

$$v_{c2} = \frac{3.14 \times 80 \times 2000}{1000} = 502 \text{ m/min}$$

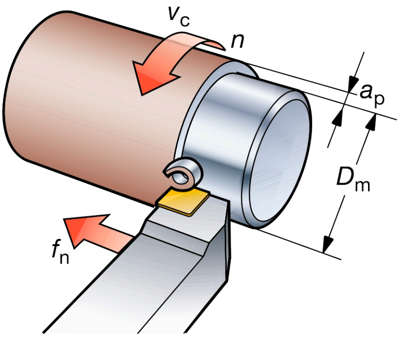
Inch

$$v_c = \frac{\pi \times D_m \times n}{12} \text{ ft/min}$$

$$v_{c1} = \frac{3.14 \times 1.969 \times 2000}{12} = 1030 \text{ ft/min}$$

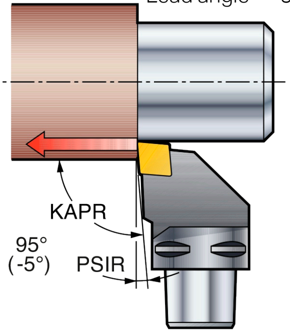
$$v_{c2} = \frac{3.14 \times 3.150 \times 2000}{12} = 1649 \text{ ft/min}$$

Definitions of terms



n	= spindle speed (rpm)
v_c	= cutting speed m/min (ft/min)
f_n	= cutting feed mm/r (inch/r)
a_p	= depth of cut mm (inch)
KAPR	= entering angle
PSIR	= lead angle

Entering angle = 95°
Lead angle = -5°



Spindle speed

The workpiece rotates in the lathe, with a certain spindle speed (n), at a certain number of revolutions per minute (rpm).

Surface/cutting speed

The cutting speed (v_c) in m/min (ft/min) at which the periphery of the cut workpiece diameter passes the cutting edge.

Feed

The cutting feed (f_n) in mm/r (inch/r) is the movement of the tool in relation to the revolving workpiece. This is a key value in determining the quality of the surface being machined and for ensuring that the chip formation is within the scope of the tool geometry. This value influences, not only how thick the chip is, but also how the chip forms against the insert geometry.

Depth of cut

The cutting depth (a_p) in mm (inch) is half of the difference between the un-cut and cut diameter of the workpiece. The cutting depth is always measured at right angles to the feed direction of the tool.

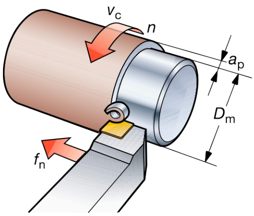
Entering angle KAPR, lead angle PSIR

The cutting edge approach to the work-piece is expressed through the entering angle (KAPR), which is the angle between the cutting edge and the direction of feed. It can also be expressed as the lead angle (PSIR), the angle between the cutting edge and the workpiece plane. The entering angle is important in the basic selection of the correct turning tool for an operation.

Calculating cutting data

Cutting speed

Example of how to calculate the spindle speed (n) from cutting speed (v_c).



Given:

Cutting speed, v_c = 400 m/min (1312 ft/min)

Diameter D_m = 100 mm (3.937 inch)

Metric

$$n = \frac{v_c \times 1000}{\pi \times D_m} \quad \text{r/min}$$

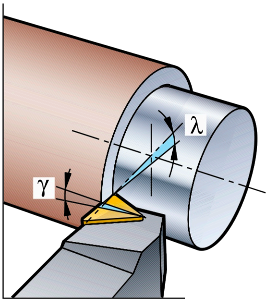
$$n = \frac{400 \times 1000}{3.14 \times 100} = 1274 \text{ r/min}$$

Inch

$$n = \frac{v_c \times 12}{\pi \times D_m} \quad \text{r/min}$$

$$n = \frac{1312 \times 12}{3.14 \times 3.937} = 1274 \text{ r/min}$$

Inclination and rake angles



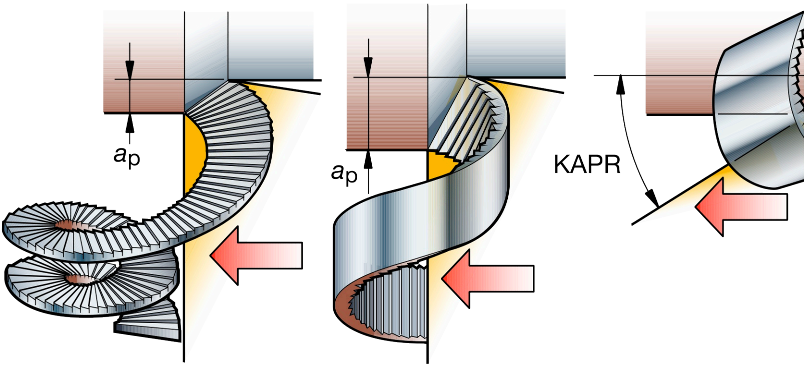
Rake angle

The rake angle gamma (GAMO) is a measurement of the edge in relation to the cut. The rake angle of the insert itself is usually positive and the clearance face is in the form of a radius, chamfer or land and affects tool strength, power consumption, finishing ability of the tool, vibration tendency and chip formation.

Inclination angle

The inclination angle lamda (LAMS) is the angle the insert is mounted in the tool holder. When mounted in the tool holder, the insert geometry and inclination in the tool holder will determine the resulting cutting angle with which the cutting edge cuts.

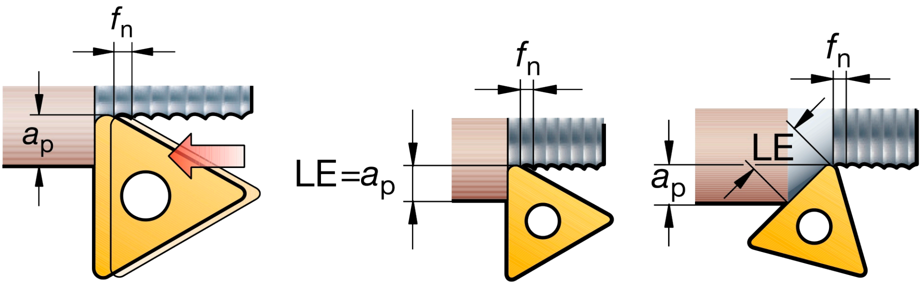
Cutting depth and chip formation



The cutting depth (a_p) is the length the edge goes into the workpiece.

Chip formation varies with depth of cut, entering (lead) angle, feed, material and insert geometry.

Feed rate and the effective cutting edge length



Feed rate

The feed rate (f_n) is the distance the edge moves along the cut per revolution.

Cutting edge length

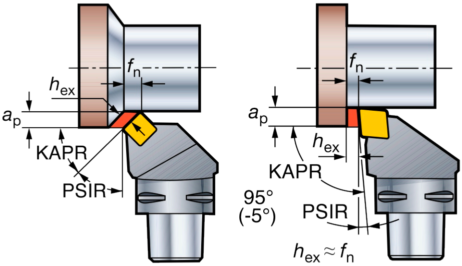
The effective cutting edge length (LE) relates to cutting depth and entering (lead) angle.

Insert shape selection, entering angle (lead angle) and chip thickness

The entering angle KAPR (lead angle PSIR), of the tool and the nose radius RE of the insert effect the chip formation, in that the chip cross-section changes.

The chip thickness is reduced and the width increases with a smaller entering angle or larger lead angle.

The direction of chip flow is also changed.



KAPR = 45°
PSIR = 45°
$h_{ex} \approx f_n \times 0.71$

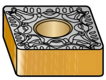
Entering angle KAPR (Lead angle PSIR)

- Is defined by the holder tip seat in combination with insert shape selected.

Maximum chip thickness h_{ex}

- Reduces relative to the feed rate as the entering angle reduces (lead angle increases).

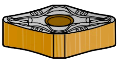
Possible entering (lead) angle positions for insert shapes



CNMG

Entering angle KAPR:
95°

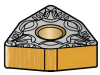
Lead angle PSIR:
-5°



DNMG

Entering angle KAPR:
107.5°, 93°, 62.5°

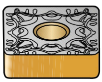
Lead angle PSIR:
-17.5°, -3°, 27.5°



WNMG

Entering angle KAPR:
95°

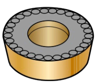
Lead angle PSIR:
-5°



SNMG

Entering angle KAPR:
45°, 75°

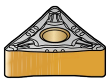
Lead angle PSIR:
45°, 15°



RCMT

Entering angle KAPR:
Variable

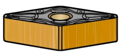
Lead angle PSIR:
Variable



TNMG

Entering angle KAPR:
93°, 91°, 60°

Lead angle PSIR:
-3°, -1°, 30°



VNMG

Entering angle KAPR:
117.5°, 107.5°, 72.5°

Lead angle PSIR:
-27.5°, -17.5°, 17.5°

The effect of entering angle (lead angle) on chip thickness

Maximum chip thickness h_{ex} reduces relative to the feed rate as the entering angle reduces (lead angle increases).

					
Entering angle KAPR Lead angle PSIR	95° -5°	75° 15°	60° 30°	45° 45°	90° min 0° max
Chip thickness compared to feed, mm (inch)	1	0.96	0.87	0.71	Variable
Contact length l_a, mm (inch) at a_p 2 mm (.079 inch)	2 (.079)	2.08 (.082)	2.3 (.091)	2.82 (.111)	Variable

Calculating power consumption

The net power (P_c) required for metal cutting is mainly of interest when roughing, where it is essential to ensure that the machine has sufficient power for the operation and is measured in kW and HP. The efficiency factor of the machine is also of great importance.

For information about the k_c value, see page H 16.

n = spindle speed (rpm)

v_c = cutting speed m/min (ft/min)

f_n = cutting feed mm/rev (inch/rev)

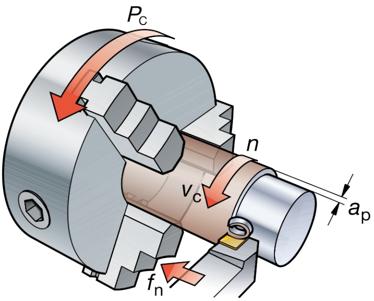
a_p = depth of cut mm (inch)

k_c = specific cutting force N/mm² (lbs/in²)

P_c = net power kW (HP)

kW = kilowatts

HP = horsepower



$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{60 \times 10^3} \quad \text{kW}$$

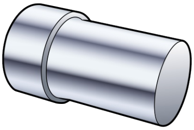
$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{33 \times 10^3} \quad \text{HP}$$

Selection procedure

Production planning process

1 Component

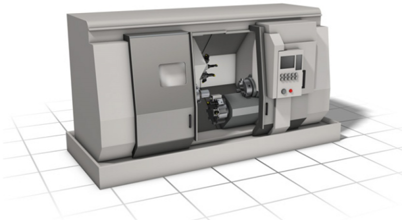
Dimension and type of operation



Workpiece material and quantity

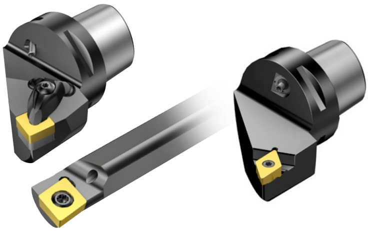
2 Machine

Machine parameters



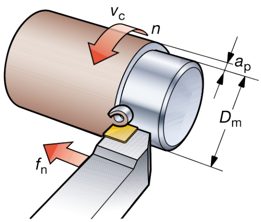
3 Choice of tool

Type of turning tool:
- External/internal
- Longitudinal
- Facing
- Profiling



4 How to apply

Cutting data, tool path, etc.

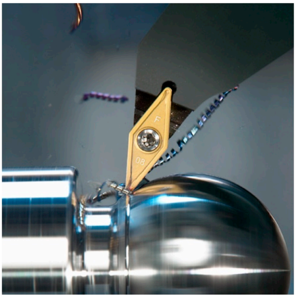


5 Troubleshooting

Remedies and solutions

1. Component and the workpiece material

Parameters to be considered



Component

- Analyze the dimensions and quality demands of the surface to be machined
- Type of operation (longitudinal, facing and profiling)
- External, internal
- Roughing, medium or finishing
- Tool paths
- Number of passes
- Tolerances.

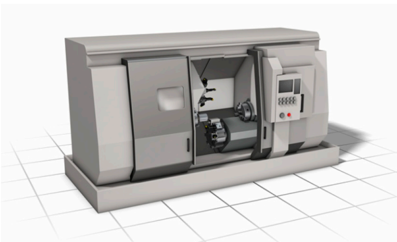
P	Steel
M	Stainless steel
K	Cast iron
N	Non-ferrous
S	Heat resistant super alloys and titanium
H	Hardened steel

Material

- Machinability
- Cast or pre-machined
- Chip breaking
- Hardness
- Alloy elements.

2. Machine parameters

Condition of the machine



Some important machine considerations:

- Stability, power and torque, especially for larger diameters
- Component clamping
- Tool position
- Tool changing times/number of tools in turret
- Spindle speed (rpm) limitations, bar feed magazine
- Sub spindle, or tail stock available?
- Use all possible support
- Easy to program
- Cutting fluid pressure.

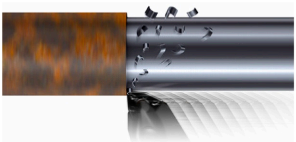
3. Choice of tools

General application - Turning with rhombic inserts

Advantages
- Operational versatility
- Large entering angle
- For turning and facing
- Good roughing strength.

Disadvantages
- Can cause vibration when turning slender components.

Turning with wiper inserts



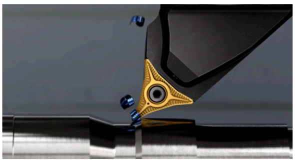
Advantages
- Increase feed and gain productivity
- Use normal feed rate and gain surface quality
- Productivity booster.

Disadvantages
- In back turning and profiling the wiper edge is not effective.

Coromant unique Turning concepts

Advantages
- Increased cutting data in profiling
- Increased ability to hold tolerance.

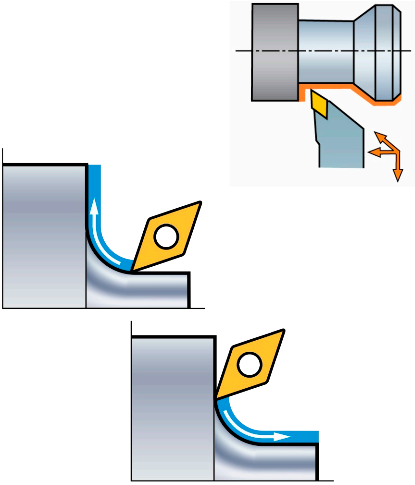
Advantages
- Multiple edge solution
- Chip control and predictable tool life.



Advantages
- Turning in all directions
- Efficient and productive turning.

4. How to apply

Important application considerations



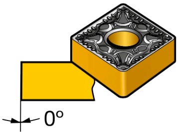
The tool path has a significant impact on the machining process.

It influences:
- Chip control
- Insert wear
- Surface quality
- Tool life.

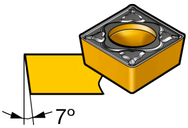
In practice, the tool holder, insert geometry, grade, workpiece material and tool path influences the cycle time and productivity considerably.

5. Troubleshooting

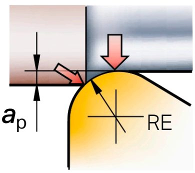
Some areas to consider



Negative style



Positive style



Insert clearance angle
- Use positive inserts for lower cutting forces in general and for internal turning.

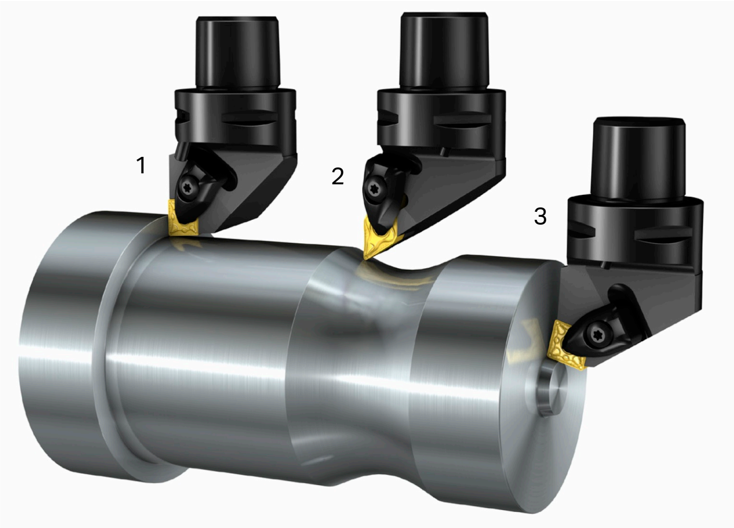
Chip breaking
- Optimize the chip breaking by changing the depth of cut, the feed or the insert geometry.

Nose radius
- The depth of cut should be no less then the nose radius (RE).

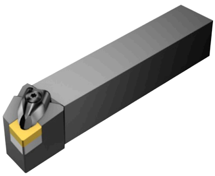
Insert wear
- Make sure that the flank wear does not exceed the general recommendation of 0.5 mm (.020 inch).

External Turning - negative inserts

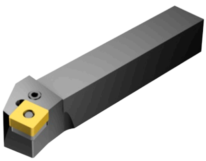
1. Longitudinal turning
2. Profiling
3. Facing



Overview of tool holders

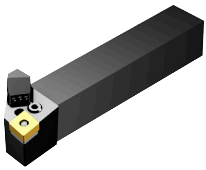


- Negative insert
- Rigid clamping system
- Modular/shank tools.



- Negative insert
- Lever clamping system
- Modular/shank tools.

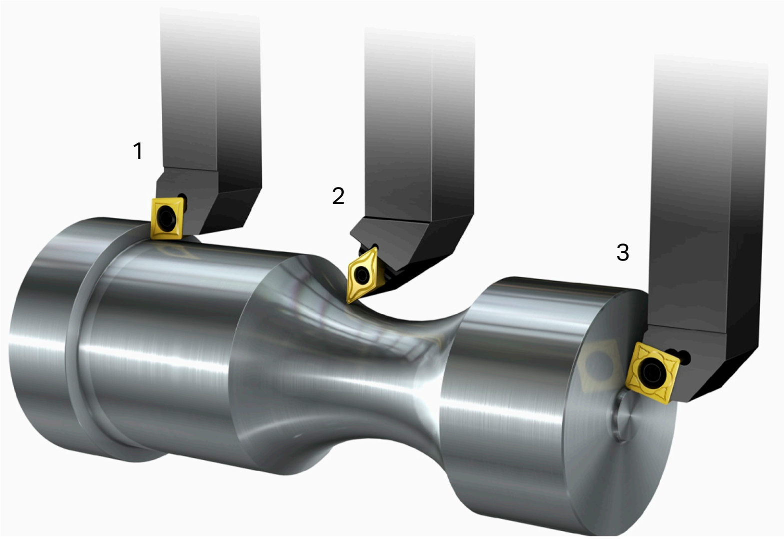
- Negative/positive inserts
- All clamping systems
- Cutting heads
- Modular/shank tools.



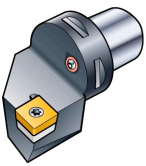
- Negative inserts
- Lever clamping system
- Precision coolant
- Modular/shank tools.

External Turning - positive inserts

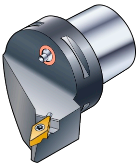
1. Longitudinal turning
2. Profiling
3. Facing



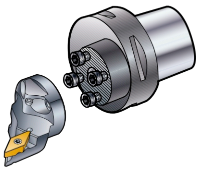
Overview of tool holders



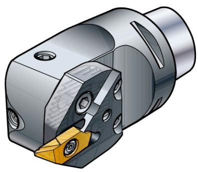
- Positive insert
- Screw clamping system
- Modular/shank tools
- Precision coolant.



- Positive insert
- Screw clamping system
- iLock™ interface
- Modular/shank tools.

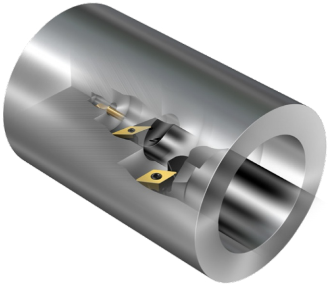


- Negative/positive insert
- All clamping systems
- Cutting heads
- Modular/shank tools.

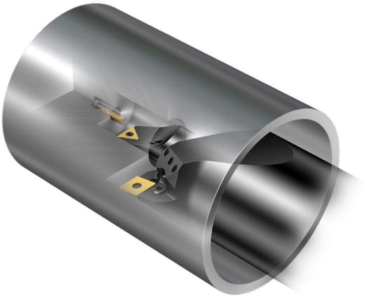


- Positive insert
- Screw clamping system
- Modular/shank tools.

Internal turning, negative/positive inserts

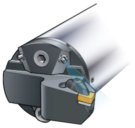


Positive inserts

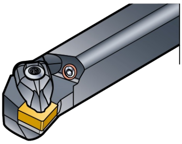


Negative inserts

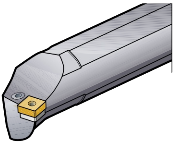
Overview of internal tool holders



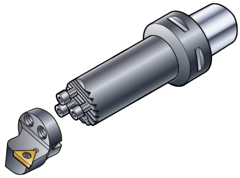
- Negative/positive inserts
- Dampened boring bars
- Boring bars



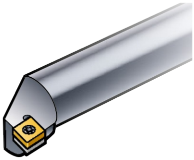
- Negative insert
- Rigid clamping system
- Modular/boring bars.



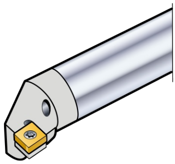
- Negative insert
- Lever clamping system
- Modular/boring bars.



- Negative/positive insert
- All clamping systems
- Cutting heads
- Dampened modular/ boring bars
- Precision coolant.

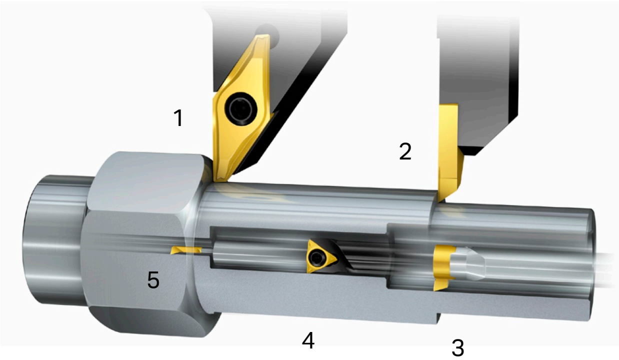


- Positive insert
- Screw clamping system
- Cutting heads
- Modular/boring bars.
- Precision coolant.



- Dampened boring bars
- Boring bars.

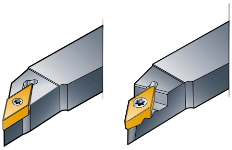
Tools for small part machining



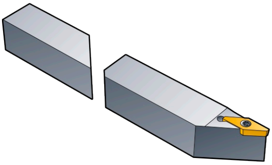
1. External turning
2. External turning (Sliding head machines)
3. Internal turning (Exchangeable inserts)
4. Internal turning
5. Internal turning (Carbide rods)

Overview of tool holders

External tools



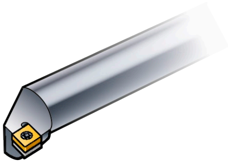
- Positive insert
- Screw clamping system
- Shank tools
- Precision coolant.



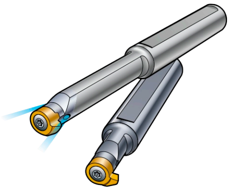
- Quick change tools
- Positive insert
- Screw clamping system.

- Positive insert
- Screw clamping system.

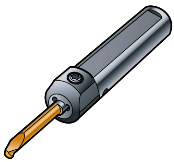
Internal tools



- Positive insert
- Screw clamping system
- Precision coolant.



- Positive insert
- Screw clamping system.

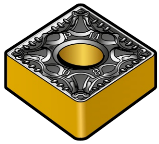


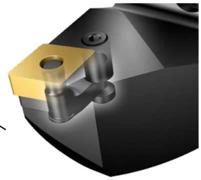
- Positive insert
- Carbide rods
- Machine adapted bars.

Overview of insert clamping systems

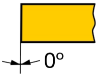
Clamping of negative basic-shape inserts

Rigid clamping system

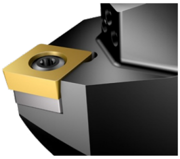




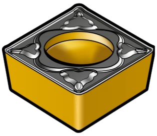
Lever clamping system

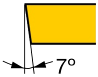


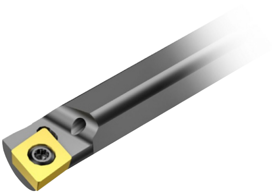
Clamping of positive basic-shape inserts



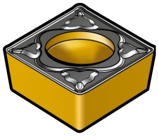
Screw clamping system

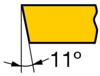






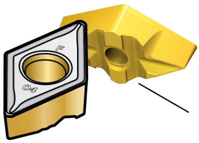
Screw clamping system



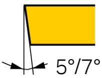


Clamping of positive iLock™ inserts

Screw clamping system

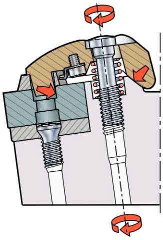


iLock™

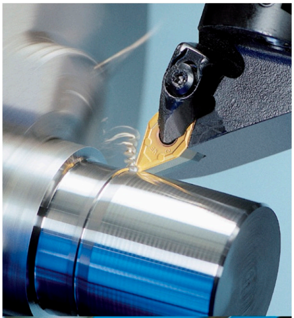


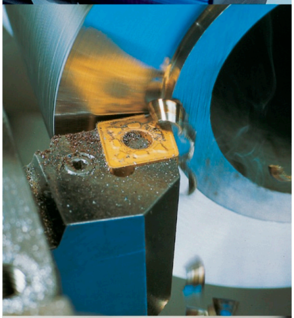
Modern insert clamping for turning tools

Rigid clamping

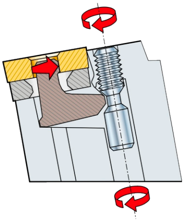


- Negative inserts
- Excellent clamping
- Easy indexing.



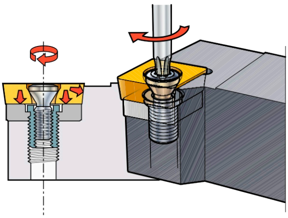


Lever clamping

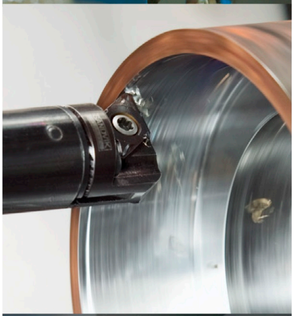


- Negative inserts
- Free chip flow
- Easy indexing.

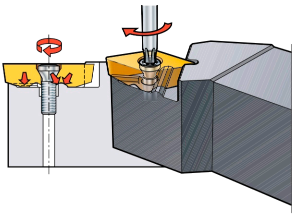
Screw clamping



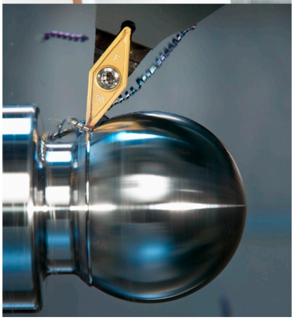
- Positive inserts
- Secure clamping of the insert
- Free chip flow.

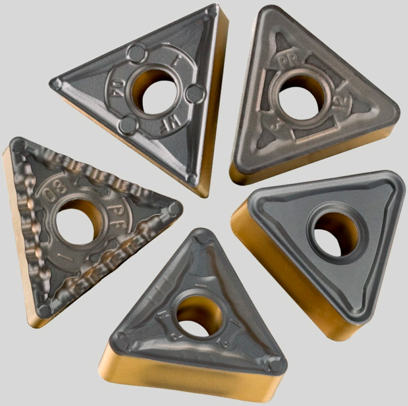


Screw clamping system, iLock™



- Positive inserts
- Very secure clamping
- High accuracy.

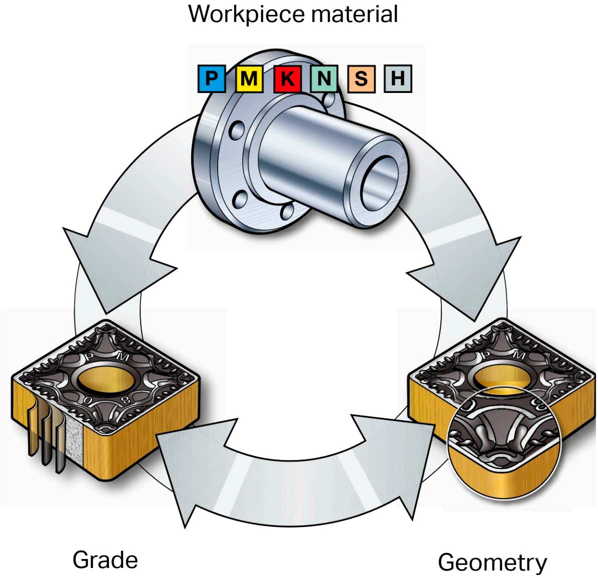




Choice of inserts

The complex world of metal cutting

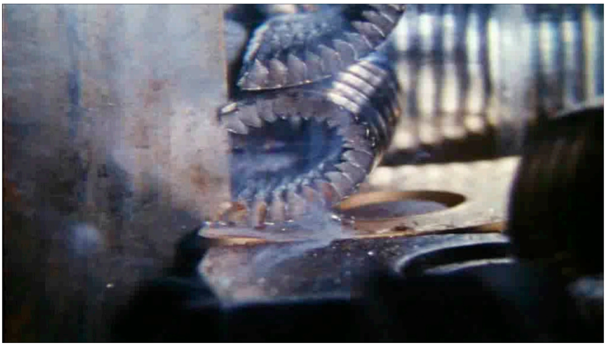
Getting metal cutting processes right means knowing the workpiece material, then choosing the correct insert geometry and grade to suit the specific application.



- The interaction between an optimized insert geometry and grade for a certain workpiece material is the key to successful machining.
- These three main basic factors must be carefully considered and adapted for the machining operation in question.
- The knowledge and understanding of how to work with and employ these factors is of vital importance.

The machining starts at the cutting edge

Typical chip breaking sequences with high speed imaging



Six material groups

In the metal cutting industry there is an incredibly broad range of component designs made from different materials. Each material has its own unique characteristics influenced by the alloying elements, heat treatment, hardness, etc. This strongly influences the selection of cutting tool geometry, grade and cutting data.

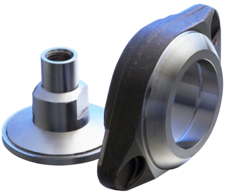
Workpiece materials are divided into 6 major groups in accordance with the ISO-standard, where each group has unique properties regarding machinability.

Workpiece material groups

P Steel

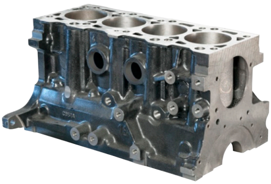
- ISO P – Steel is the largest material group in the metal cutting area, ranging from unalloyed to high-alloyed material including steel castings and ferritic and martensitic stainless steels. The machinability is normally good, but differs a lot depending on material hardness, carbon content, etc.

M Stainless steel



- ISO M – Stainless steels are materials alloyed with a minimum of 12% chromium; other alloys are, e.g., nickel and molybdenum. Different conditions such as ferritic, martensitic, austenitic and austenitic-ferritic (duplex), makes this an extensive material group. Common for all these types are that they expose cutting edges to a great deal of heat, notch wear and built-up edge.

K Cast iron



• ISO K – Cast iron is, contrary to steel, a short-chipping type of material. Gray cast iron (GCI) and malleable cast irons (MCI) are quite easy to machine, while nodular cast iron (NCI), compacted graphite iron (CGI) and austempered cast iron (ADI) are more difficult. All cast irons contain silicon carbide (SiC) which is very abrasive to the cutting edge.

N Aluminum

• ISO N – Non-ferrous metals are softer types of metals such as aluminum, copper, brass, etc. Aluminum with a silicon content (Si) of 13% is very abrasive. Generally high cutting speeds and long tool life can be expected for inserts with sharp edges.

S Heat resistant alloys

• ISO S – Heat Resistant Super Alloys include a great number of high-alloyed iron, nickel, cobalt and titanium-based materials. They are sticky, create built-up edge, workharden and generate heat, very similar to the ISO M-area, but they are much more difficult to cut, leading to shorter tool life for the cutting edges.

H Hardened steel

• ISO H – This group covers steels with a hardness between 45-65 HRc and also chilled cast iron around 400-600 HB. The hardness makes them all difficult to machine. The materials generate heat during cutting and are very abrasive to the cutting edge.

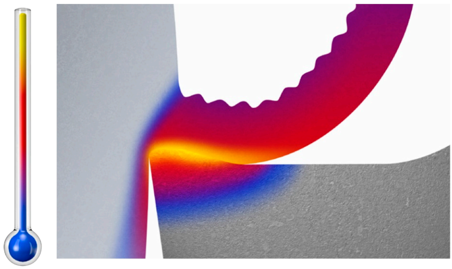
Cutting forces

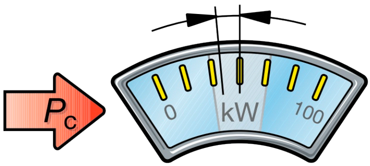
Another expression of the differences in the six material groups is through the force (F_T) needed to shear off a specific chip cross-section in certain conditions.

This value, the specific cutting force value (k_c), is indicated for various types of workpiece materials and used in the calculation of how much power is needed for an operation.

k_{c1} = specific cutting force for average chip thickness 1 mm (.039 inch).

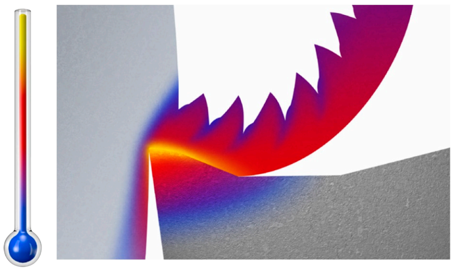
P Steel

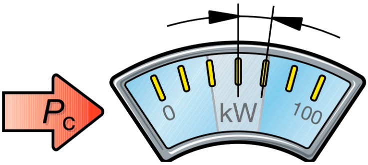




- P materials have a k_{c1} variation of: 1500-3100 N/mm² (217,500-449,500 lbs/inch²)

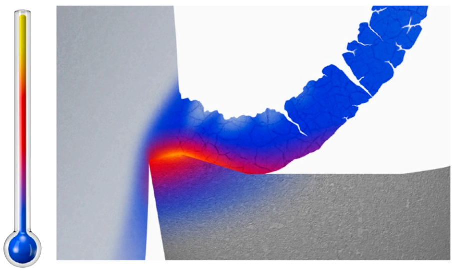
M Stainless steel

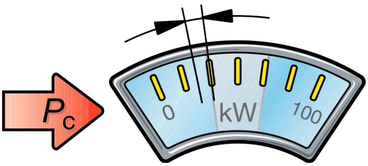




- M materials have *a* k_{c1} variation of: 1800-2850 N/mm² (261,000-413,250 lbs/inch²)

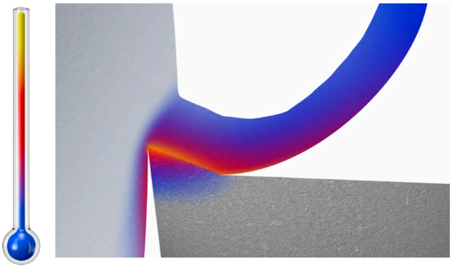
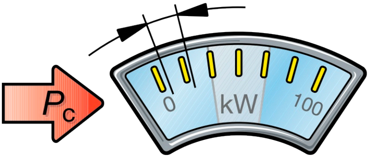
K Cast iron





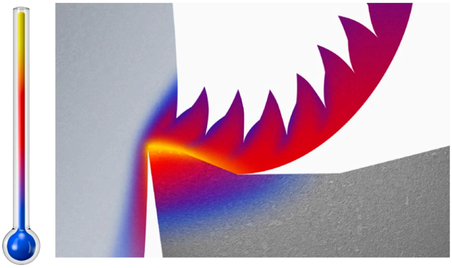
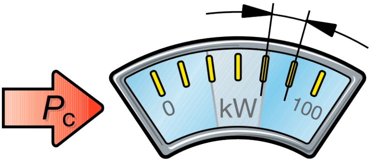
- K materials have a k_{c1} variation of: 790-1350 N/mm². (114,550-195,750 lbs/inch²)

N Aluminum

- N materials have a k_{c1} variation of:
  350-1350 N/mm²
  (50,750-195,750 lbs/inch²)

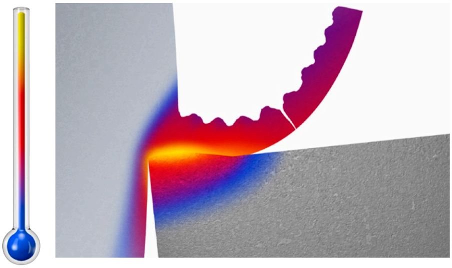
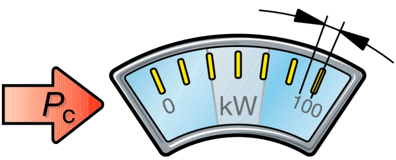
S Heat resistant super alloys

- S materials have a k_{c1} variation of:
  2400-3100 N/mm²
  (348,000-449,500 lbs/inch²) for HRSA

  1300-1400 N/mm²
  (188,500-203,000 lbs/inch²) for titanium alloys

H Hardened steel

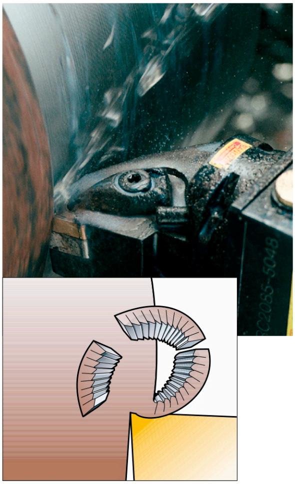



- H materials have a k_{c1} variation of:
  2550 – 4870 N/mm²
  (369,750-706,150 lbs/inch²)

Chip formation

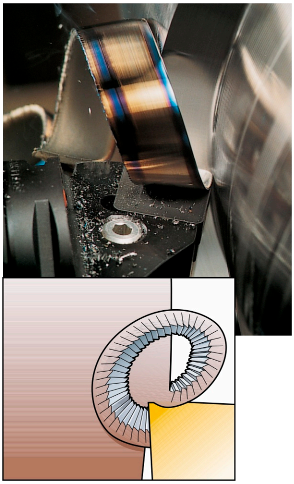
There are three patterns for a chip to break after it has been cut.

Self-breaking



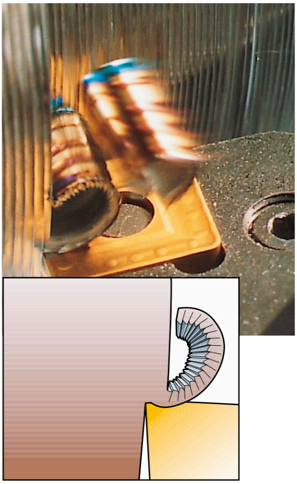
Self-breaking, where the material, in combination with how the chip is curved, leads to the chips being parted as they come off the insert.

Against the tool



Chips breaking against the tool, where the chip curves around until it makes contact with the clearance face of the insert or tool holder, and the resulting strain snaps it. Although often accepted, this method can in some cases lead to chip hammering, where the chip damages the insert.

Against the workpiece



Chips breaking against the workpiece, where the chip snaps when making contact with the surface that has just been machined. This type of chip breaking is usually not suitable in applications where a good surface finish is needed, because of possible damage caused to the component.

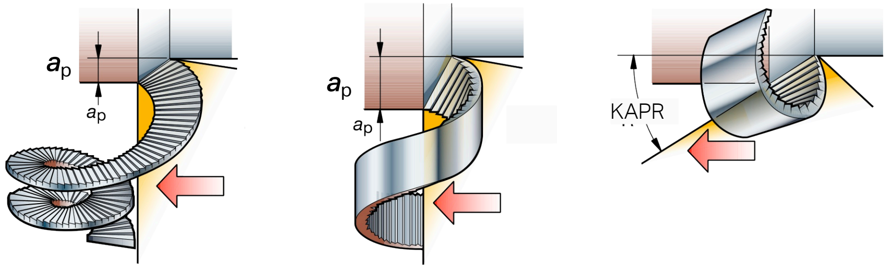
Chip formation varies with different parameters

Chip formation varies with depth of cut, feed, material and tool geometry.

Self-breaking

Against the tool

Against the workpiece

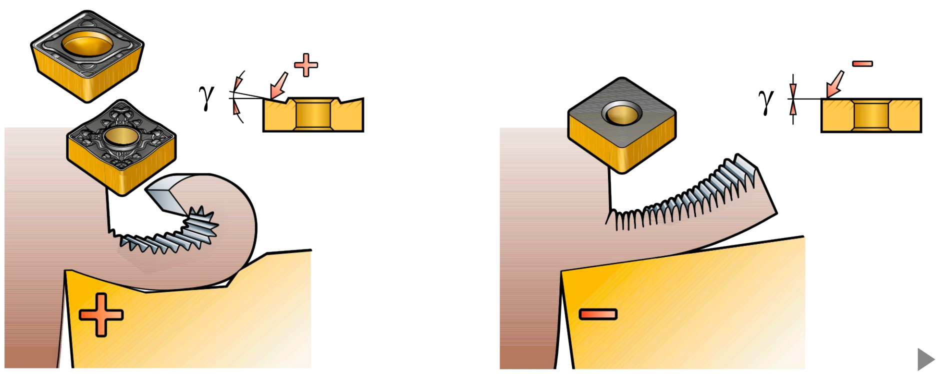


Insert rake angle

The rake angle (γ) gamma (GAMO) is a measurement of the edge in relation to the cut. This can be either negative or positive tools. Based on this, there are negative and positive inserts, where the clearance angles are either zero or several degrees plus. This determines how the insert can be tilted in the tool holder, giving rise to a negative or positive cutting action.

Positive cutting action

Negative cutting action

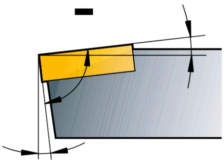


Insert rake angle

There is a distinction in cutting edge geometry between negative and positive insert geometry:
- A negative insert has a wedge angle of 90° seen in a cross-section of the basic shape of the cutting edge.
- A positive insert has an wedge angle of less than 90°.

The negative insert has to be inclined negatively in the tool holder so as to provide a clearance angle tangential to the workpiece while the positive insert has this clearance built in.

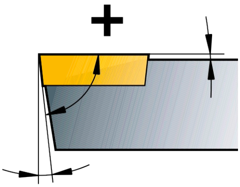
Negative style



- Double/single sided
- Edge strength
- Zero clearance
- External/internal machining
- Heavy cutting conditions.

Note: The clearance angle is the angle between the front face of the insert and the vertical axis of the workpiece.

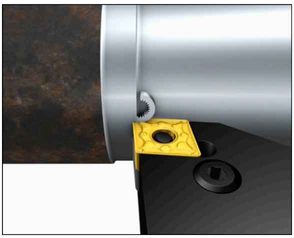
Positive style



- Single sided
- Low cutting forces
- Side clearance
- Internal/external machining
- Slender shafts, small bores.

Insert geometries

Metal cutting is very much the science of removing chips from the workpiece material in the right way. Chips have to be shaped and broken off into lengths that are manageable in the machine.



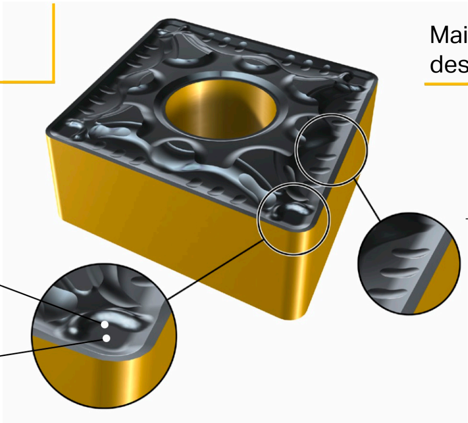
- In milling and drilling a lot of parameters influence the chip formation compared to turning.
- Turning is a single-cut operation with a stationary tool and a rotating workpiece.
- The insert rake angle, geometry and feed play an important role in the chip formation process.
- Removing heat from the cutting zone through the chip (80%) is a key issue.

The design of a modern insert

Definitions of terms and geometry design

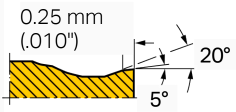
Nose radius edge design

Main cutting edge design



Macro geometry with chip breaker

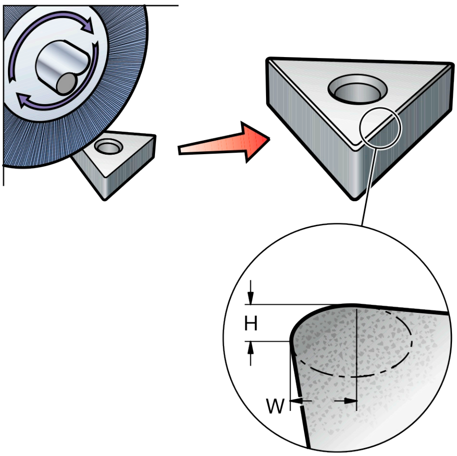
Geometry for small cutting depths



- Cutting edge reinforcement 0.25 mm (.010")
- Rake angle 20°
- Primary land 5°

The reinforcement of the cutting edge

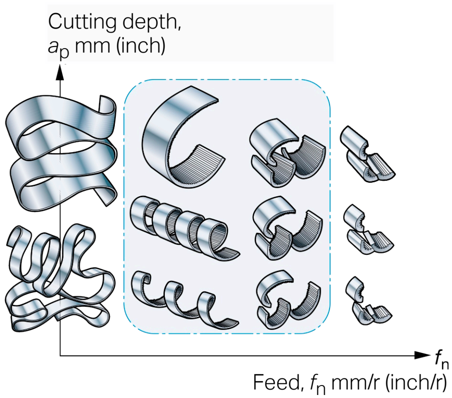
The Edge Roundness (ER) treatment gives the cutting edge the final micro-geometry.



- ER-treatment is done before coating, and gives the final shape of the cutting edge (micro-geometry).
- The relationship between W/H is what makes inserts suitable for different applications.

The working area of an insert geometry

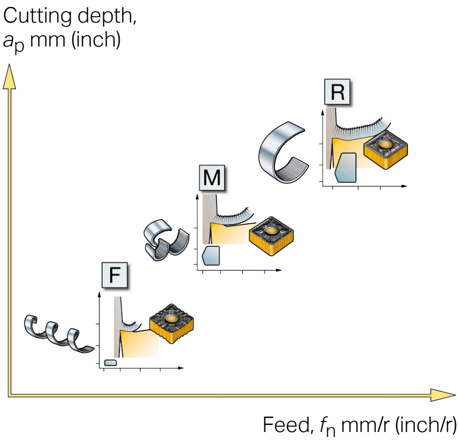
A chip breaking diagram for an insert geometry is defined by acceptable chip breaking for feed and depth of cut.



- Cutting depth (a_p) and feed (f_n) must be adapted to the chipbreaking area of the geometry to get acceptable chip control.
- Chip breaking which is too hard can lead to insert breakage.
- Chips which are too long can lead to disturbances in the machining process and bad surface finish.

Three main methods in Turning

R = Roughing
M = Medium machining
F = Finishing



Roughing

- Maximum metal removal rate and/or severe conditions
- Large cutting depth and feed rate combinations
- High cutting forces.

Medium machining

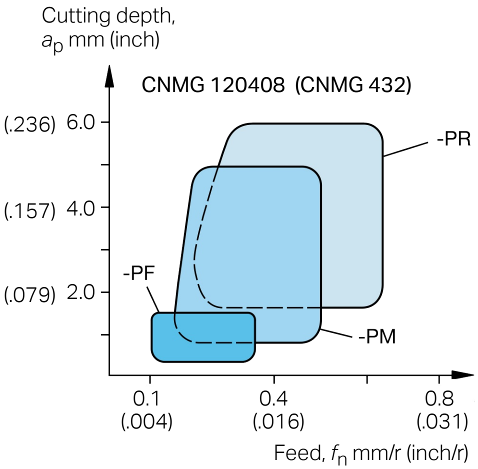
- Most applications – general purpose
- Medium operations to light roughing
- Wide range of cutting depth and feed rate combinations

Finishing

- Small cutting depths and low feed rates
- Low cutting forces.

Chip breaking areas

Turning of low alloy steel



Roughing – R
High depth of cut and feed rate combinations. Operations requiring the highest edge security.

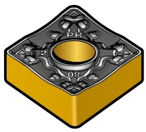
Medium – M
Medium operations to light roughing. Wide range of depth of cut and feed rate combinations.

Finishing – F
Operations at light depths of cut and low feed rates.
Operations requiring low cutting forces.

Chip breaking diagram

Roughing of steel
CMC 02.1

P R

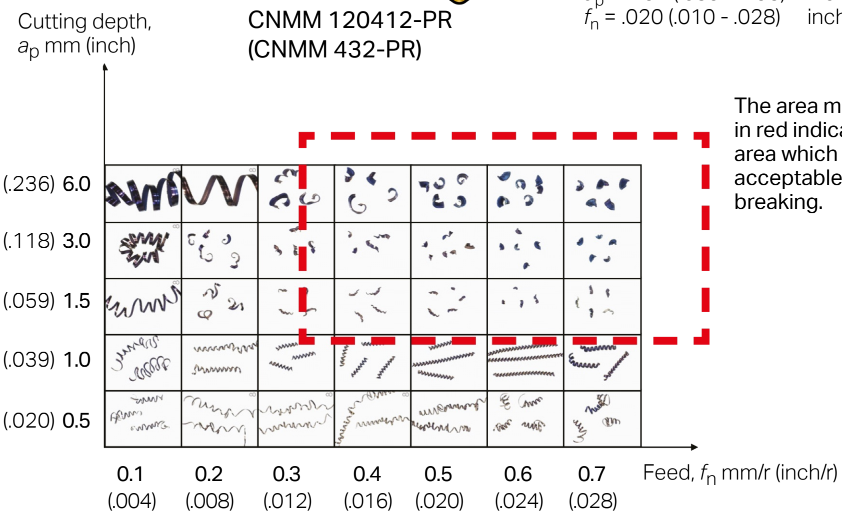


CNMM 120412-PR
(CNMM 432-PR)

Chip breaking area:

a_p = 5.0 (1.0 - 7.5) mm
f_n = 0.5 (0.25 - 0.7) mm/r

a_p = .197 (.039 - .295) inch
f_n = .020 (.010 - .028) inch/r

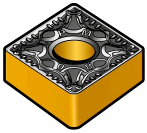


The area marked in red indicates the area which gives acceptable chip breaking.

A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

Medium machining of steel CMC 02.1

P M

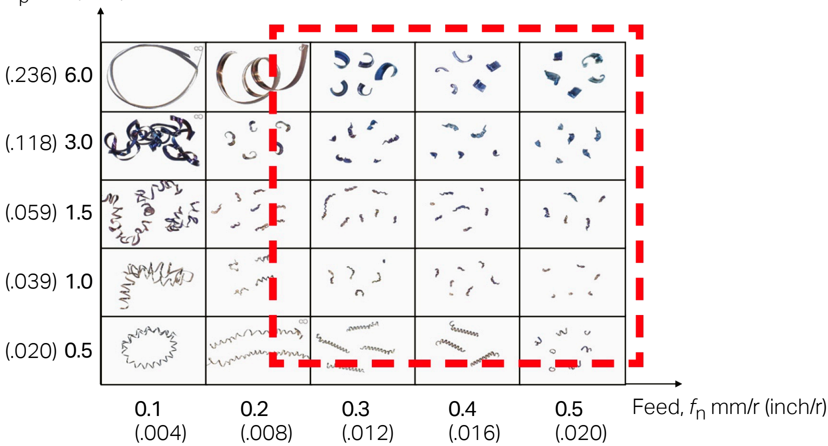


CNMG 120408-PM
(CNMG 432-PM)

Chip breaking area:

a_p = 3.0 (0.5 - 5.5) mm
f_n = 0.3 (0.15 - 0.5) mm/r

a_p = .118 (.020 - .217) inch
f_n = .012 (.006 - .020) inch/r



Finishing of steel CMC 02.1

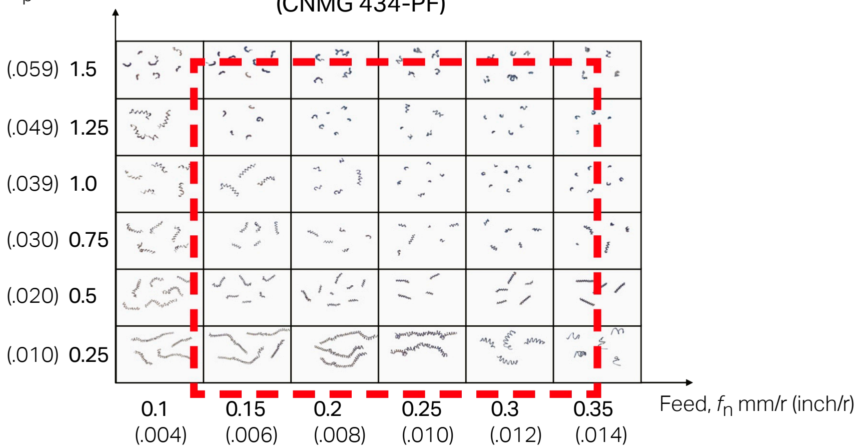
P F

CNMG 120404-PF
(CNMG 434-PF)

Chip breaking area:

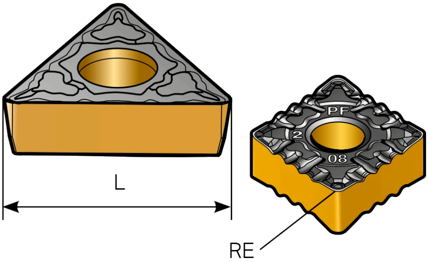
a_p = 0.4 (0.25 - 1.5) mm
f_n = 0.15 (0.07 - 0.3) mm/r

a_p = .016 (.010 - .059) inch
f_n = .006 (.003 - .012) inch/r



Selection of inserts

Considerations when selecting inserts



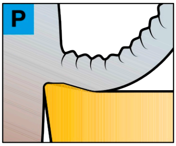
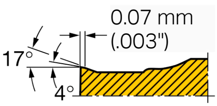
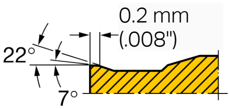
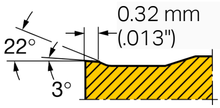
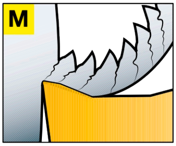
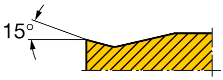
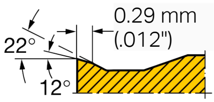
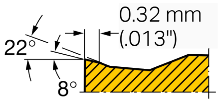
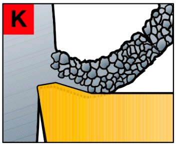
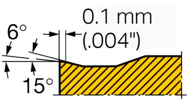
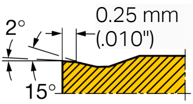
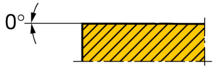
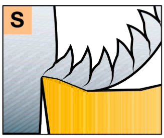
L= cutting edge length (insert size)
RE = nose radius

It is important to select the correct insert size, insert shape, geometry and insert nose radius to achieve good chip control.

- Select the largest possible point angle on the insert for strength and economy.
- Select the largest possible nose radius for insert strength.
- Select a smaller nose radius if there is a tendency for vibration.

Dedicated inserts for the ISO P, M, K and S area

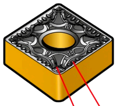
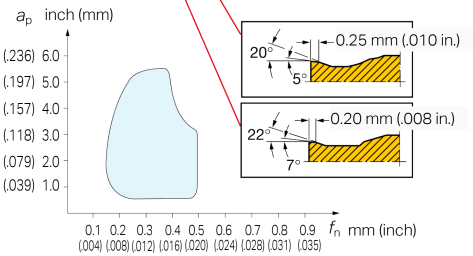
The different micro and macro-geometries are adapted to the various requirements in the applications.

Workpiece material	Finishing	Medium	Roughing
			
			
			
			

Geometry description

Every insert has a working area with optimized chip control.

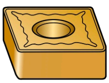
A geometry description and application information are also available.

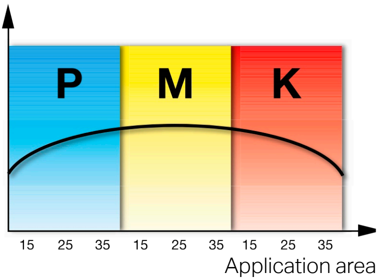
Geometry working area	Geometry description	Application
-PM  	CNMG 432-PM (CNMG 12 04 08-PM) a_p = 0.5 – 5.5 mm f_n = 0.15 – 0.5 mm/r a_p = .020 – .217 inch f_n = .006 – .020 inch/r	**-PM – for medium turning** with broad capability for steel. **Feed (f_n):** 0.1 – 0.65 mm/r (.004 – .026 inch/r). **Depth of cut (a_p):** 0.4 – 8.6 mm (.016 – .339 inch). **Operations:** turning, facing and profiling. **Advantages:** all-purpose, reliable, with problem-free machining. **Components:** axles, shafts, hubs, gears, etc. **Limitations:** depth of cut and feed, risk of over-loading the cutting edge. **General recommendations:** Combine with a wear resistant grade for best productivity. **Possible optimization:** geometry WMX.

From universal to optimized turning inserts

Universal inserts

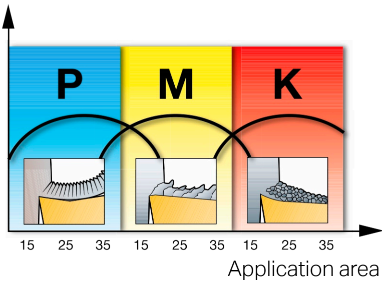
- Universal geometry
- Optimizing with grades
- Performance compromised.





Optimized inserts

- Dedicated geometries and grades
- Optimized performance according to workpiece material and machinability.

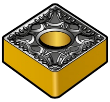


Inserts for general turning

The choice of different insert concepts

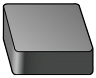
Negative, double/single-sided inserts

Plain inserts

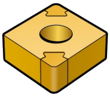


Double sided

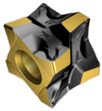
Single sided



Without hole



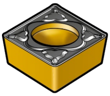
With hole



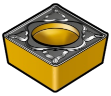
Concept multi-edge

- A negative insert has a wedge angle of 90° seen in a cross-section of the basic shape of the cutting edge.
- Available as double/single-sided inserts with P-hole or plain.

Positive, single-sided inserts

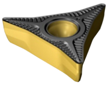


Positive 11°



Positive 7°

Positive iLock™ clamping



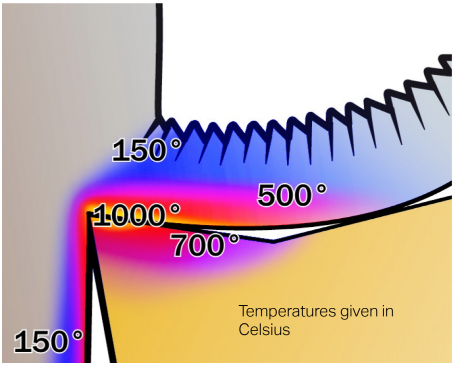
Concept iLock™ clamping

Concept

- A positive insert has a wedge angle less than 90°.
- Available with 7° or 11° clearance angle.
- The positive iLock™ inserts have a clearance angle of 5° or 7°.

Chip forming at high pressure and temperatures

The choice of cutting material and grade is critical for success



The ideal cutting tool material should:

- be hard to resist flank wear and deformation
- be tough to resist bulk breakage
- not chemically interact with the workpiece material
- be chemically stable to resist oxidation and diffusion
- have good resistance to sudden thermal changes.

The main range of cutting tool materials

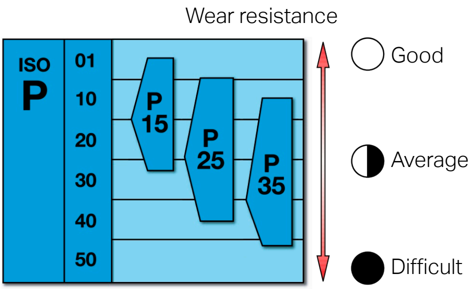
The most common cutting tool materials are divided into the following main groups:

- **Uncoated cemented carbide (HW)**
- **Coated cemented carbides (HC)**
- **Cermets (HT, HC)**
  - HT Uncoated cermet containing primarily titanium carbides (TiC) or titanium nitrides (TiN) or both.
  - HC Cermet as above, but coated.
- **Ceramics (CA, CM, CN, CC)**
  - CA Oxide ceramics containing primarily aluminum oxide (Al_2O_3).
  - CM Mixed ceramics containing primarily aluminum oxide (Al_2O_3) but containing components other than oxides.
  - CN Nitride ceramics containing primarily silicon nitride (Si_3N_4).
  - CC Ceramics as above, but coated.
- **Cubic boron nitrides (BN)**
- **Polycrystalline diamonds (DP, HC)**
  - DP Polycrystalline diamonds.
  - HC Polycrystalline diamonds, but coated.

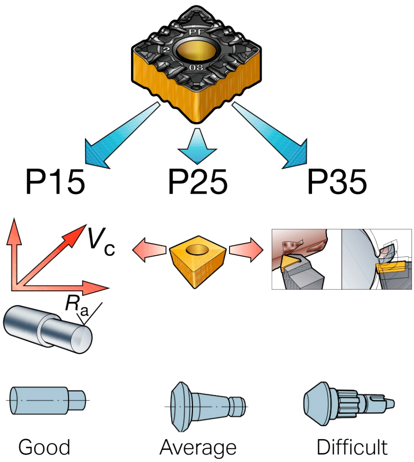
How to select insert geometry and grade

Select the geometry and grade according to the application.

Build up of a grade chart



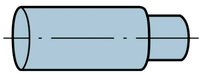
Machining conditions



Machining conditions

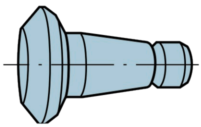
Good conditions

- Continuous cuts
- High speeds
- Pre-machined workpiece
- Excellent component clamping
- Small overhangs.



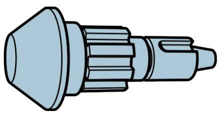
Average conditions

- Profiling cuts
- Moderate speeds
- Forged or cast workpiece
- Good component clamping.



Difficult conditions

- Interrupted cuts
- Low speeds
- Heavy cast or forged skin on workpiece
- Poor component clamping.



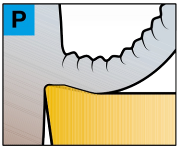
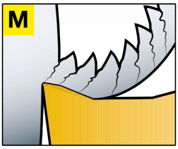
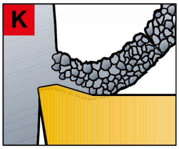
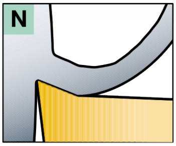
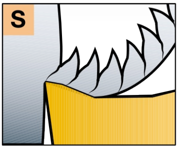
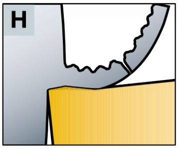
Dedicated grades

Dedicated grades minimize tool wear development

The workpiece material influences the wear during the cutting action in different ways. Therefore dedicated grades have been developed to cope with the basic wear mechanisms, e.g.:

- Flank wear, crater wear and plastic deformation

- Built-up edge and notch wear.

ISO P Steel	ISO M Stainless steel	ISO K Cast iron
		
ISO N Non-ferrous	**ISO S Heat resistant and super alloys**	**ISO H Hardened steel**
		

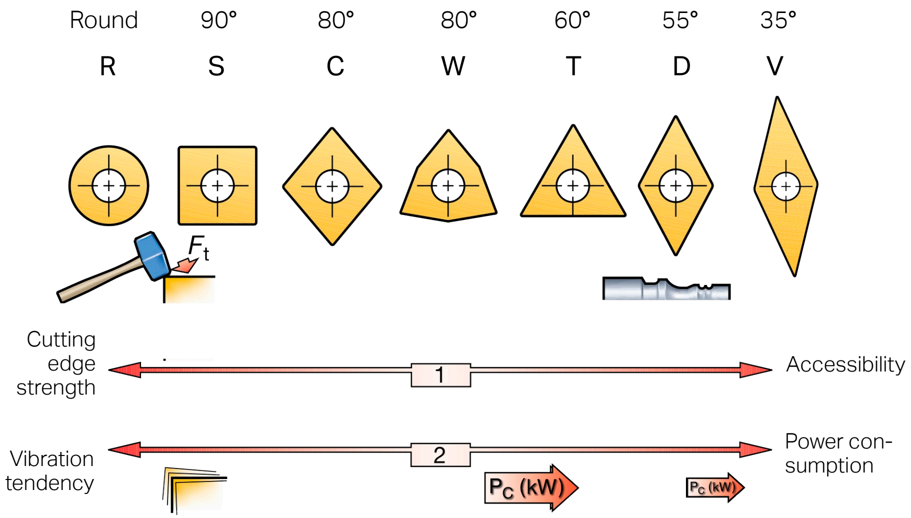
Selection of the insert shape

The influence of large and small point angle

The insert shape and point angle varies considerably from the smallest, at 35°, to the round insert.

Each shape has unique properties:
- some provide the highest roughing strength.
- others give the best profiling accessibility.

Each shape also has unique limitations. For example:
- high edge accessibility during machining leads to a weaker cutting edge.



Large point angle

- Stronger cutting edge
- Higher feed rates
- Increased cutting forces
- Increased vibration.

Small point angle

- Weaker cutting edge
- Increased accessibility
- Decreased cutting forces
- Decreased vibration.

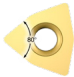
Factors affecting choice of insert shape

Insert shape should be selected relative to the entering (lead) angle accessibility required of the tool. The largest possible point angle should be applied to give insert strength and reliability.

Insert shape							
Roughing strength	++	++	++	+	+		
Light roughing/semi-finishing		+	++	+	++	++	
Finishing			+	+	++	++	++
Longitudinal turning			++	+	+	++	+
Profiling	+				+	++	++
Facing	+	++	++	+	+	+	
Operational versatility	+		++	+	+	++	+
Limited machine power			+	+	++	++	++
Vibration tendencies				+	++	++	++
Hard material	++	++					
Intermittent machining	++	++	+	+	+		

++ = Most suitable + = Suitable

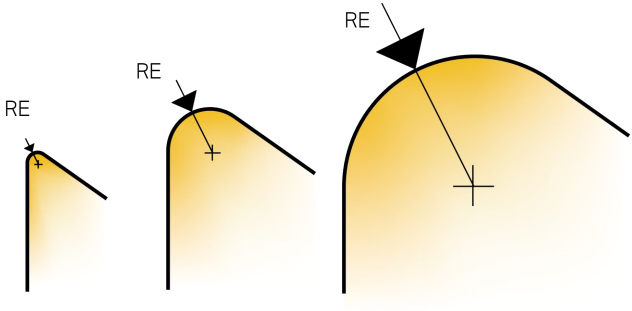
Number of cutting edges

Insert shape							
ISO (first letter)	R	S	C	W	T	D	V
Number of edges, negative inserts	8*	8	4	6	6	4	4
Number of edges, positive inserts	4*	4	2	3	3	2	2

*Depending on a_p

Selection of the Corner radius

Effect of small and large nose radius



Small corner radius

- Ideal for small cutting depth
- Reduces vibration
- Weak cutting edge.

Large corner radius

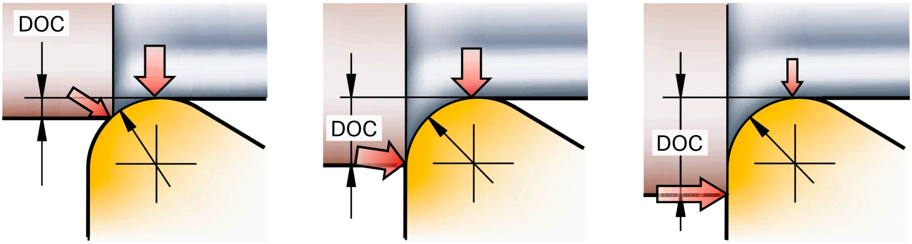
- Heavy feed rates
- Large depths of cut
- Strong edge security
- Increased radial pressures.

Rule of thumb

The depth of cut should be no less then the nose radius (RE).

A small nose radius should be first choice

With a small nose radius, the radial cutting forces can be kept to a minimum, while utilizing the advantages of a larger nose radius leads to a stronger cutting edge, better surface texture and more even pressure on the cutting edge.

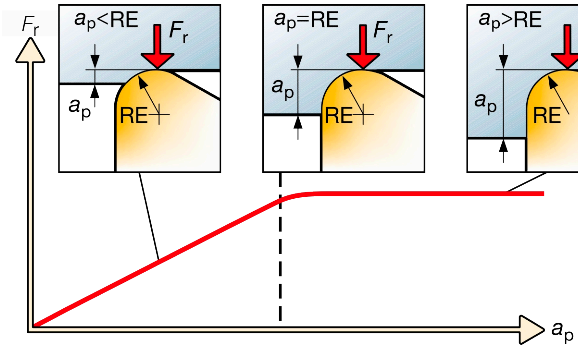


- The relationship between nose radius and DOC (depth of cut) affects vibration tendencies. It is often an advantage to choose a nose radius which is smaller than the DOC.

Effect of nose radius and DOC

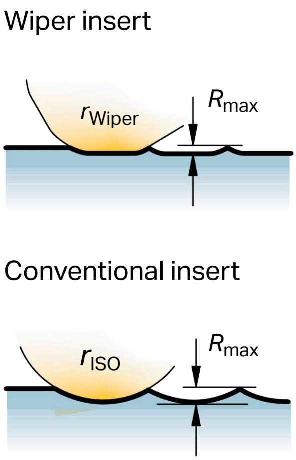
The radial force exerted on the workpiece grows linearly until the nose radius of the insert is less than the depth of cut where it stabilizes at the maximum value.

However with a round insert, radial pressure will never stabilize because the theoretical nose radius is half the insert diameter (IC).



High feed turning with wiper inserts

Wiper – General information



Why use a wiper

- Increase feed and gain productivity
- Use normal feed rate and gain surface quality.

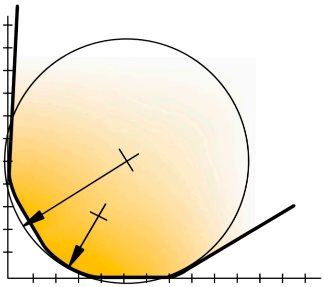
When to use wipers

- Use wipers as a first choice when it's possible.

Limitations

- General limitation is vibration
- Visually, surfaces can look different even though the measured surface is great.

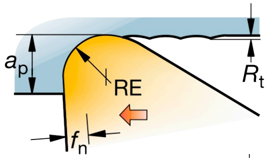
Wiper – Technical solution



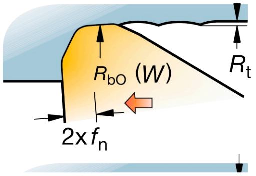
- One wiper cutting edge is based on 3-9 radii.
- Contact surface between insert and component is longer with wipers.
- Longer contact surface makes a better surface finish.
- Longer contact surface increases cutting forces which makes a wiper insert more sensitive to vibration when machining unstable components.

A conventional nose radius compared with a wiper nose radius.

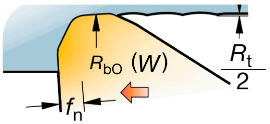
Wiper – Surface finish



Traditional insert



Wiper insert
Twice the feed, same R_a



Wiper insert
Same feed, half R_a

Wiper TECHNOLOGY

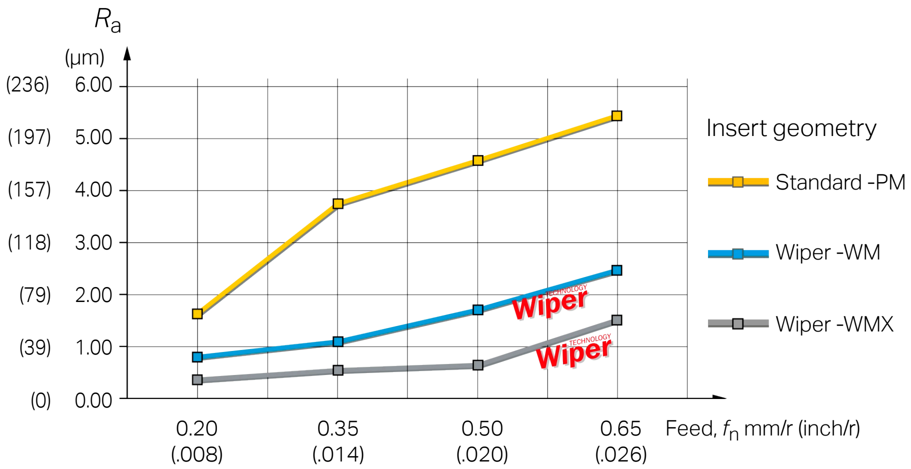
Rule of thumb

- Two times feed with a wiper will generate as good surface as conventional geometries with normal feed.
- The same feed with a wiper will generate twice as good surface compared with conventional geometries.

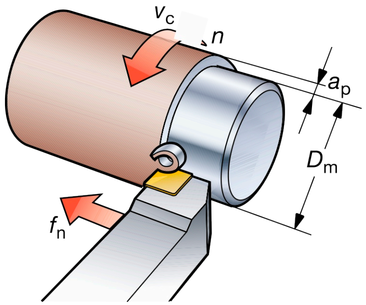
R_t = Maximum value peak-to-valley height

R_a = Arithmetic average height of the profile

Achieved surface – traditional ISO inserts and wipers

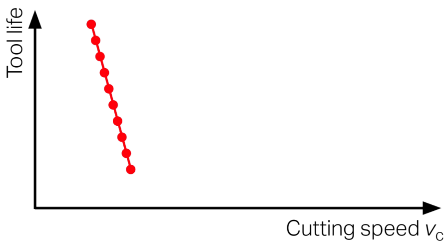


Cutting data parameters affect tool life



Use the potential of:

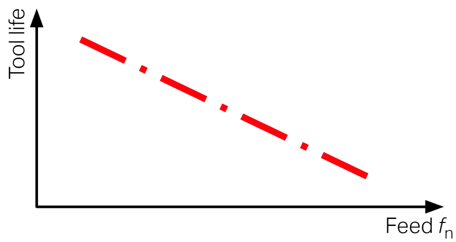
- a_p – to reduce number of cuts
- f_n – for shorter cutting time
- v_c – for best tool life



Cutting speed

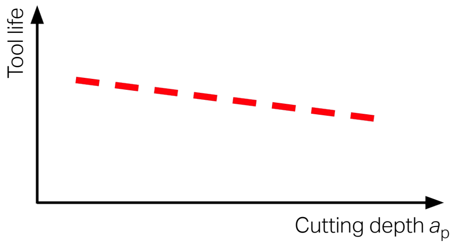
v_c – large effect on tool life.

Adjust v_c for best economy



Feed

f_n – less effect on tool life than v_c

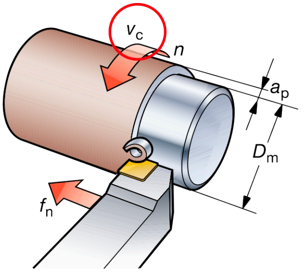


Cutting depth

a_p – little effect on tool life

Effects of cutting speed

The single largest factor determining tool life



Too high

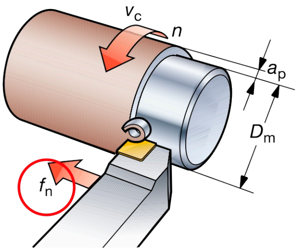
- Rapid flank wear
- Poor finish
- Rapid cratering
- Plastic deformation.

Too low

- Built-up edge
- Uneconomical.

Effects of feed rate

The single largest factor determining productivity



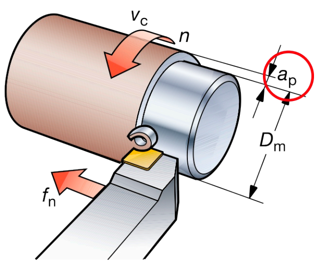
Too high

- Loss of chip control
- Poor surface finish
- Cratering, plastic deformation
- High power consumption
- Chip welding
- Chip hammering.

Too low

- Stringers
- Uneconomical.

Effects of depth of cut



Too deep

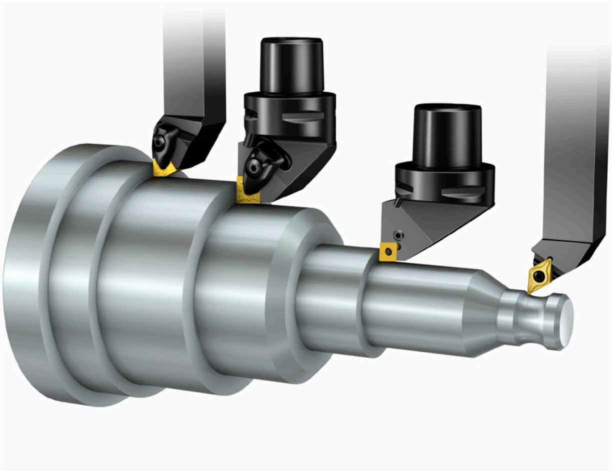
- High power consumption
- Insert breakage
- Increased cutting forces.

Too small

- Loss of chip control
- Vibrations
- Excessive heat
- Uneconomical.

External turning

Tool selection and how to apply

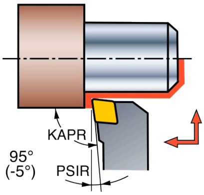


General guidelines

- Secure insert and tool holder clamping is an essential factor for stability in turning.
- Tool holder types are defined by the entering (lead) angle, the shape and size of the insert used.
- The selection of tool holder system is mainly based on the type of operation.
- Another important selection is the use of negative versus positive inserts.
- Whenever possible choose modular tools.

Four main application areas

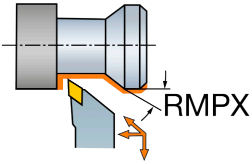
Longitudinal turning/facing



The most common turning operation

- Rhombic shape C-style (80°) insert is frequently used.
- Holders with entering angles of 95° and 93° (lead angles of –5° and –3°) are commonly used.
- Alternatives to the C-style insert are D-style (55°), W-style (80°) and T-style (60°).

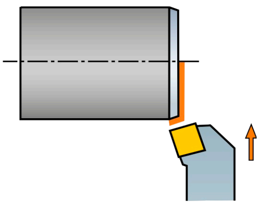
Profiling



Versatility and accessibility is the determining factor

- The effective entering angle KAPR (lead angle PSIR) should be considered for satisfactory machining.
- Most commonly used entering angle = 93° (lead angle is –3°) because it allows an in-copying angle between 22°-27°.
- The most frequently used insert shapes are D-style (55°) and V-style (35°) inserts.

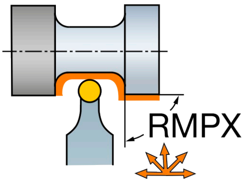
Facing



The tool is fed in towards the center

- Pay attention to the cutting speed which will change progressively when feeding towards the centre.
- Entering angles of 75° and 95°/91° (Lead angles of 15° and –5°/–1°) are commonly used.
- C-style (80°) and S-style (90°), inserts are frequently used.

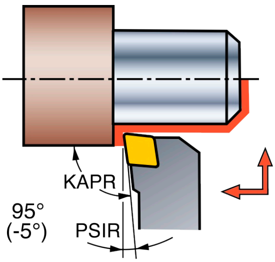
Pocketing



A method to produce or widen shallow grooves

- Round inserts are very suitable for plunge turning as they can be used for both radial and axial feeds.
- Neutral 90° holders for round inserts are commonly used.

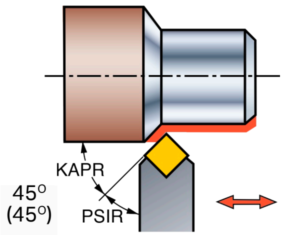
Large entering angle (small lead angle)



Features / Benefits

- Cutting forces directed towards chuck
- Can turn against a shoulder
- Higher cutting forces at entrance and exit of cut
- Tendency to notch in HRSA and hard materials.

Small entering angle (large lead angle)



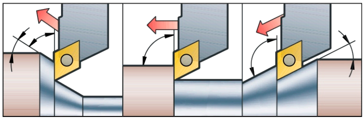
Features / Benefits

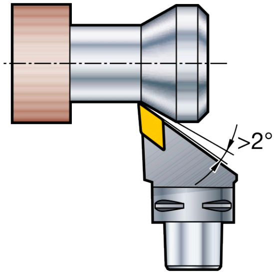
- Produces a thinner chip
  - Increased productivity
- Reduced notch wear
- Cannot turn against a shoulder.

The entering and copying angle

Important consideration in profile turning

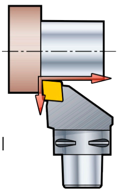
- In profile turning, the cut can vary with regard to cutting depth, chip thickness and speed.
- The largest suitable nose angle on the insert should be selected for strength and cost efficiency, but the insert nose angle also has to be considered in relation to accessibility for proper clearance between material and cutting edge.
- The most common nose angles used are 55° and 35°.
- The entering/lead and insert nose angle are both important factors for accessibility. The work piece profile has to be analyzed in order to select the most suitable copying angle.
- A free cutting angle of at least 2° between the workpiece and the insert has to be maintained.





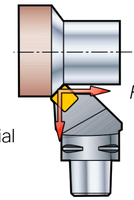
Axial and radial cutting forces

Large entering angle (small lead angle)



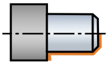
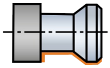
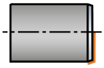
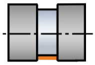
- Forces directed toward the chuck. Less tendency for vibration.
- Higher cutting forces especially at entrance and exit of cut.

Small entering angle (large lead angle)

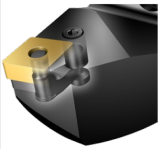
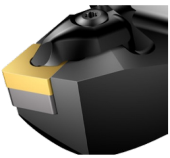
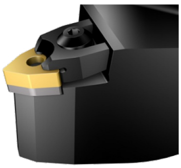
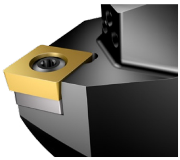


- Forces are directed both axially and radially.
- Reduced load on the cutting edge.
- Forces are directed both axially and radially - Vibration tendencies.

Insert recommendation depending on operation

Insert shape ++ = Recommended + = Alternative	Longitudinal turning 	Profiling 	Facing 	Pocketing 
 Rhombic 80°	++		+	
 Rhombic 55°	+	++	+	
 Round	+	+	+	++
 Square	+		++	
 Triangular	+	+	+	
 Trigon 80°	+		+	
 Rhombic 35°		+		

Selecting the insert clearance angle

Lever	Rigid clamping	Wedge clamping	Screw clamping	Concept clamping
				

Internal turning

Tool selection and how to apply

General guidelines

- In internal turning (boring operations) the choice of tool is very much restricted by the component's hole diameter and length.
  - Choose the largest possible bar diameter and the smallest possible overhang
  - Chip evacuation is a critical factor for successful boring
  - The clamping method has a decisive effect on the performance and result
  - Applying coolant can improve chip evacuation.

Selection factors

Tool and insert geometry

- Entering (lead) angle
- Insert shape, negative/ positive
- Insert geometry
- Nose radius
- Corner radius.

Chip evacuation

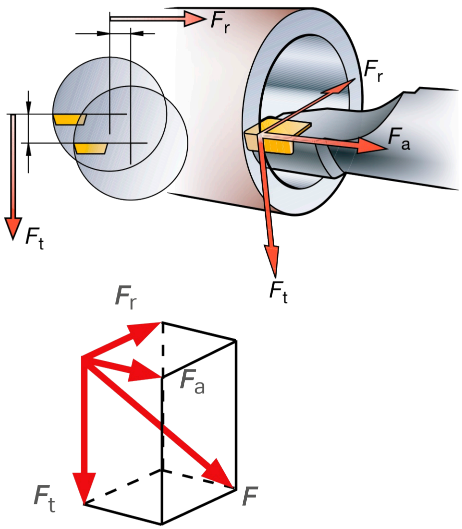
- Chip size
- Chip control
- Techniques
- Coolant.

Tool requirements

- Reduced length
- Increased diameters
- Optimized shape
- Different tool materials
- Clamping
- Dampened solutions.

Effect of cutting forces on internal turning

Radial and tangential cutting forces deflect the boring bar



Tangential cutting force, F_t

- Forces the tool down, away from the center line
- Gives a reduced clearance angle.

Radial cutting force, F_r

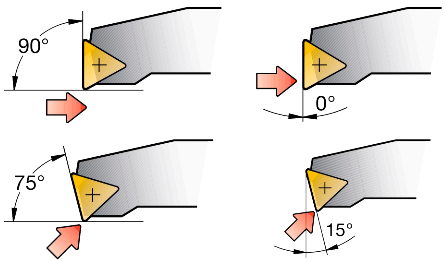
- Alters cutting depth and chip thickness
- Gives out of tolerance dimension and risk of vibration.

Feed force, F_a

- Directed along the feed of the tool.

Selecting entering (lead) angles

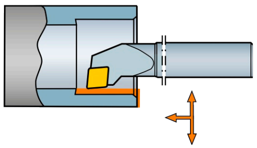
Entering (lead) angle and cutting forces



- Select an entering angle close to 90° (lead angle close to 0°).
- If possible, do not choose an entering angle less than 75° (lead angle not more than 15°), since this leads to a dramatic increase of the radial cutting force F_r.
  - Less force in radial direction = less deflection.

Four main application areas

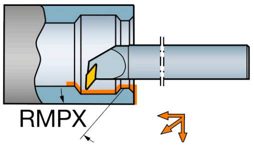
Longitudinal turning/facing



The most commonly used internal turning operation.

- Rhombic shape C-style 80° insert is frequently used.
- Boring bars with an entering (lead) angle of 95° (-5°) and 93° (-3°) are commonly used.
- D-style 55°, W-style 80° and T-style 60° insert shapes are also frequently used.

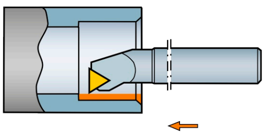
Profiling



Versatility and accessibility is the determining factor.

- The effective entering angle, KAPR (lead angle, PSIR) should be considered.
- Bars with entering (lead) angle of 93° (–3°), allowing an in-copy-ing angle between 22–27°, are commonly used.
- D-style 55° and V-style 35° inserts are frequently used.

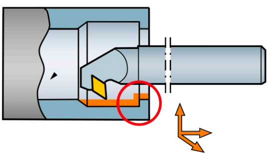
Longitudinal turning



Boring operations are performed to open up existing holes.

- An entering (lead) angle of close to 90° (0°) is recommended.
- Use smallest possible overhang.
- C-style 80°, S-style 90° and T-style 60° inserts are frequently used.

Back boring

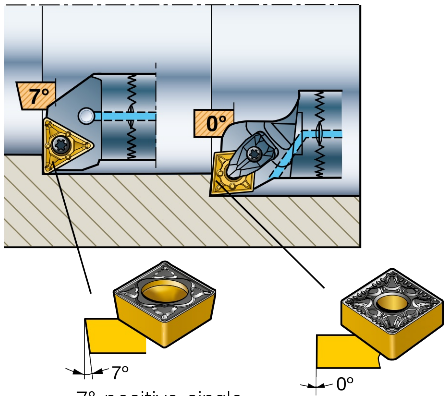


Back boring is a boring operation with reverse feed.

- It is used for turning shoulders less than 90°.
- Boring bars with 93° (–3°) entering (lead) angles and D-style 55° inserts are commonly used.

Selecting the insert clearance angle

Positive inserts generate lower cutting force and tool deflection

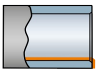
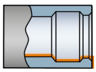
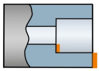


7°, positive, single sided inserts

Negative, double sided inserts

- Inserts with clearance angle 7°
  - First choice for small and medium holes from 6 mm (.236 inch) diameter.
- For best economy
  - Use negative inserts in stable conditions and with short overhang.

Insert recommendation depending on operation

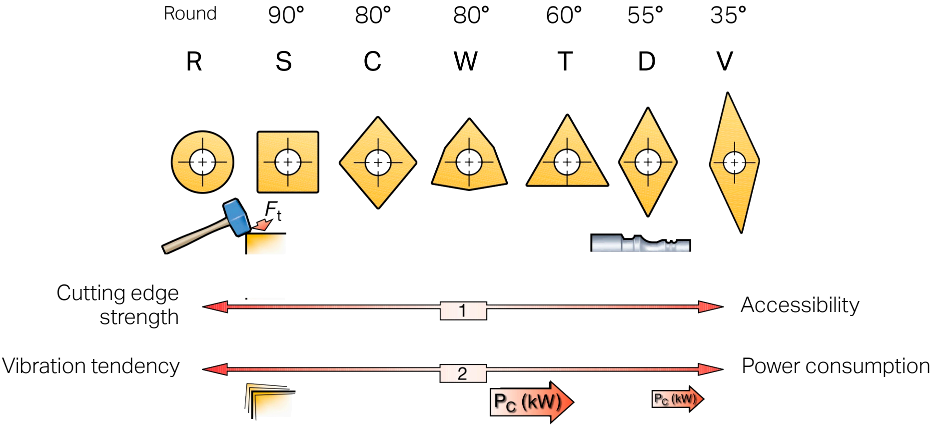
Insert shape ++ = Recommended + = Alternative	Longitudinal turning 	Profiling 	Facing 
 Rhombic 80°	+		++
 Rhombic 55°	+	++	+
 Round	+		+
 Square	+		
 Triangular	++		+
 Trigon 80°	+		+
 Rhombic 35°		+	

Insert point angle

Large point angle:
- Stronger cutting edge
- Higher feed rates
- Increases cutting forces
- Increases vibration

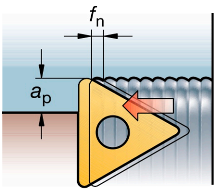
Small point angle:
- Increases accessibility
- Decreases vibration
- Decreases cutting forces

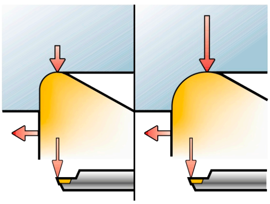
Use the smallest angle giving acceptable strength and economy.



Chip area and corner radius

Cutting forces and cutting tool deflection





- Both small and large chip areas can cause vibration:
  - Large due too high cutting forces
  - Small due too high friction between the tool and the workpiece.

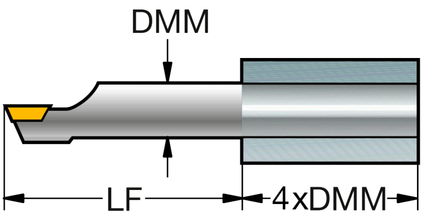
- The relationship between RE (nose radius) and a_p (depth of cut) affects vibration tendencies.
- Less force in radial direction = less deflection.

Rule of thumb!

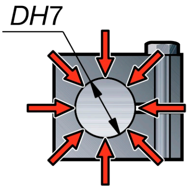
Choose a nose radius which is somewhat less than the cutting depth.

Clamping the boring bar

Critical stability factors for optimized performance

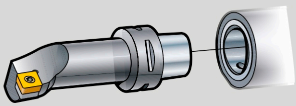


- Maximum contact between tool and tool holder (design, dimensional tolerance).
- Clamping length 3 to 4 times bar diameter (to balance cutting forces).
- Holder strength and stability.



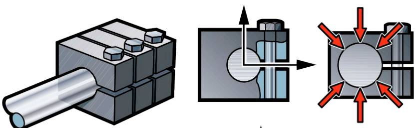
Tool requirements for clamping

Maximum contact between tool and tool holder

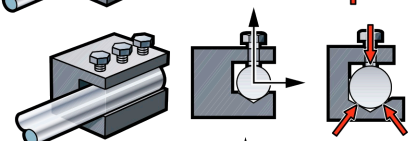


Coromant Capto® coupling

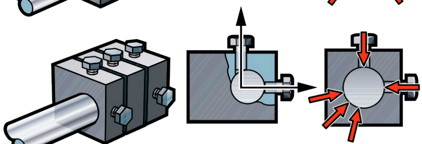
Recommended



Acceptable



Not recommended



Not recommended

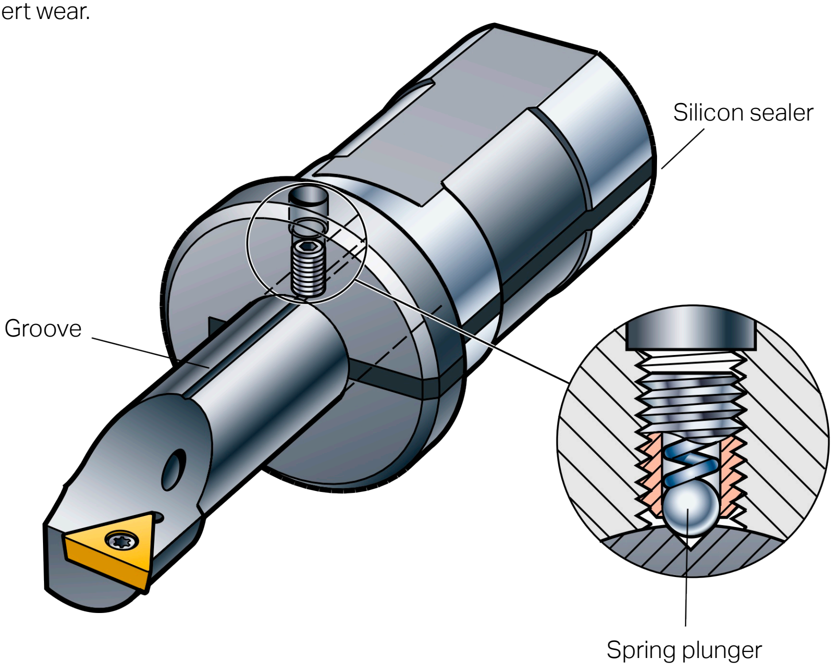
EasyFix sleeves

For correct clamping of cylindrical bars

Guarantees correct center height

Benefits:

- Cutting edge in right position
- Best cutting action gives better surface finish
- Reduced setup time
- Even insert wear.



A spring plunger mounted in the sleeve clicks into a groove in the bar and guarantees correct center height.

The slot in the cylindrical sleeve is filled with a silicon sealer which allows the existing coolant supply system to be used.

Factors that affect vibration tendencies

Vibration tendencies grow towards the right

Entering angle

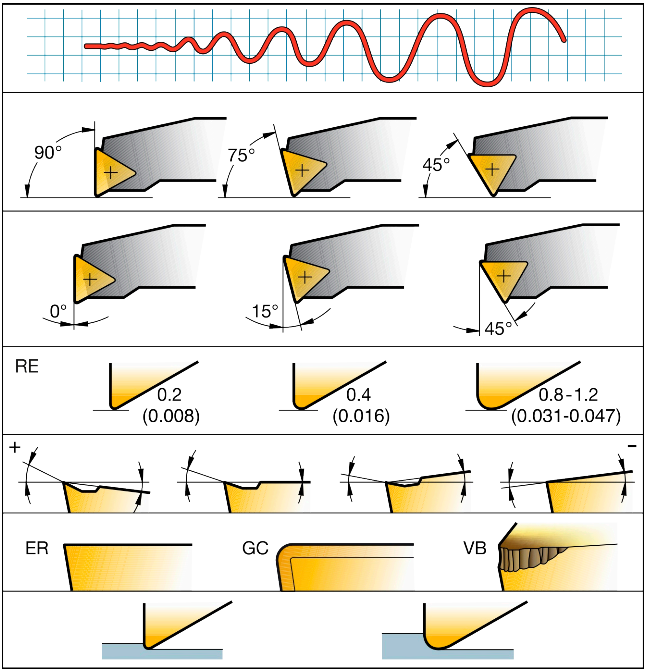
Lead angle

Corner radius

Micro and macro geometry

Edge design

Depth of cut (DOC)



Lead (entering) angle

- Choose an entering angle as close to 90° (lead angle as close to 0°) as possible, never less than 75° (more than 15° for lead angle).

Corner radius

- Choose a corner radius which is somewhat smaller than the cutting depth.

Micro and macro geometry

- Use a positive basic-shape insert, as these give lower cutting forces compared to negative inserts.

Edge design

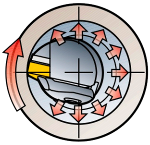
- Insert wear changes the clearance between the insert and the hole wall. This can affect the cutting action and lead to vibration.
- Inserts with thin coatings, or uncoated inserts, are to be preferred as they normally give lower cutting forces.

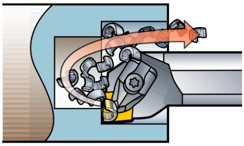
Depth of cut (DOC)

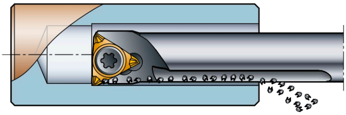
- Choose a corner radius which is somewhat smaller than the cutting depth.

Chip evacuation

Chip evacuation is a critical factor for successful boring

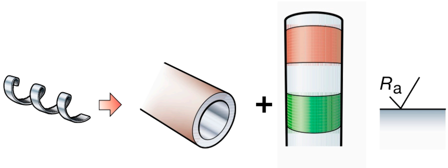






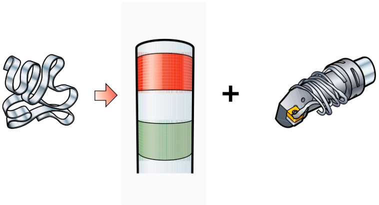
- Centrifugal force presses the chips to the inside wall of the bore.
- The chips can damage the inside of the bore.
  - Internal coolant can help with chip evacuation.
  - Boring upside down helps to keep chips away from the cutting edge.

Chip evacuation and chip control



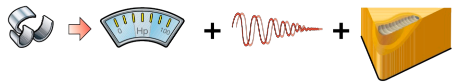
Short and spiral chips

- Preferred. Easy to transport and do not cause a lot of stress on the cutting edge during chip breaking.



Long chips

- Can cause chip evacuation problems.
- Causes little vibration tendency, but can in automated production cause problems due to chip evacuation difficulties.



Hard breaking of chips, short chips

- Power demanding and can increase the vibration.
- Can cause excessive crater wear and result in poor tool life and chip jamming.

Recommended tool overhang

Maximum overhang for different types of bars

Steel bar
– up to 4 x DMM

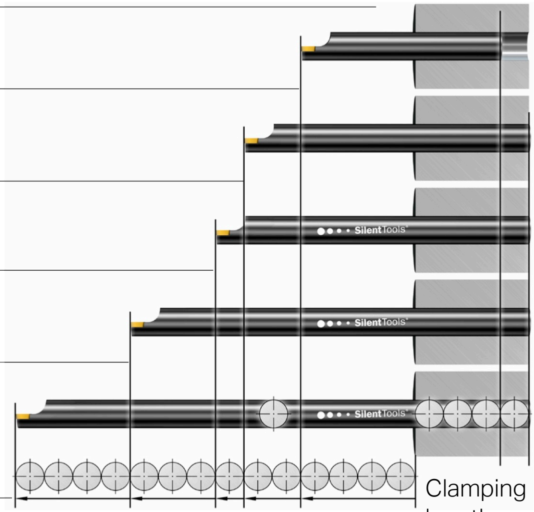
Carbide bar
– up to 6 x DMM

Short, dampened bar
– up to 7 x DMM

Long, dampened bar
– up to 10 x DMM

Carbide reinforced, dampened bar
– up to 14 x DMM

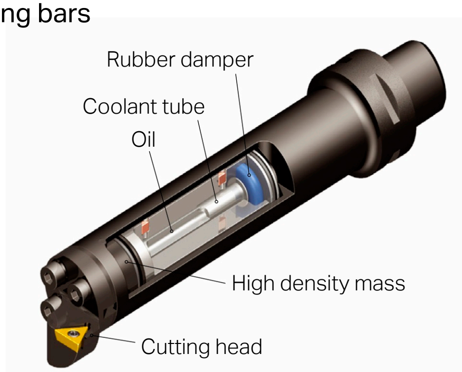
Overhang: ... x DMM



Eliminate vibrations

Internal machining with dampened boring bars

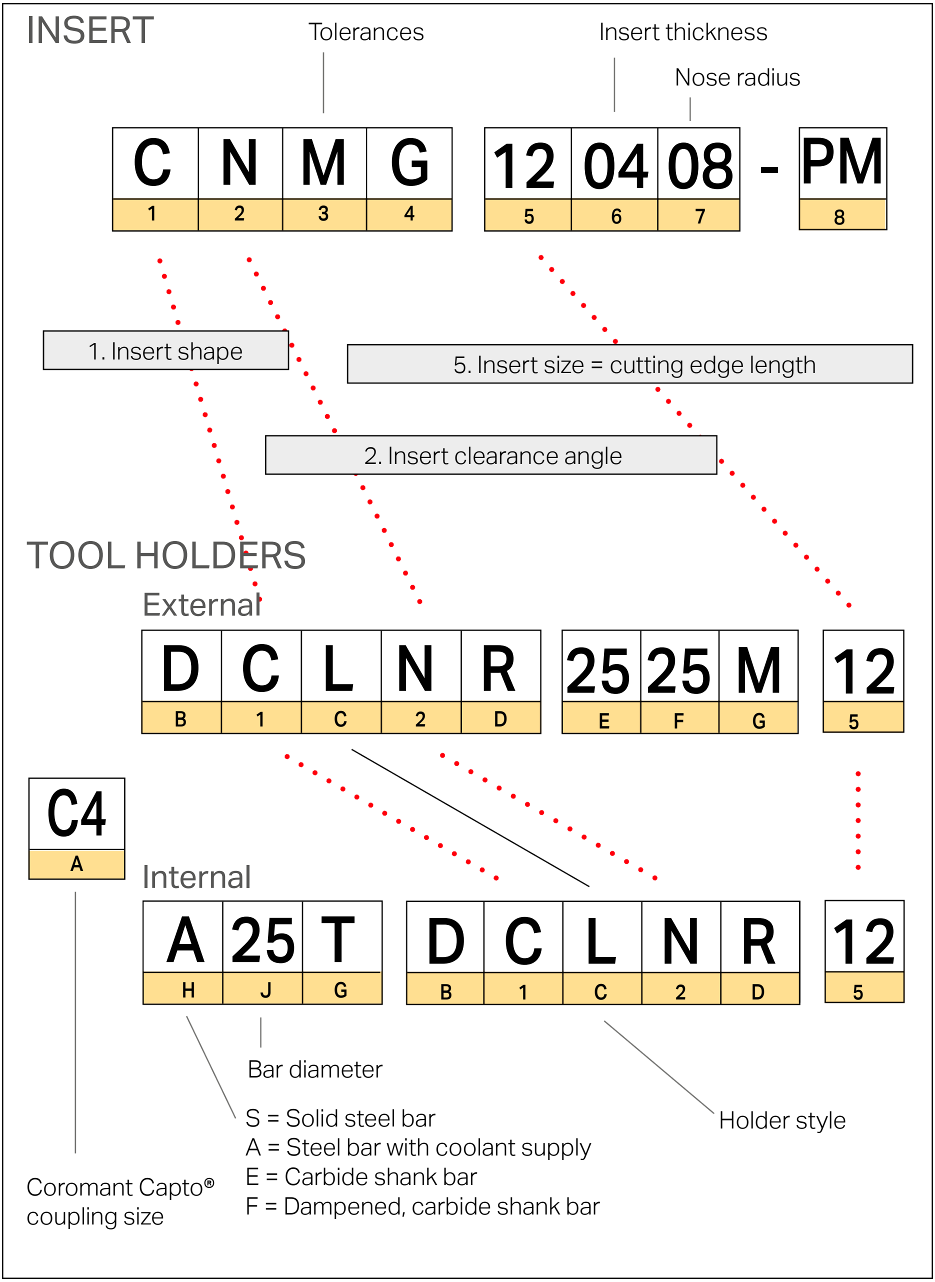
- Increase productivity in deep bores.
- Minimize vibration.
- Machining performance can be maintained or improved.
- Dampened boring bars are available in diameters from 10 mm (.394 inch).
- For max overhang 14 x DMM (carbide reinforced).



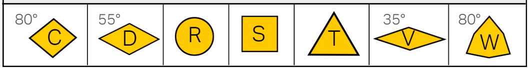
SilentTools®

Code key for inserts and toolholders - METRIC

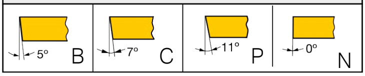
Extract from ISO 1832:1991



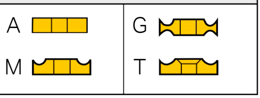
1. Insert shape



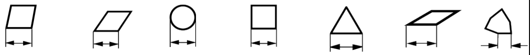
2. Insert clearance angle



4. Insert type

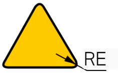


5. Insert size = Cutting edge length



l mm:	06–25	07–15	06–32	09–25	06–27	11–16	06–08

7. Nose radius



02	RE = 0.2
04	RE = 0.4
08	RE = 0.8
12	RE = 1.2
16	RE = 1.6
24	RE = 2.4

First choice nose radius recommendations:

	T-MAX P	CoroTurn 107
Finishing	08	04
Medium	08	08
Roughing	12	08

8. Geometry — manufacturer's option

The manufacturer may add a further two symbols to the code describing the insert geometry e. g.

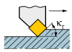
-PF = ISO P Finishing
-MR = ISO M Roughing

B. Clamping system

D	M	P	S
Rigid clamping (RC)	Top and hole clamping	Hole clamping	Screw clamping

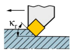
D. Hand of tool

R



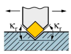
Right-hand style

L



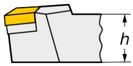
Left-hand style

N

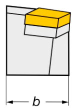


Neutral

E. Shank height

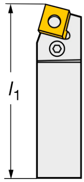


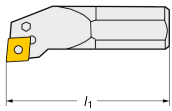
F. Shank width



G. Tool length

Tool length
= l_1 in mm

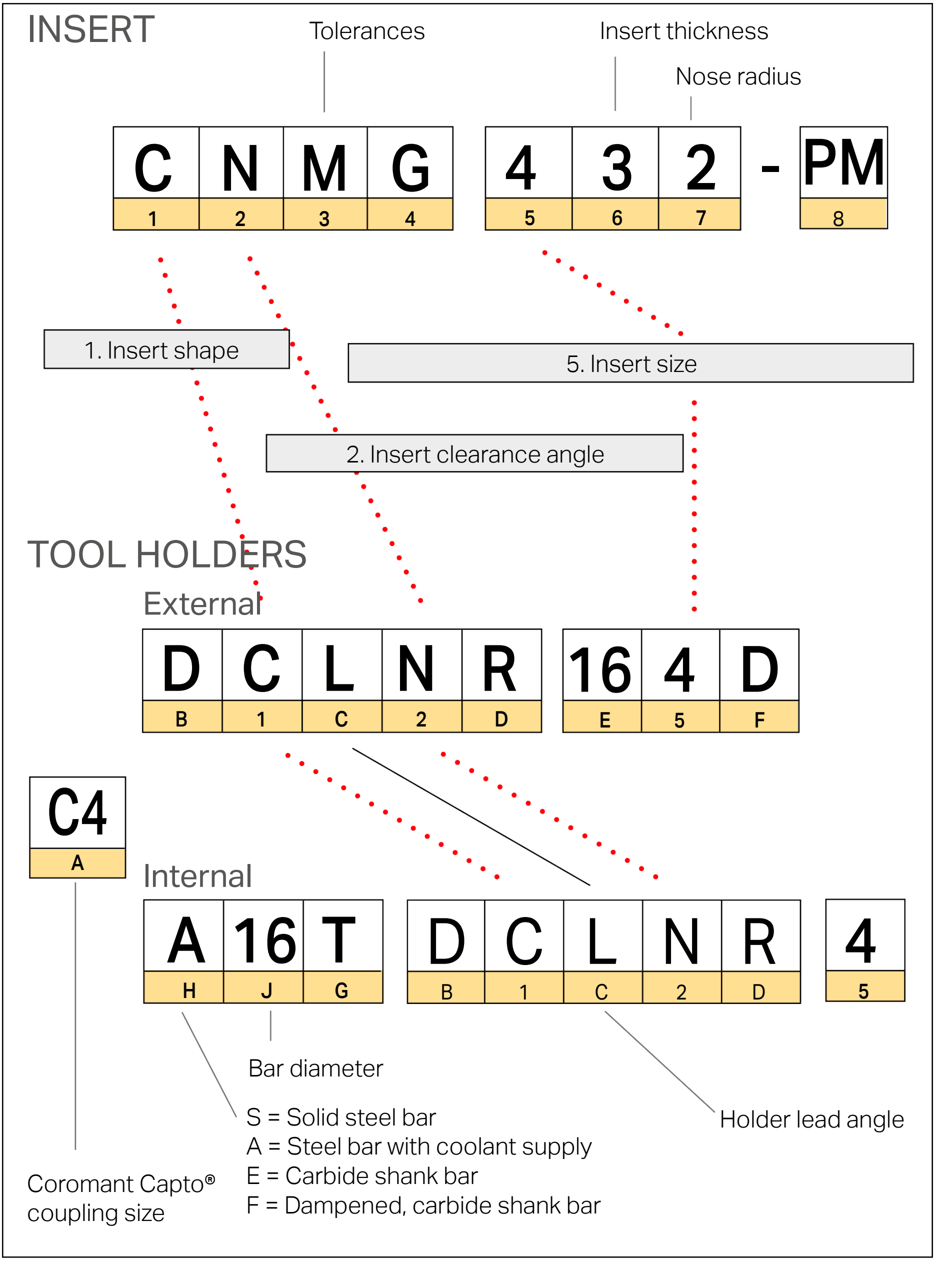




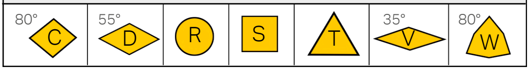
H = 100	S = 250
K = 125	T = 300
M = 150	U = 350
P = 170	V = 400
Q = 180	W = 450
R = 200	Y = 500

Code key for inserts and toolholders - INCH

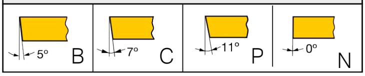
Extract from ANSI/ISO standards



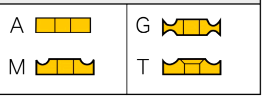
1. Insert shape



2. Insert clearance angle



4. Insert type



5. Insert size

Inscribed circle is indicated in 1/8"

7. Nose radius

0	RE = .008
1	RE = 1/64
2	RE = 1/32
3	RE = 3/64
4	RE = 1/16
6	RE = 3/32

First choice nose radius recommendations:

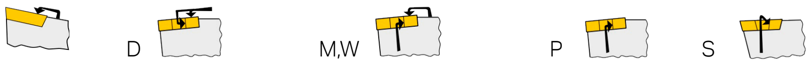
	T-MAX P	CoroTurn 107
Finishing	2	1
Medium	2	2
Roughing	3	2

8. Geometry — manufacturer's option

The manufacturer may add a further two symbols to the code describing the insert geometry e. g.

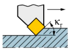
-PF = ISO P Finishing
-MR = ISO M Roughing

B. Clamping system

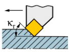


C Top clamping
D Rigid clamping (RC)
M,W Top and hole clamping
P Hole clamping
S Screw clamping

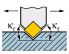
D. Hand of tool

R 

Right-hand style

L 

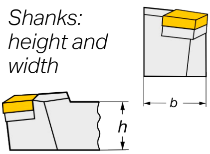
Left-hand style

N 

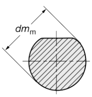
Neutral

E. Shank or bar size

Shanks: height and width

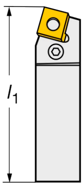


Bars:



G. Tool length

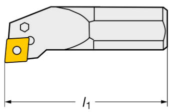
External, l_1 in inch



A = 4.0
B = 4.5
C = 5.0
D = 6.0

M = 4.0

Internal, l_1 in inch



M= 6.0
R= 8.0
S= 10.0
T= 12.0
U= 14.0

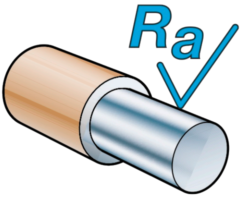
Troubleshooting

Chip control

Problem	Cause	Solution
Long unbroken snarls winding around the tool or workpieces.	• Feed too low for the chosen geometry.	• Increase the feed. • Select an insert geometry with better chip breaking capabilities. • Use a tool with high precision coolant.
	• Depth of cut too shallow for the chosen geometry.	• Increase the depth of cut or select a geometry with better chip breaking capability.
	• Nose radius too large.	• Select a smaller nose radius.
	• Unsuitable entering (lead) angle.	• Select a holder with as large entering angle (small lead angle) as possible KAPR =90° (PSIR =0°).
Very short chips, often sticking together, caused by too hard chip breaking. Hard chip breaking often causes reduced tool life or even insert breakages due to too high chip load on the cutting edge.	• Feed too high for the chosen geometry.	• Choose a geometry designed for higher feeds, preferably a single-sided insert. • Reduce the feed.
	• Unsuitable entering (lead) angle.	• Select a holder with as small entering angle (large lead angle) as possible KAPR =45°–75° (PSIR=45°–15°).
	• Nose radius too small.	• Select a larger nose radius.

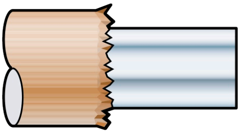
Surface finish

Problem	Cause	Solution
The surface looks and feels "hairy" and does not meet the tolerance requirements.	• The chips are breaking against the component and marking the finished surface.	• Select a geometry which guides the chips away. • Change entering (lead) angle. • Reduce the depth of cut. • Select a positive tool system with a neutral angle of inclination.
	• Hairy surface caused by excessive notch wear on the cutting edge.	• Select a grade with better resistance to oxidation wear, e.g., a cermet grade. • Reduce the cutting speed.
	• Too high feed in combination with too small nose radius generates a rough surface.	• Select a wiper insert or a larger nose radius. • Reduce the feed.

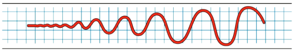
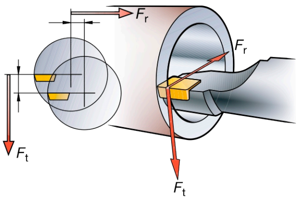


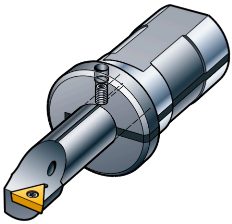
Burr formation

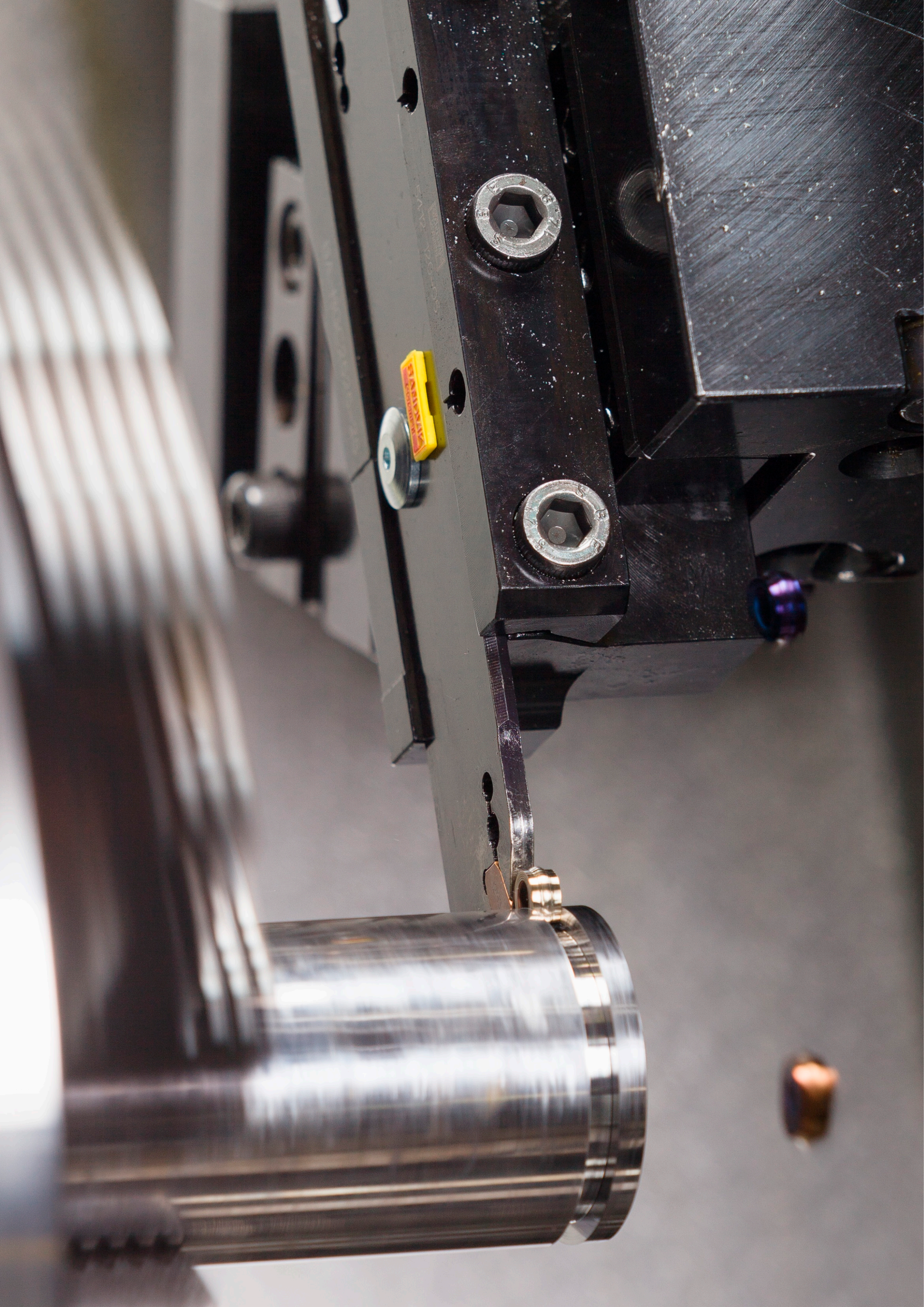
Problem	Cause	Solution
Burr formation at the end of the cut when the cutting edge is leaving the workpiece.	• The cutting edge is not sharp enough. • The feed is too low for the edge roundness.	• Use inserts with sharp edges: - PVD coated inserts. - ground inserts at small feed rates, < 0.1 mm/r (.004 inch/r).
	• Notch wear at depth of cut, or chipping.	• Use a holder with a small entering angle (large lead angle).
		• End the cut with a chamfer or a radius when leaving the workpiece.



Vibration

Problem	Cause	Solution
High radial cutting forces due to:  Vibrations or chatter marks which are caused by the tooling or the tool mounting. Typical for internal machining with boring bars.	• Unsuitable entering/lead angle.	• Select as large as possible entering angle (KAPR = 90°) or as small as possible lead angle (PSIR = 0°).
	• Nose radius too large.	• Select a smaller nose radius.
	• Unsuitable edge rounding, or negative chamfer.	• Select a more positive geometry or a grade with a thin coating, or an uncoated grade.
	• Excessive flank wear on cutting edge.	• Select a more wear resistant grade or reduce speed.
High tangential cutting forces due to: 	• Insert geometry creating high cutting forces.	• Select a positive insert geometry.
	• Chip-breaking is too hard giving high cutting forces.	• Reduce the feed or select a geometry for higher feeds.
	• Varying or too low cutting forces due to small depth of cut.	• Increase the depth of cut slightly to make the insert cut.
	• Tool incorrectly positioned.	• Check the center height.

Problem	Cause	Solution
	• Instability in the tool due to long overhang.	• Reduce the overhang. • Use the largest bar diameter. • Use a Silent Tool or a carbide bar.
	• Unstable clamping offers insufficient rigidity.	• Extend the clamping length of the boring bar. • Use EasyFix for cylindrical bars.



Parting & Grooving

Parting and grooving is a category of turning. It has a wide range of machining applications requiring dedicated tools.

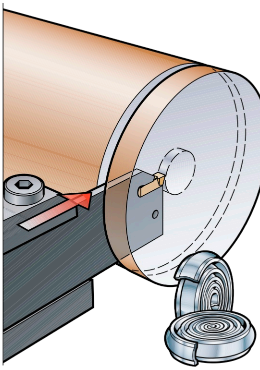
These tools can be used, to some extent, for general turning.

Parting & grooving theory

Parting off

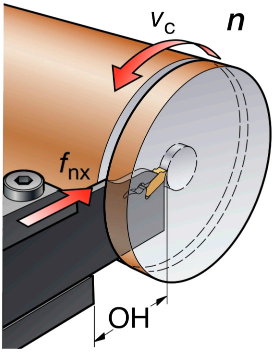
Chip evacuation is essential

Chip evacuation is a critical factor in parting operations. There is little opportunity to break chips in the confined space as the tool moves deeper. The cutting edge is designed largely to form the chip so it can be evacuated smoothly. Consequences of poor chip evacuation are chip obstruction, which leads to poor surface quality, and chip jamming, leading to tool breakdown.



- Chip evacuation is a critical factor in parting operations.
- Chip breaking is difficult in the confined slots created as tools cut deep into the workpiece.
- Typical chips are clock-spring shaped, narrower than the groove.
- The insert geometry shrinks the chip width.

Parting off – definition of terms



n = spindle speed (rpm)

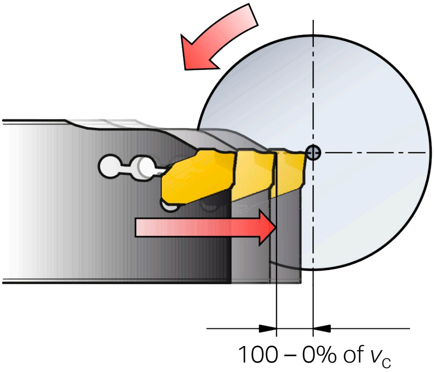
v_c = cutting speed m/min (ft/min)

f_{nx} = radial cutting feed mm/r (inch/r)

OH = overhang recommended

Cutting speed value

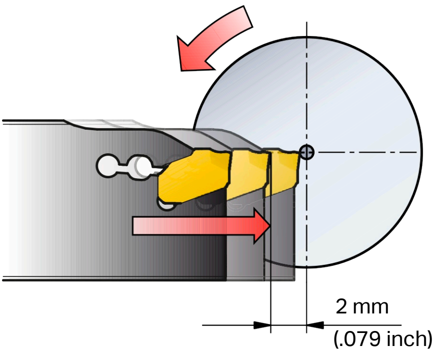
When parting off to center, the cutting speed will gradually be reduced to zero when the machine has reached its rpm limit.



- Cutting speed declines to zero at the center.

Feed reduction towards center

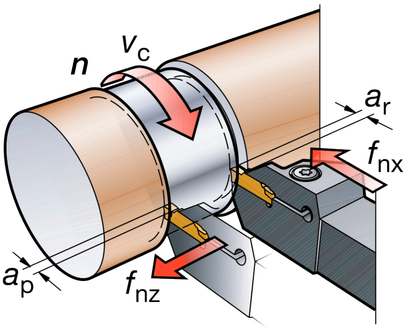
Cutting speed decreases toward the part center line, causing unbalance. Feed rate must be reduced to maintain cutting force balance during the part-off. The feed rate should be reduced to the minimum recommended or about 0.05 mm/rev (.002"/rev) at 2 mm (.079") before reaching centerline.



- Start cut with recommended feed rate, reference insert box
- Reduce feed to 0.05 mm/rev (.002"), 2 mm (.079") before centerline
- Feed reduction reduces vibration and increases tool life
- Feed reduction also reduces pip size.

Grooving– definition of terms

The tool movement in directions X and Z is called feed rate (f_n), or f_{nx}/f_{nz}, mm/r (inch/r). When feeding towards center (f_{nx}), the rpm will increase until it reaches the rpm limit of the machine spindle. When this limitation is passed, the cutting speed (v_c) will decrease until it reaches 0 m/min (ft/min) at the component center.



n = spindle speed (rpm)

v_c = cutting speed m/min (ft/min)

f_{nz} = axial cutting feed mm/r (inch/r)

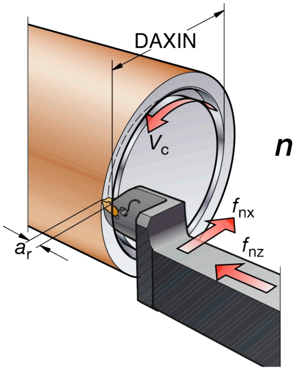
f_{nx} = radial cutting feed mm/r (inch/r)

a_r = depth of groove mm (inch) (outer dia. to center or bottom of groove)

a_p = depth of cut in turning

Face grooving– definition of terms

The feed has a great influence on chip formation, chip breaking, and thickness, and also influences how chips form in the insert geometry. In sideways turning or profiling (f_{nz}), the depth of the cut (a_p) will also influence chip formation. The groove diameter for the first cut must be within the range specified on the used tool holder.



n = spindle speed (rpm)

v_c = cutting speed m/min (ft/min)

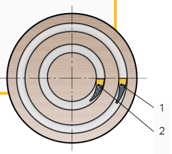
f_{nz} = axial cutting feed mm/r (inch/r)

f_{nx} = radial cutting feed mm/r (inch/r)

a_r = depth of groove mm (inch)

DAXIN = minimum first groove diameter (2 on this illustration)

DAXX = maximum first groove diameter (1 on this illustration)

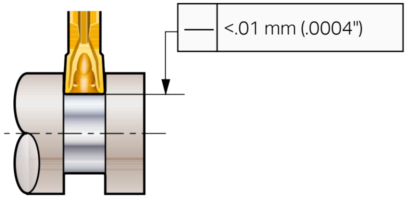


Tool selection procedure

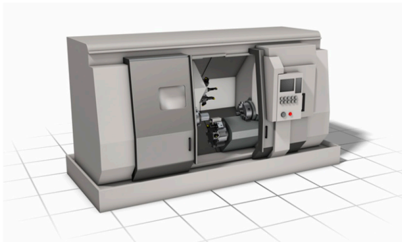
Production planning process

1 **Component**

Dimension and quality of the groove or face



Workpiece material, chip evacuation

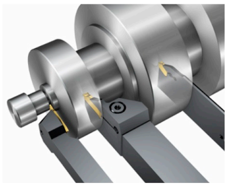


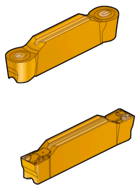
2 **Machine**

Machine parameters

3 **Choice of tool**

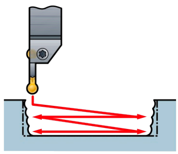
Type of tool:
- Spring-clamp
- Screw-clamp
- Insert type

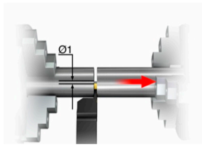




4 **How to apply**

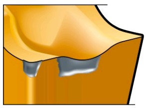
Cutting data, method, cutting fluid, etc

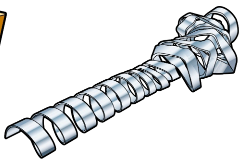




5 **Troubleshooting**

Remedies and solutions

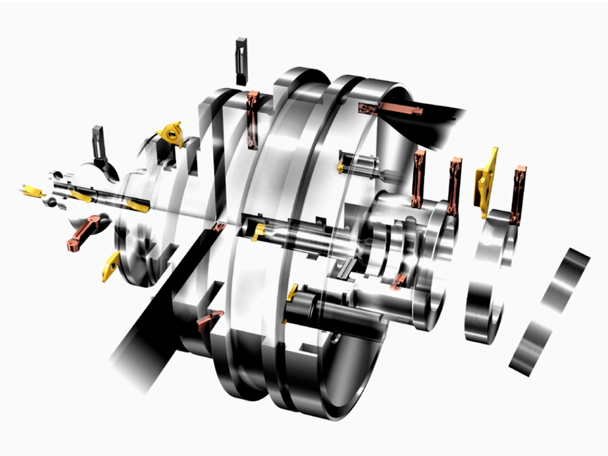




A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

1. Component and the workpiece material

Parameters to be considered



Component

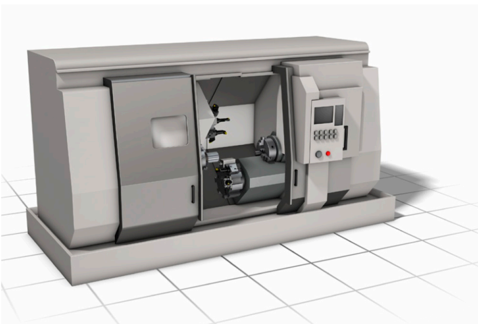
- Analyze the dimensions and quality demands of the groove or face to be machined
- Type of operation: parting, grooving
- Cutting depth
- Cutting width
- Corner radius.

P	M	K
N	S	H

Material

- Machinability
- Chip breaking
- Hardness
- Alloy elements.

2. Machine parameters



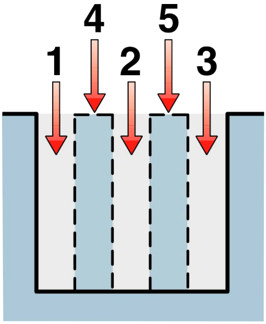
Some important machine considerations:

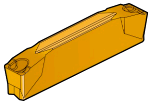
- Stability, power and torque especially for larger diameters
- Component clamping
- Turret interface
- Tool changing times/number of tools in turret
- Chip evacuation
- Cutting fluid and coolant.

3. Choice of tools

Example of different machining methods

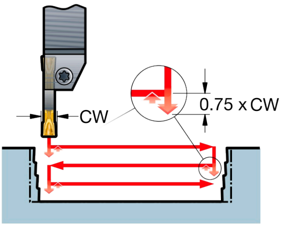
Multiple grooving

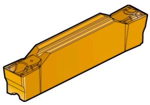




- Multiple grooving is the best method for rough grooving when the depth is bigger than the width.
- Make a “fork”. This will improve chip flow and increase tool life.

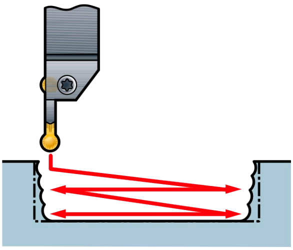
Plunge turning

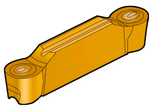




- Plunge turning is the best choice when machining steel and stainless steel and when the width of the groove is larger than the depth.
- Good chip control.

Ramping

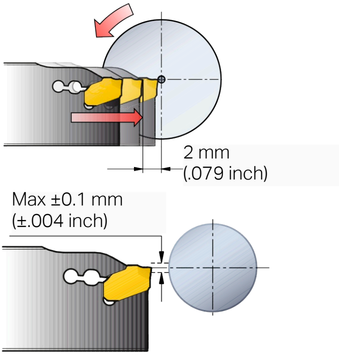


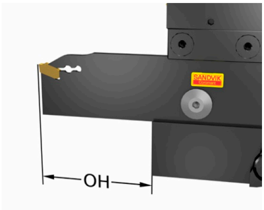


- Ramping avoids vibration and minimizes radial forces.
- Round inserts are the strongest inserts available.
- Double the number of cuts/passes.
- First choice in heat resistant super alloys (HRSA). Reduces notch wear.

4. How to apply

Application considerations

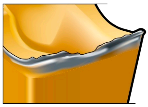


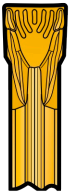


- Center height is important, ±0.1 mm (±.004 inch).
- Recommended feed rate 0.05 mm (.002 inch) / rev approximately 2 mm (.079 inch) before center.
- Use shortest possible overhang, OH mm (inch).
- Largest height dimension on blade for bending stiffness.
- Use coolant to improve chip flow.

5. Troubleshooting

Some areas to consider





Insert wear and tool life

- Check the wear pattern and if necessary adjust cutting data accordingly.

To improve chip formation & tool wear

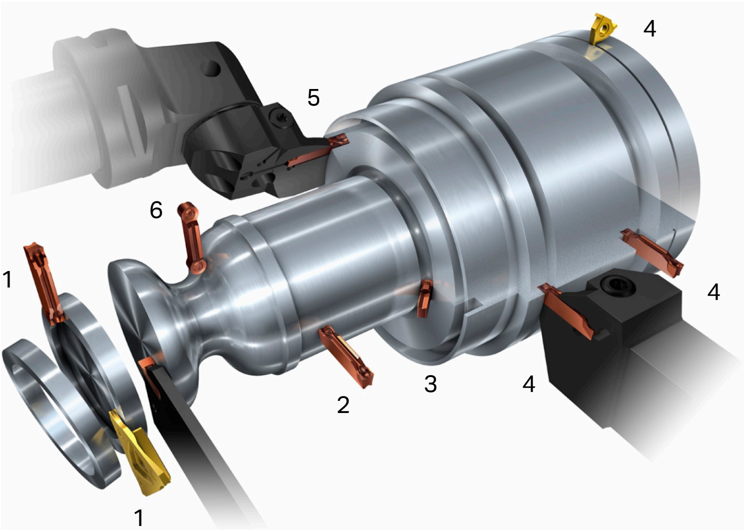
- Use recommended chip former.
- Use neutral front angle.
- Check center height.
- Use cutting fluid.

A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

System overview

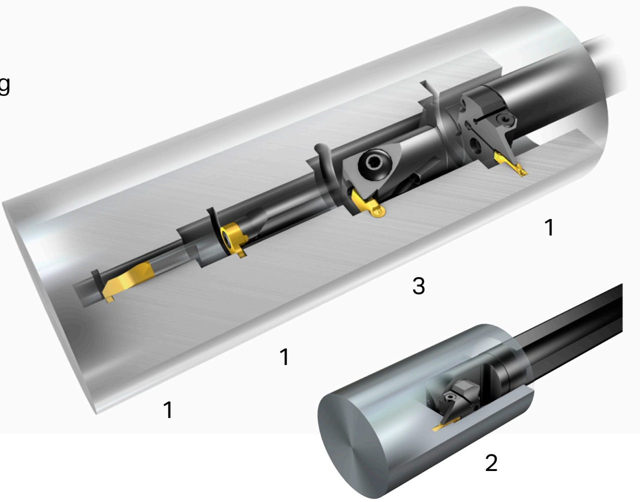
External parting and grooving

1. Parting-off solid bars and tubes
2. Turning and recessing
3. Undercutting
4. Shallow to deep grooving
5. Face grooving
6. Profiling

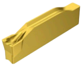
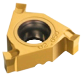
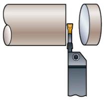
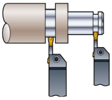
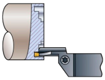
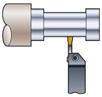
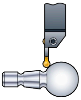
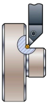
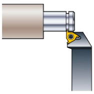


Internal grooving

1. Grooving and pre-parting
2. Face grooving
3. Profiling



Different systems

Insert type / Application		 CoroCut2	 CoroCut1	 CoroCut3	 CoroCut QD	 CoroCut QF	 Circlip 266
Parting (Cut off)		◯ Medium	◯ Deep	◯ Shallow	◯ Deep		
Grooving		◯	○		◯		
Face groov-ing		○	○			◯	
Turning		◯	○				
Profiling		◯	○				
Undercutting		◯			○		
Circlip grooving		○		◯			◯

◯ First choice ○ Second choice

External parting and grooving

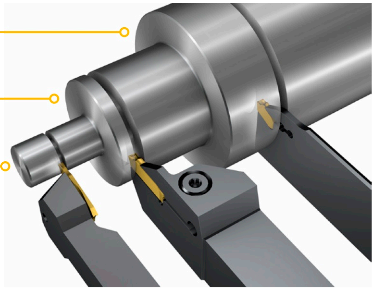
Different systems

External parting – diameter ranges

Deep parting – Ø < 160 mm (6.299”)

Medium parting – Ø < 40 mm (1.575”)

Shallow parting – Ø < 12 mm (.472”)



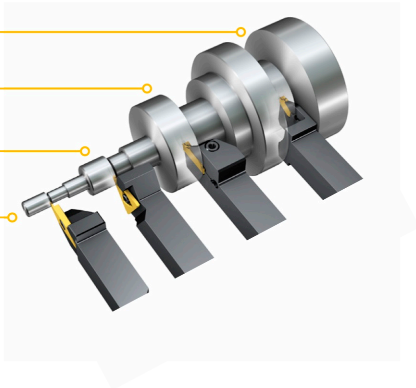
Grooving – depth ranges

Deep grooving – depth < 100 mm (3.937”)

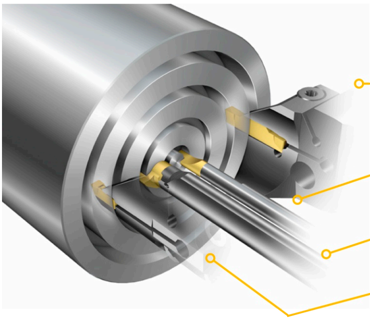
Medium grooving – depth < 50 mm (2.000”)

Shallow grooving – depth < 6 mm (.236”)

Shallow grooving – depth < 3.7 mm (.146”)



Face grooving – diameter ranges



Large diameters > 34 mm (1.338")

Small diameters > 0.2 mm (.0078")

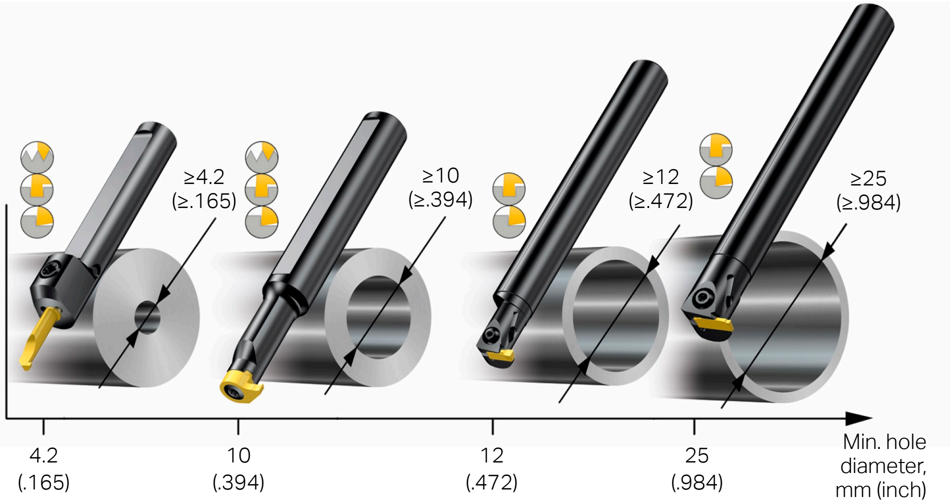
Small diameters > 6 mm (.236")

Medium to large diameters > 16 mm (.629")

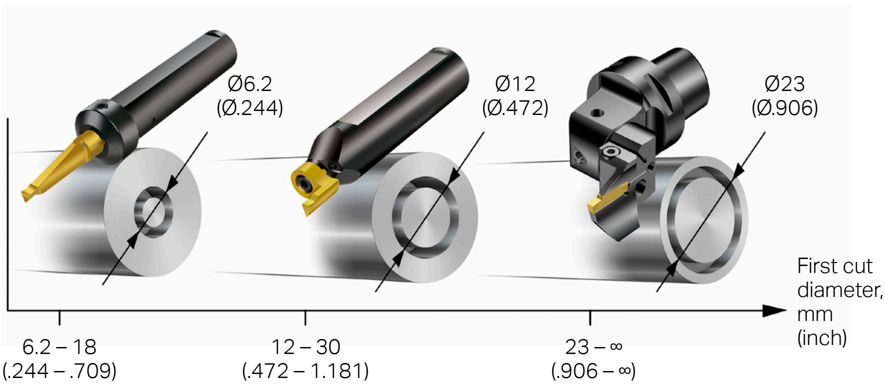
Internal parting and grooving

Different systems

Internal grooving – min hole diameter

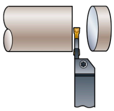
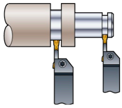
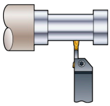
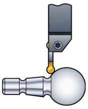
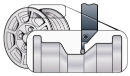
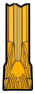
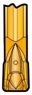
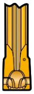
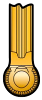
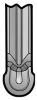
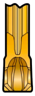
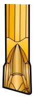
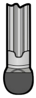
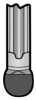


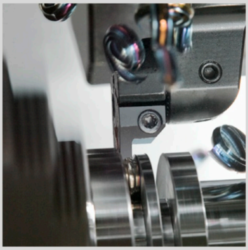
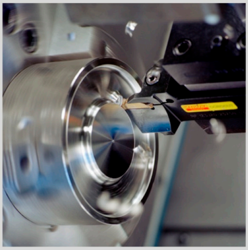
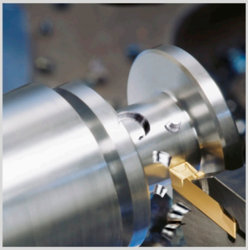
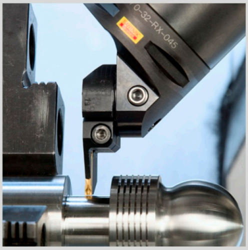
Face grooving – hole diameter range



Inserts

Geometry overview

Application / Machining condition	 Parting (Cut off)	 Grooving	 Turning	  Profiling
Finishing	**CF** 	**GF** 	**TF** 	
Medium	**CM** 	**GM** 	**TM** 	**RM**  **AM** 
Roughing	**CR** 			
Optimizer				**RO** 
	CS 			**RS** 
		GE 		**RE** 

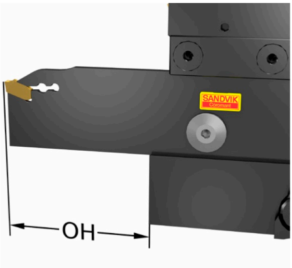






Parting & grooving and how to apply

Tool overhang and workpiece deflection

The tool overhang should always be minimized for improved stability. In parting and grooving operations consideration must be given to the depth of cut and the width of the groove, which means that stability must often be compromised to meet the demands of accessibility.

Best stability

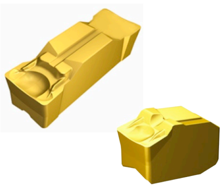


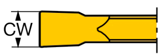
- Overhang (OH) should be as small as possible
- Largest seat size should be used

Internal machining

Shank type:

- Steel bars ≤3 x DMM
- Dampened steel bars ≤5 x DMM
- Carbide bars ≤5 x DMM
- Carbide reinforced dampened bars, up to 7 x DMM.



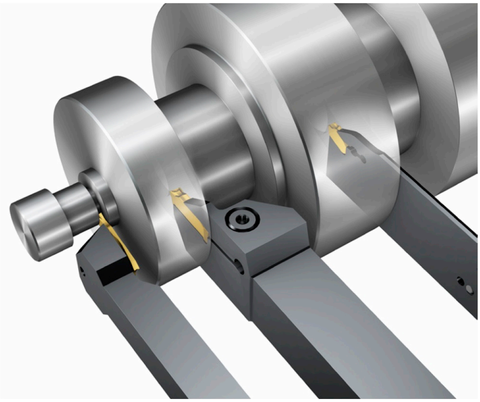


Inserts:

- Use smallest possible width
- Use light cutting geometries.

Tool holder selection parameters

System considerations



Deep parting

Shallow parting

Medium parting

Deep parting

- First choice are spring-clamp blades with single-edge inserts.

Medium parting

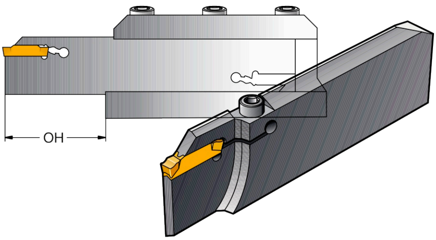
- First choice for medium parting are holders with 2-edge inserts.

Shallow parting

- Use the 3-edge insert for economic parting in mass production.

General tool holder considerations

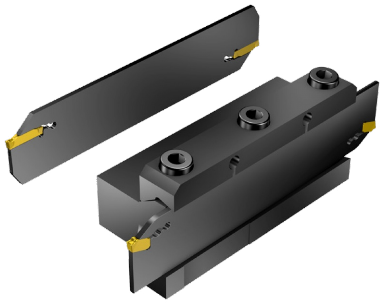
Tool block with spring-clamp tool blade for tool overhang adjustment.



- Shortest possible overhang, OH mm (inch)
- Maximum tool holder shank
- Largest height dimension
- Maximum blade width.

A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

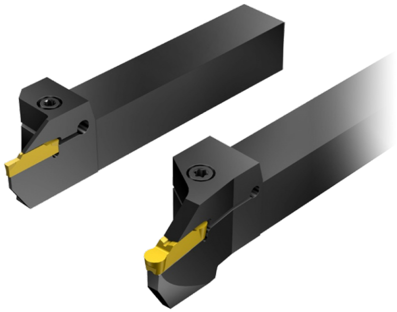
Spring-clamp design blades



Features/Benefits

- Quicker insert change
- Cut off larger diameter
- Adjustability
- Deep grooving
- Double ended
- Radial feed only
- Precision coolant.

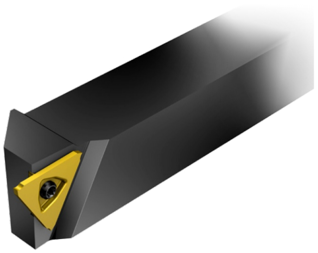
Screw-clamp and spring-lock design holders



Features/Benefits

- Smaller diameters
- Shallow grooving
- Radial & axial feed
- Increased rigidity
- Single ended
- Precision coolant.

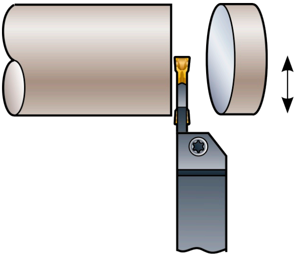
Screw-clamp design holders for 3-edge inserts



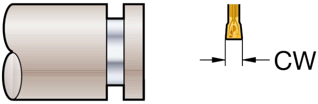
Features/Benefits

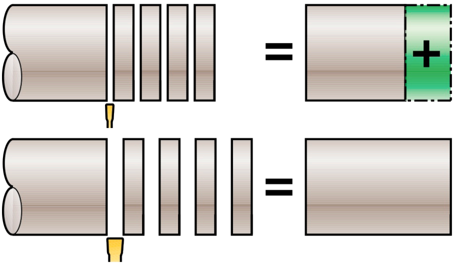
- Extremely small insert widths
  - grooving down to 0.5 mm (.020")
  - parting down to 1 mm (.039").
- Cutting depths up to 6 mm (.236").
- One holder for all insert widths.
- Very tight insert indexing tolerance.
- The productivity choice, 3 cutting edges.

Parting-off bars



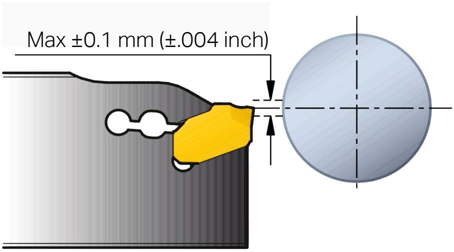
Use as narrow an insert as possible:
- To save material
- Minimize cutting force
- Minimize environmental pollution.





Material savings

Positioning of the tool

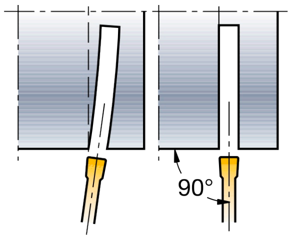


Use maximum deviation of ±0.1 mm (±.004 inch) from center line.

Too high cutting edge
- Clearance will decrease.
- Cutting edge will rub (break).

Too low cutting edge
- Tool will leave material in center (PIP).

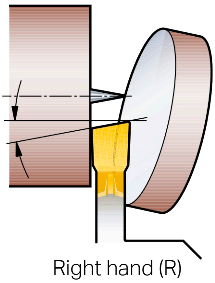
Positioning of the tool

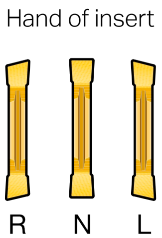


90° mounting of tool holder

- Perpendicular surface
- Reduce vibrations.

Hand of insert



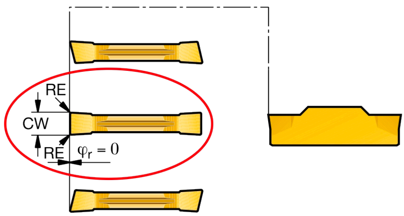


Three types of insert with different entering angles:

- Right hand (R)
- Neutral (N)
- Left hand (L).

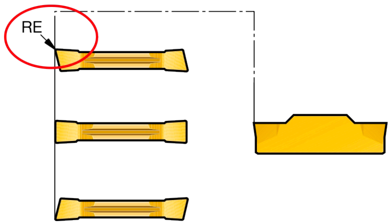
Insert geometry

Neutral entering angle



- Increases strength
- Higher feed/productivity
- Better surface finish
- Straighter cut
- Pip stays on part falling off.

Small/large corner radius



Small corner radius

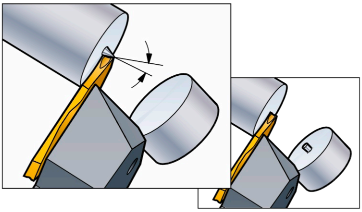
- Smaller pip
- Better chip control
- Lower feed rate.

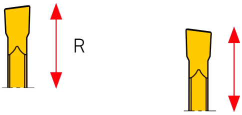
Large corner radius

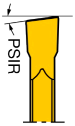
- Increased feed rate
- Longer tool life.

Parting off

Pip reduction by using different front angles





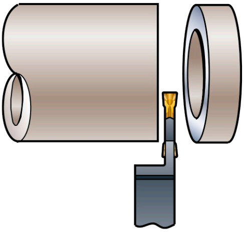


Example of front angles on 1-, 2- and 3-edge inserts:
KAPR = 95°, 98°, 100°, 102°, 105°, 110°
(PSIR = 5°, 8°, 10°, 12°, 15°, 20°)

- Choose left or right hand front angle to control the pip or burr.
- When the front angle is:
  - increased, the pip/burr is decreased
  - decreased, the chip control and tool life are improved.
- Centrifugal force will always push away the parted off component
  - Tool will leave material in center (pip).

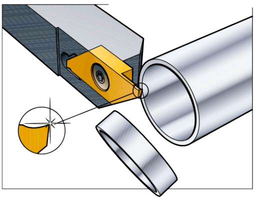
Note!
A front-angled insert will give reduced chip control due to the direction of the chip flow. (A neutral insert directs the chip straight out of the groove).

Parting-off tubes



Use insert with the smallest possible width (CW) to save material, minimize cutting force and environmental impact.

Parting-off thin walled tubes



Make sure that the lowest possible cutting forces are generated. Use inserts with the smallest possible width and sharpest cutting edges.

Tool selection - Review

General recommendations:

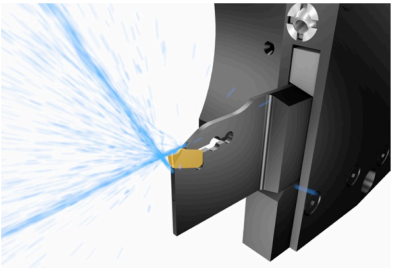
- Neutral inserts
- Smallest possible insert width
- Largest possible tool holder.

Consider:

- Cutting depth
- Insert width
- Front angle
- Corner radius.

Use cutting fluid

Cutting fluid has an important function since the space is often restricted and obstructed by the chips. It is therefore important that precision coolant always us used in large amounts and directed at the cutting edge throughout the whole operation.



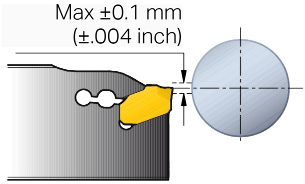
Apply:

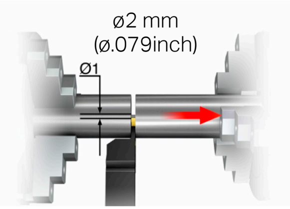
- Use large amounts
- Directly at the cutting edge
- Precision coolant.

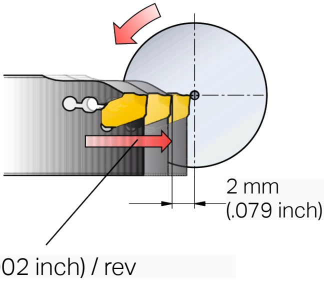
Result:

- Positive effect on chip formation
- Prevents chip jamming
- Adds tool life.

Practical hints



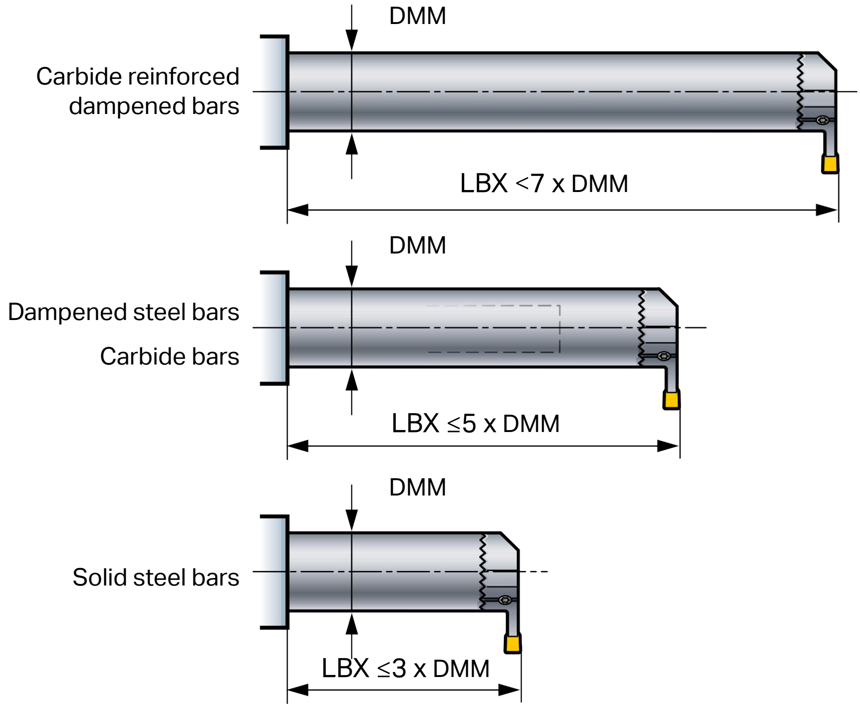




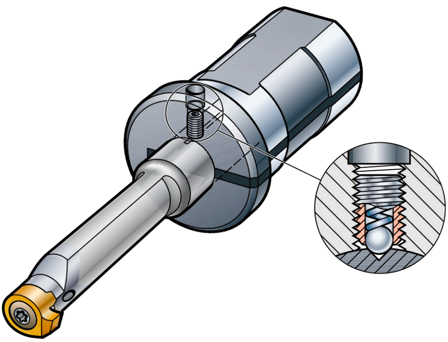
- Center height is important, ±0.1 mm (±.004 inch).
- If subspindle is used, pull away the component approximately 2 mm (.079 inch) before center.
- Recommended feed rate 0.05 (.002 inch) / per rev approximately 2 mm (.079 inch) before center – also for tube parting.

Recommendations for boring bar solutions

Recommended overhang

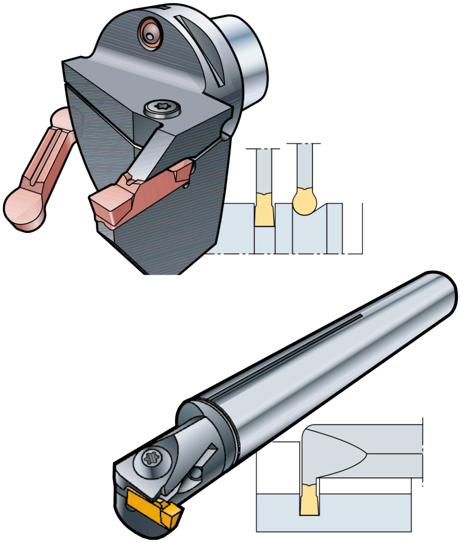


EasyFix sleeves



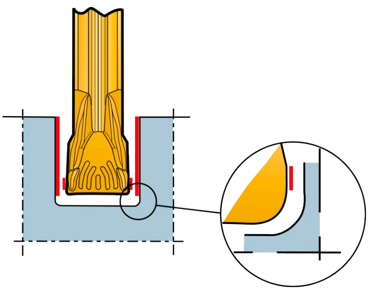
Use EasyFix clamping sleeves for accurate machining with less vibration and precise height.

General grooving

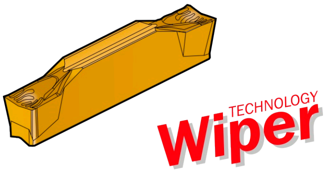


- Single cut grooving is the most economic and productive method to produce grooves.
- If the depth of the groove is bigger than the width, multiple grooving is the best method for rough grooving.
- A screw-clamp or spring-lock design holder should be selected for grooving operations.

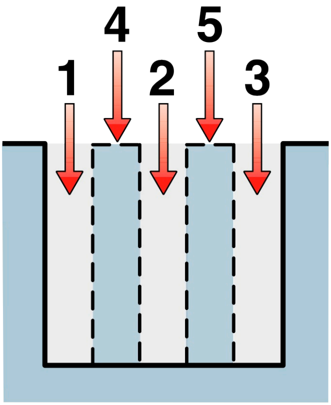
Single cut grooving



- Economic and productive method to produce grooves.
- Finishing geometry has width tolerance of ±0.02 mm (±.0008 inch) and works well in low feeds.
- Wiper inserts give extremely high quality surface on the side of the groove.



Multiple grooving

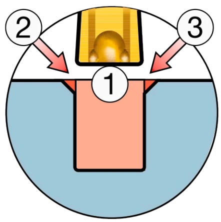


- The best method for rough grooving when depth is bigger than width.
- Use the insert width to produce full grooves and then remove the rings.

Practical hints

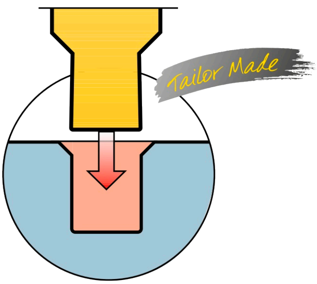
When producing high quality grooves, there is often a need for chamfered corners.

A



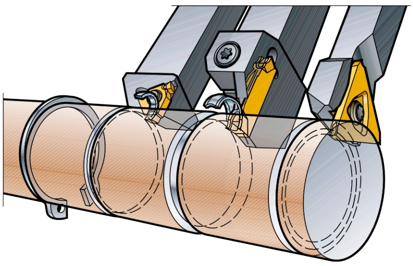
- One way is to use the corners on the insert, for example, of a finishing grooving insert, to chamfer; see illustration A.

B



- A better way to make grooves with chamfer in mass production is to order a Tailor Made insert with the exact chamfer form; see illustration B.

Circlip grooving

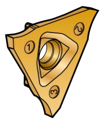


Circlips on shafts and axle components are very common.

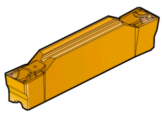
- Circlip grooving can be performed with three-edge inserts or two-edge grooving inserts.
- For internal grooving there is also a wide choice of inserts and boring bars.

Systems to choose from

3-edge inserts



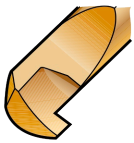
2-edge inserts



- For best economy, use 3-edge inserts in widths 1.00 - 3.18 mm (.039 - .125 inch).
- Or 2-edge inserts in widths 1.50 - 6.00 mm (.059 - .236 inch).

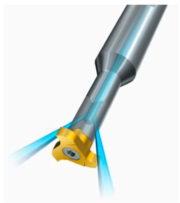
Internal inserts

Carbide rod inserts



- Internal inserts are available for min. hole diameter 10 mm (.394 inch) and with circlip widths 1.10 - 4.15 mm (.043 - .163 inch).
- Min hole diameter for carbide rod inserts is 4.2 mm (.165 inch) and circlip widths are 0.78 - 2.00 mm (.031 - .079 inch).

Internal



Cutter diameter
9.7 – 34.7 mm
(.382 - 1.366 inch)

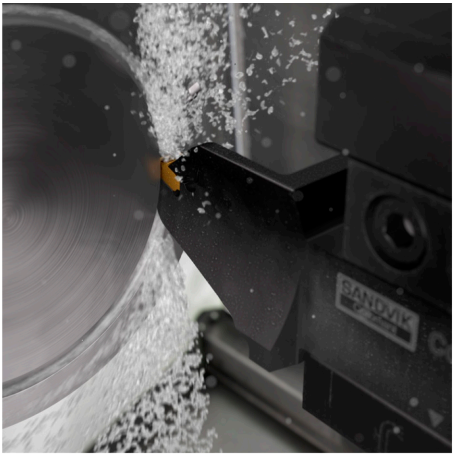
Internal/external

Cutter diameter
39 – 80 mm
(1.535 - 2.480 inch)

Milling is an alternative for non-rotating components

- The circlip widths for diameters 9.7 – 34.7 mm (.382 - 1.366 inch) cutters are 0.70 - 5.15 mm (.028 - .203 inch).
- The circlip widths for diameters 39 – 80 mm (1.535 - 2.480 inch) cutters are 1.10 - 5.15 mm (.043 - .203 inch).

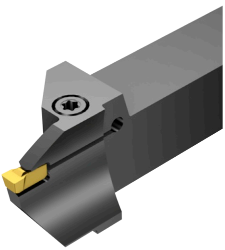
Face grooving



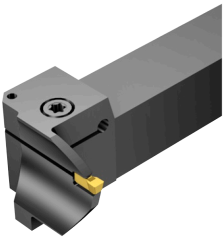
Making grooves axially on the faces on a component requires tools dedicated to the application.

- The correct curve on the tool is dependent on the radius of the workpiece.
- The inner and outer diameters of the groove need to be taken into account in order to select the tool.

Tools for face grooving



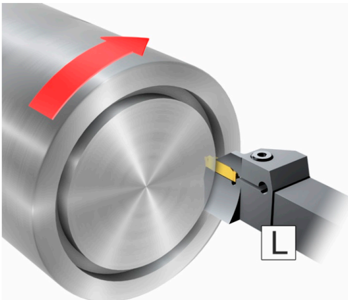
- Curved tool for face grooving, shank 0º style.



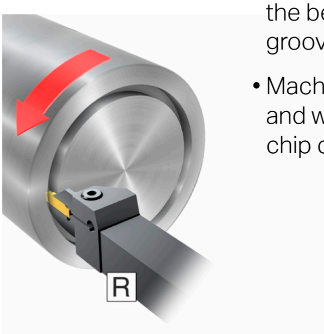
- Curved tool for face grooving, shank 90º style.

- Exchangable cutting blades make it possible to make a special tool from standard tools.

Choice of R and L tools depending on rotation



Left hand (L) tool

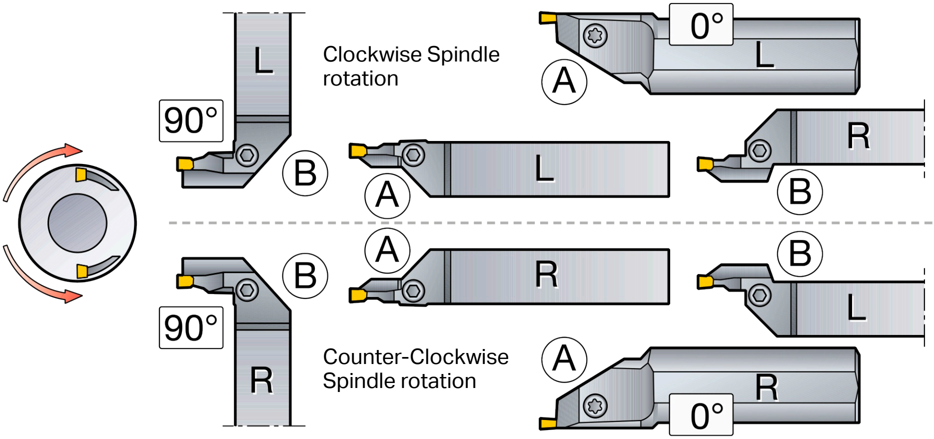


Right hand (R) tool

- Tool is fed axially towards the end surface of the part.
- Tool must be adapted to the bending radius of the groove.
- Machine largest diameter and work inwards for best chip control.

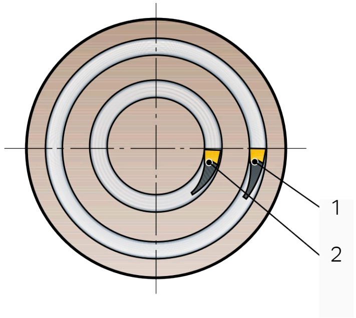
Choice of A and B curve, right or left hand tool

Choose the correct tool – A or B curve, right or left hand style – depending on machine setup and workpiece rotation.



www.tool-builder.com

First cut consederation

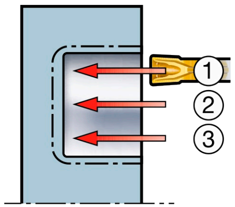


1 **If the insert support rubs workpiece inside dia:**
- maybe the dia. range is wrong
- tool is not parallel to axis
- check center height
- lower the tool below center line.

2 **If the insert support rubs workpiece outside dia:**
- maybe the dia. range is wrong
- tool is not parallel to axis
- check center height
- lift the tool above center line.

Roughing and finishing a face groove

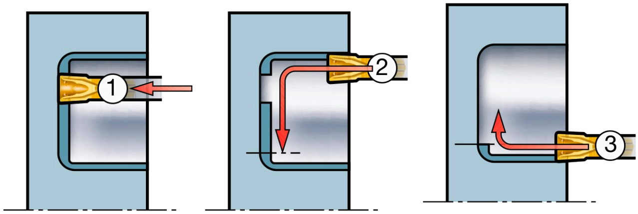
Roughing



First cut (1) always starts on the largest diameter and works inwards. The first cut offers chip control but less chip breaking.

Cuts two (2) and three (3) should be 0.5–0.8 x width of the insert. Chip breaking will now be acceptable and the feed can be increased slightly.

Finishing



Machine the first cut (1) within the given diameter range.

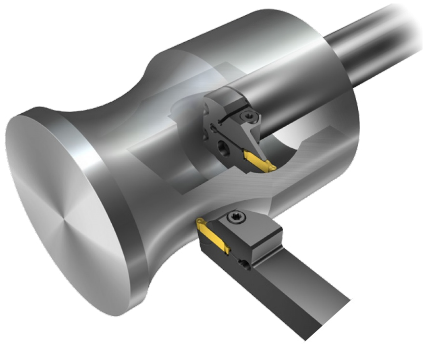
Cut two (2) finishes the diameter. Always start outside and turn inwards.

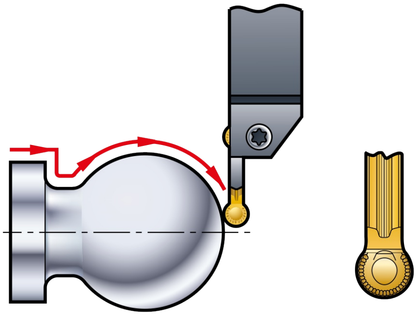
Finally, cut three (3) finishes the inner diameter to the correct dimensions.

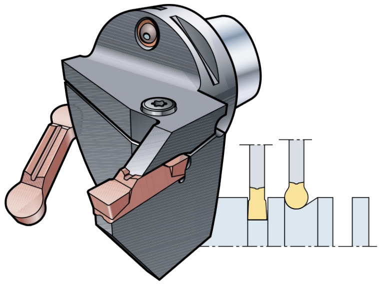
Profiling

When machining components with complex shapes, profiling inserts offer great opportunities for rationalization.

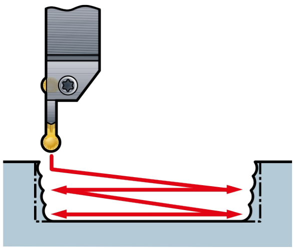
- Modern parting and grooving tool systems can also perform turning.
- A screw-clamp tool holder should be selected for turning and profiling operations in view of achieving maximum stablility.
- A neutral tool holder is suitable for both opening up or completing a recess.
- The round shape inserts have dedicated geometries for these operations.







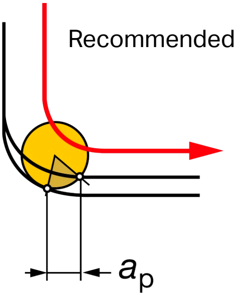
Ramping



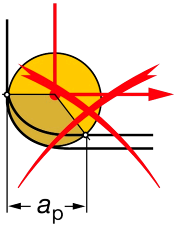
- Use round inserts for outstanding chip control and good surface finish.
- In unstable setups, use ramping to avoid vibrations.

Profile turning

Insert radius < component radius



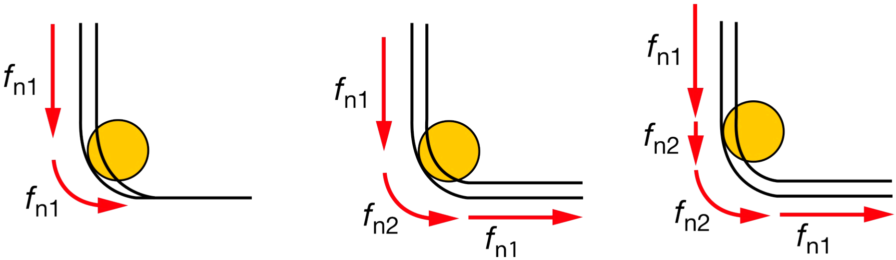
- Large area of insert creates high cutting force so feed should be reduced.
- If possible, use an insert radius that is smaller than the component radius.
- If you must have the same insert radius as the component radius, use micro-stops to make the chip short and avoid vibrations.



Insert radius ≥ component radius is not recommended

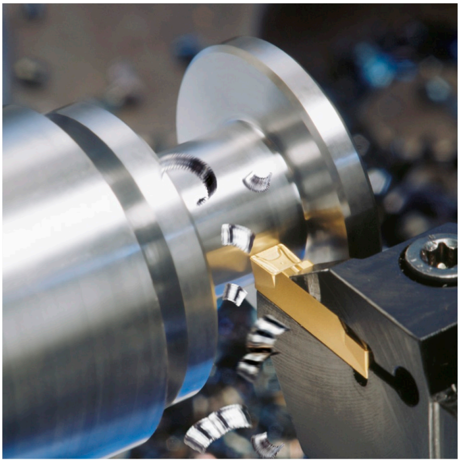
f_{n1} = parallel cuts – max. chip thickness 0.15–0.40 mm (.006 - .016 inch).

f_{n2} = radius plunging – 50% max. chip thickness.



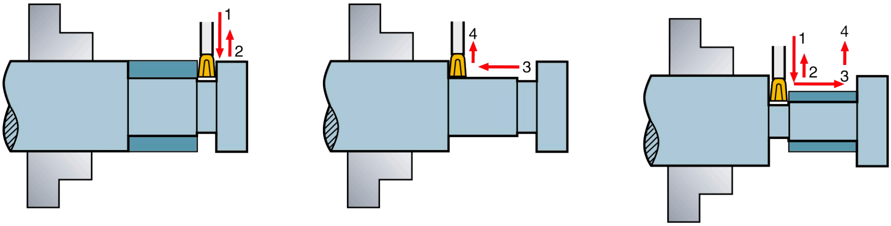
Turning

The most common applications for wide grooves or turning between shoulders are multiple grooving, plunge turning or ramping. All three methods are roughing operations and have to be followed by a separate finishing operation. A rule of thumb is that if the width of the groove is smaller than the depth – multiple grooving should be used and vice versa for plunge turning. However, for slender components, the ramping method may be used.



- Use holders with smallest possible over-hang, screw or spring-lock clamping and insert with rail shape if possible.
- Use a stable, modular tooling system if possible.
- Reinforced blade will increase stability.

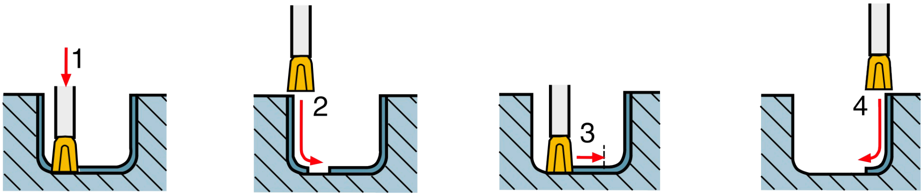
Roughing



1. Radially infeed to required depth +0.2 mm (+.008 inch) (max 0.75 x insert width).
2. Retract radially 0.2 mm (.008 inch).
3. Turn axially to opposite shoulder position.
4. Retract radially 0.5 mm (.020 inch).

Finishing

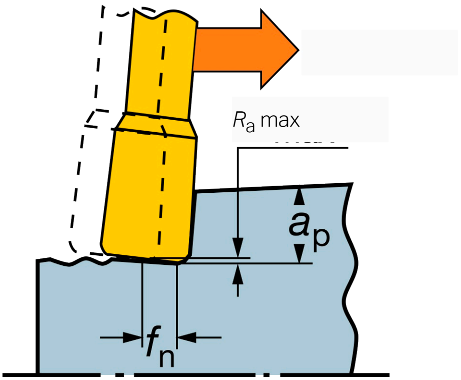
As the insert contours around the radius, most of the movement is in the Z direction. This produces an extremely thin chip along the front cutting edge which can result in rubbing and hence vibration.



- The axial and radial cutting depth should be 0.5–1.0 mm (.020–.039 inch).

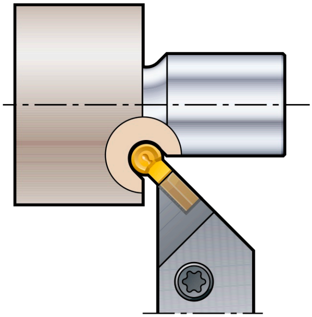
Axial turning

Surface finish



- This wiper effect generates high quality surface finish.
- You get the best wiper effect when you “find” the right combination between feed (f_n) and blade deflection.
- R_a value below 0.5 µm (20 R_a) will be generated with high bearing.

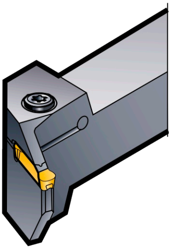
Undercutting



- When a clearance is needed.
- These applications require dedicated inserts with round cutting edges that are sharp and accurate.
- The tolerance of these inserts is low ±0.02 mm (±.0008 inch).

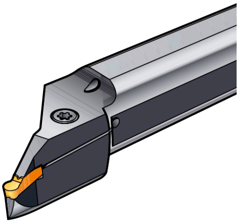
Tools for undercutting

Angled
7° ,45° and 70°



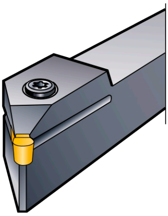
- Holder for external under-cutting. Insert with two cutting edges.

Angled 20°



- Holder for internal under-cutting. Insert with two cutting edges.

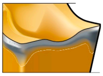
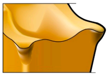
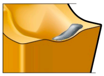
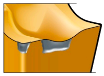
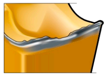
Angled 45°



- Holder for external under-cutting. Insert with one cutting edge.

Troubleshooting

Tool wear

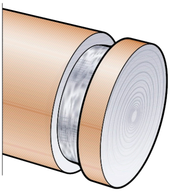
Problem / Solution	 Flank wear	 Plastic deforma-tion	 Crater wear	 Chipping	 Fracture	 Built-up edge
More positive geometry						++
Tougher grade				++		
More wear resistant grade	++	+	+			
Increase cutting speed						+
Decrease cutting speed	+	+	++			
Reduce feed rate		++		+	+	
Choose stronger geometry				+	++	

+ + = Best possible remedy

+ = Possible remedy

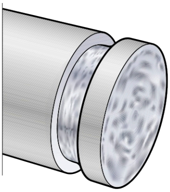
Problem	Solution

Bad surface



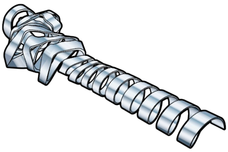
- Use a short and stable tool.
- Take away the chips, use geometry with good chip control.
- Use tools with precision coolant.
- Check speed/feed guidelines.
- Use wiper geometry.
- Check tool setup.

Bad surface on aluminum



- Select the sharpest geometry.
- Use geometry with good chip control.
- Select a special soluble oil for the material.
- Use tools with precision coolant.

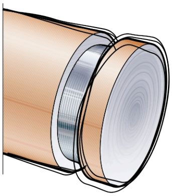
Bad chip breaking



- Change geometry.
- Select a higher feed.
- Use dwelling (pecking).
- Use tools with precision coolant.

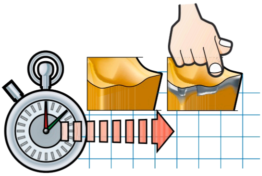
Problem	Solution

Vibration

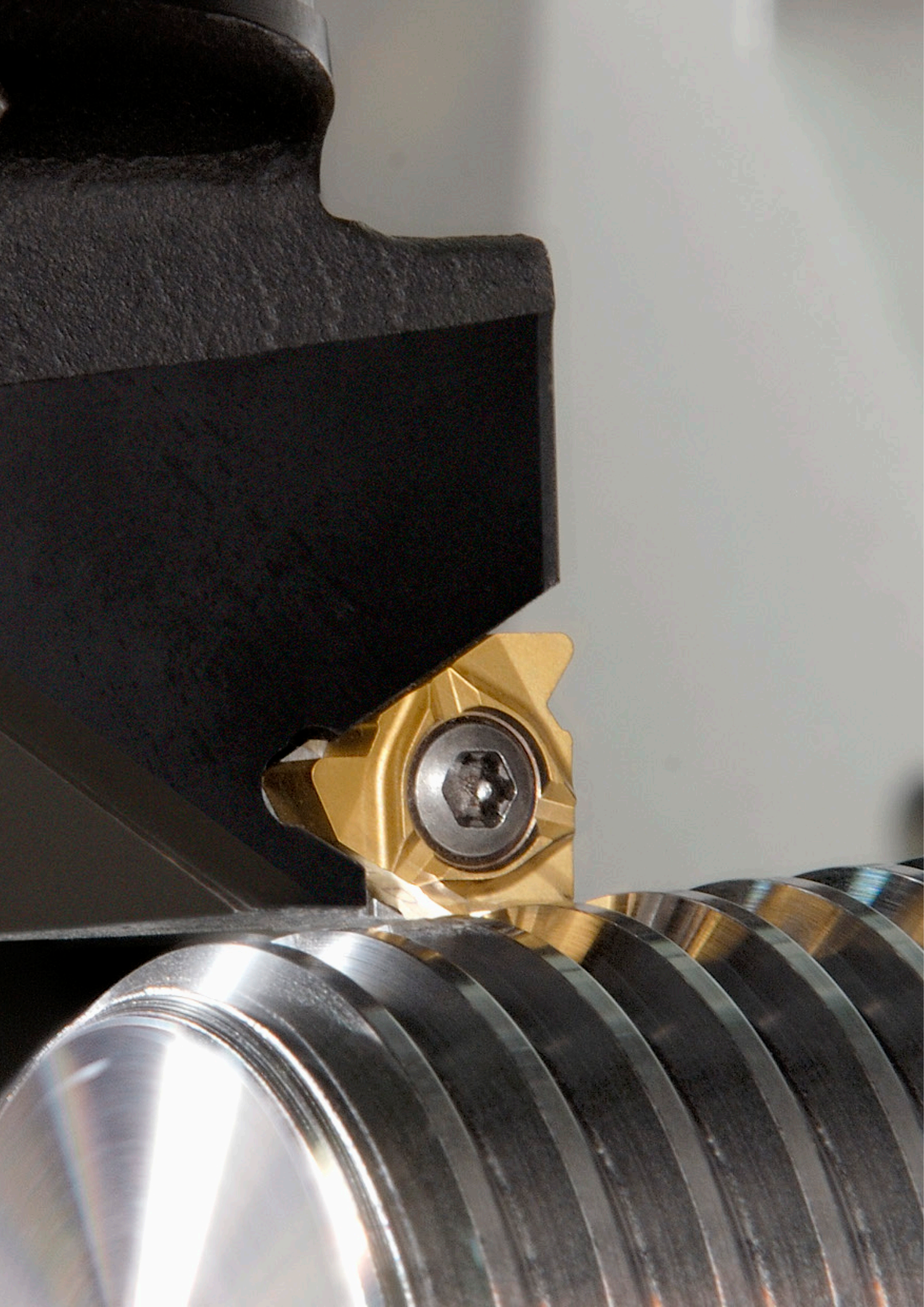


- Use a stable setup.
- Check speed/feed guidelines.
- Use shorter tool and component overhang.
- Change geometry.
- Check tool condition.
- Check tool set-up (center height).

Poor tool life



- Check center height.
- Check angle between tool and component.
- Check condition of blade. If blade is old, the insert could be unstable in the tip seat.
- Use tools with precision coolant.



Threading

Thread turning is the process of an indexable insert tool making a number of passes along the section of a work-piece requiring a screw thread.

By dividing the full cutting depth of the thread into a series of small cuts, the sensitive thread-profile point of the cutting edge is not overloaded.

Threading theory

The threading methods

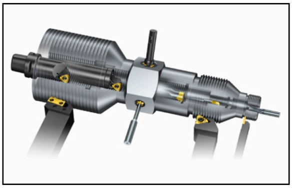
The prime functions of a thread are:

- to form a mechanical coupling
- to transmit motion by converting rotational movement into linear and vice-versa
- to obtain a mechanical advantage; using a small force to create a larger one.

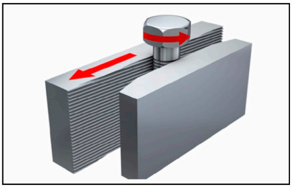
Different ways of making threads

Molding

Metal cutting

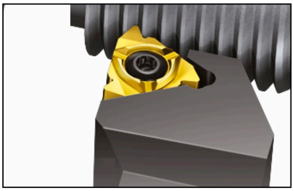


Rolling



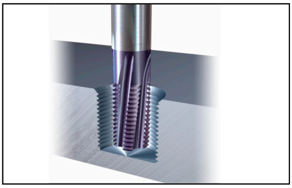
Metal cutting threading methods

Thread turning

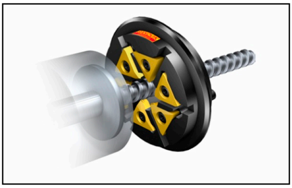


Tapping

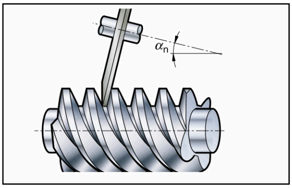
Thread milling



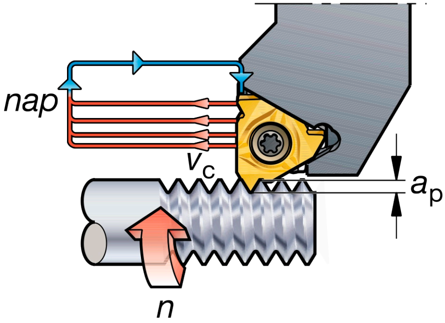
Thread whirling



Grinding



Definitions of terms

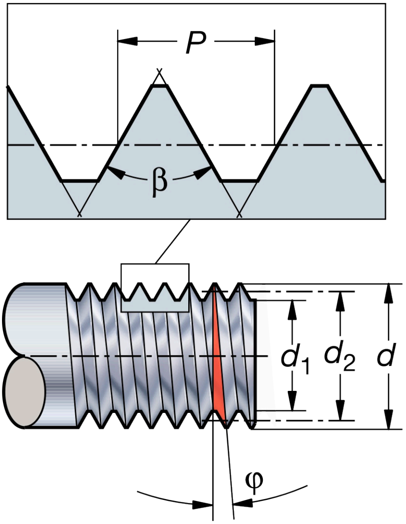


v_c = cutting speed m/min (ft/min)

n = spindle speed (rpm)

a_p = total depth of thread mm (inch)

nap = number of passes



P = pitch, mm or threads per inch (TPI.)

β = angle of the thread

d_1 = minor diameter external

D_1 = minor diameter internal

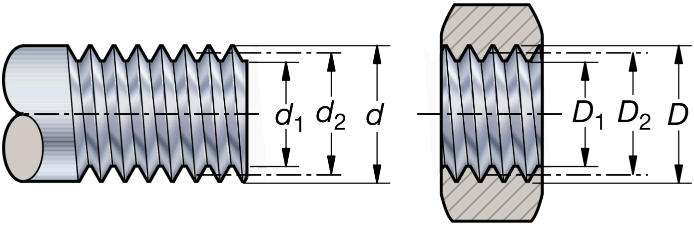
d_2 = pitch diameter external

D_2 = pitch diameter internal

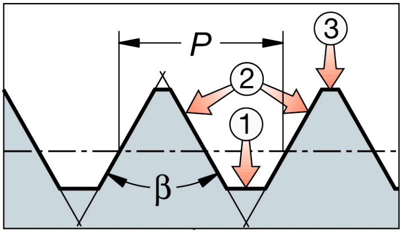
d = major diameter external

D = major diameter internal

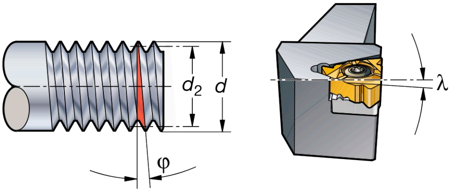
φ = helix angle of the thread



Definitions of terms



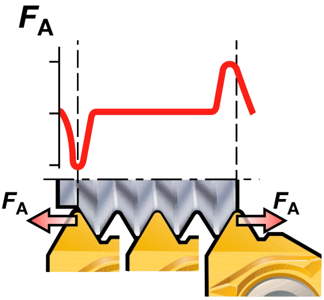
1. Root
   - The bottom surface joining the two adjacent flanks of the thread.
2. Flank
   - The side of a thread surface connecting the crest and the root.
3. Crest
   - The top surface joining the two flanks.



Helix angle

- The helix angle (φ) is dependent on and related to the diameter and pitch (*P*) of the thread.
- By changing the shim, the flank clearance of the insert is adjusted.
- The angle of inclination is lambda (λ). The most common angle of inclination is 1° which is the standard shim in the tool holder.

Cutting forces in and out of the thread

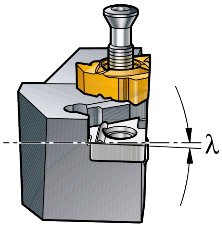


- The highest axial cutting force in the threading operation occurs during the entrance and exit of the cutting tool.
- Aggressive cutting data can lead to movement of insecurely clamped inserts.

Inclining the insert for clearance

Selecting shims for inclination

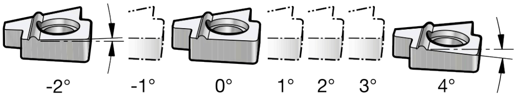
The inclination angle can be set using shims under the insert in the tool holder. The choice of which shim to use can be made by referring to a chart in the catalog.
As standard, all tool holders are delivered with the shim set at 1°.



Tangent of inclination angle

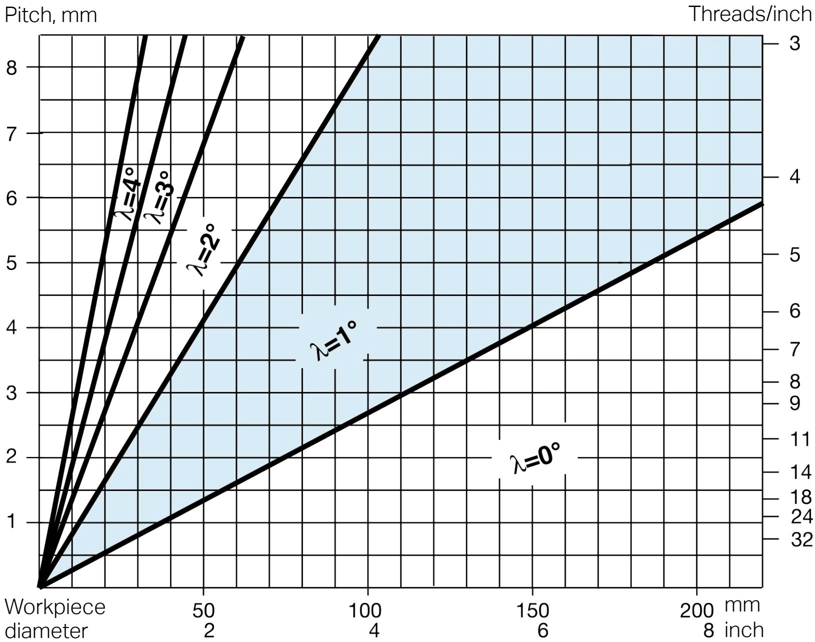
Note! that some pull threading operations require a shim with negative inclination angle.

Standard shim = 1°



$$\tan \lambda = \frac{P\ {}^{*}ns}{\pi \times d_2}$$

* ns = number of starts

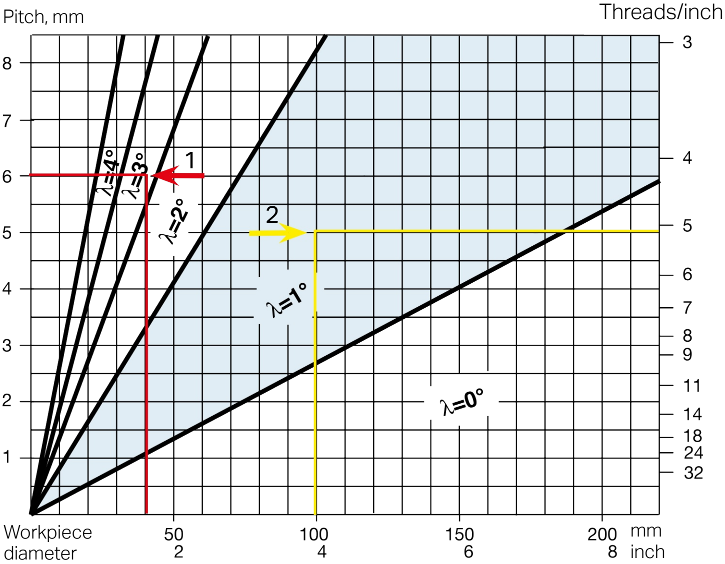


Selecting shims for inclination

The diameter and pitch influence the inclination angles.

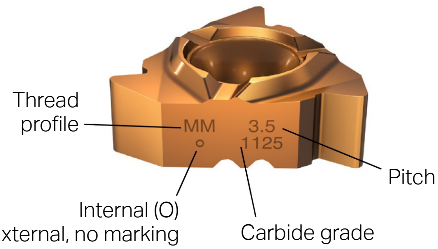
Example of how to use the diagram.

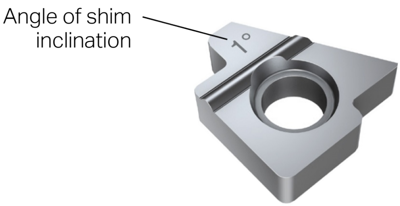
1. The workpiece diameter is 40 mm (1.575") with a thread pitch of 6 mm (.236"). From the diagram we can see that the required shim must have an inclination angle of 3º (standard shim can not be used).
2. The workpiece diameter is 102 mm (4") with a thread of 5 threads per inch. From the diagram we can see that the required shim must have an inclination angle of 1º (standard shim can be used).



Marking of threading inserts and shims

How to read and understand markings.



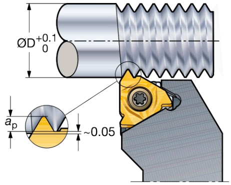


Tool selection procedure

Production planning process

1. **Component**

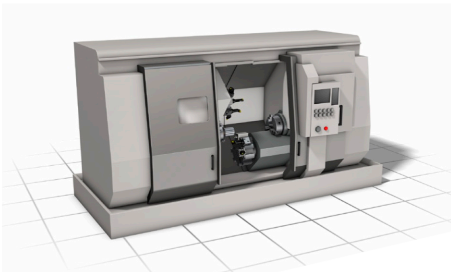
Dimension and quality of the thread



Workpiece material, thread profile and quantity

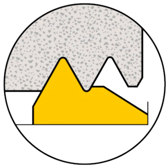
2. **Machine**

Machine parameters



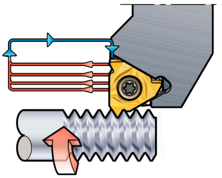
3. **Choice of tool**

Type of tool:
- Multi-point
- Full profile
- V-profile



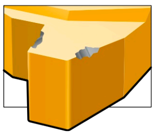
4. **How to apply**

Cutting data, infeed etc.

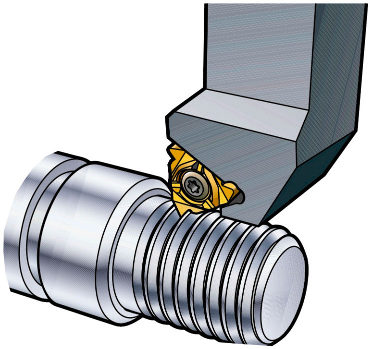


5. **Troubleshooting**

Remedies and solutions



1. Component and the workpiece material



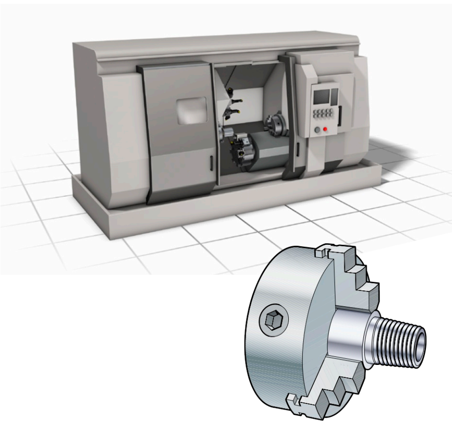
Component

- Analyze the dimensions and quality demands of the thread to be machined
- Type of operation (external or internal)
- Right- or left-hand thread
- Type of profile (metric, UN, etc.)
- Pitch size
- Number of thread starts
- Tolerance (profile, position).

Material

- Machinability
- Chip breaking
- Hardness
- Alloy elements.

2. Machine parameters



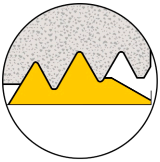
Condition of the machine and setup

- Spindle interface
- Machine stability
- Available spindle speed
- Coolant supply
- Clamping of the workpiece
- Power and torque
- Available programming cycles
- Tool reach and clearance
- Tool overhang.

3. Choice of tools

Different ways to make threads

Multi-point inserts



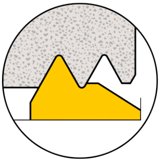
A full profile (topping) insert with several teeth reduces the number of required in-feeds and generates high productivity, e.g. a multi-point insert with two teeth reduces the number of in-feeds to half.

The tool pressure increases proportionally with the number of teeth, requiring stable setups and shortened overhangs. Sufficient room behind the thread is also needed.

Advantages
- Reduced number of infeeds
- Very high productivity.

Disadvantages
- Requires stable setups
- Needs sufficient room behind the thread.

Full profile inserts



The thread is cut by the insert with good control over the geometrical properties as the distance between the root and the crest is controlled.

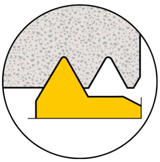
The insert can only cut one pitch.

As the insert is generating both the root and the crest, the tool pressure increases, putting high requirements on setup and overhang.

Advantages
- Better control over the thread form
- Less deburring.

Disadvantages
- Each insert can only cut one pitch.

V-profile inserts



The insert can accommodate a range of pitches thus reducing stock. The root and flanks are being formed by the insert.

The crest is controlled in a prior turning operation, resulting in high tolerances.

In setups prone to vibrations, a non-topping insert can often prove to be a solution due to the reduction of cutting pressure.

Advantages
- Flexibility, the same insert can be used for several pitches.

Disadvantages
- Can result in burr formation that needs to be taken away.

A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

4. How to apply

Important application considerations

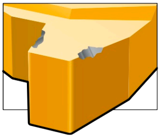
The infeed method can have a significant impact on the thread machining process.

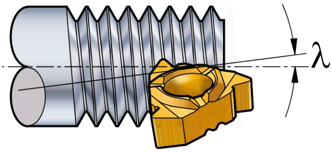
It influences:
- chip control
- insert wear
- thread quality
- tool life.

In practice, the machine tool, insert geometry, work-piece material and thread pitch influence the choice of infeed method.

5. Troubleshooting

Some areas to consider





If you run into trouble with insert tool life, chip control or poor thread quality. Please consider the following aspects.

Infeed type
- Optimize infeed method, number and depth of passes.

Insert inclination
- Ensure there is sufficient and even clearance (insert – inclination shims).

Insert geometry
- Make sure the right insert geometry is used (A, F, or C geometries).

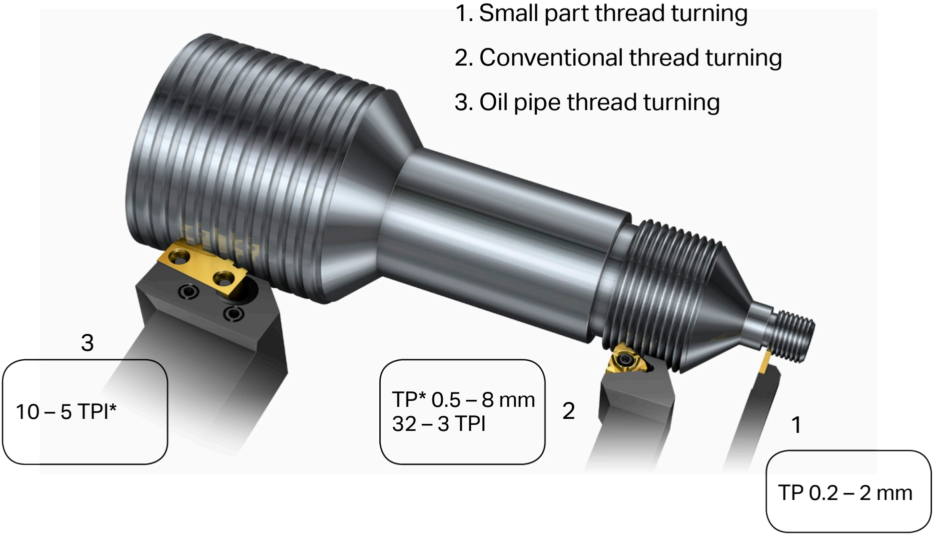
Insert grade
- Select the correct grade based on the material and toughness demands.

Cutting data
- If necessary change cutting speed and number of passes.

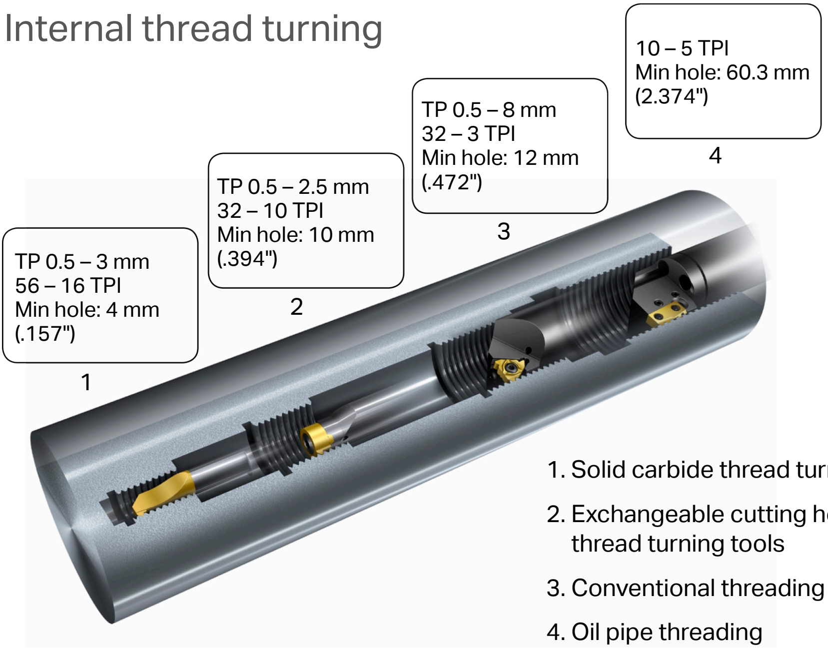
System overview

External thread turning

1. Small part thread turning
2. Conventional thread turning
3. Oil pipe thread turning



Internal thread turning



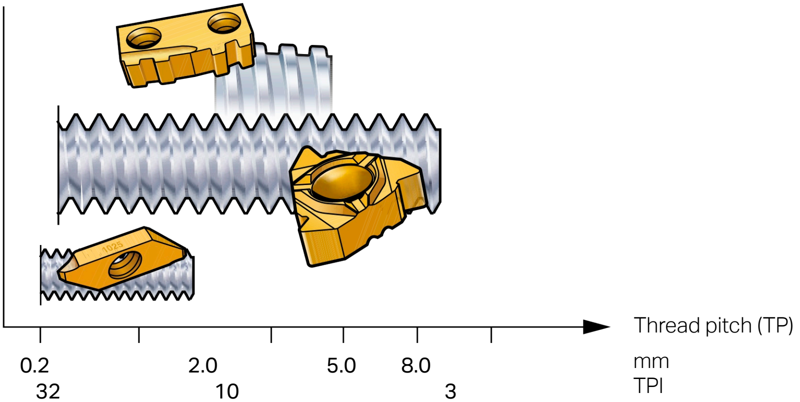
1. Solid carbide thread turning tools
2. Exchangeable cutting head thread turning tools
3. Conventional threading
4. Oil pipe threading

*TPI = Threads per inch
*TP = Thread pitch

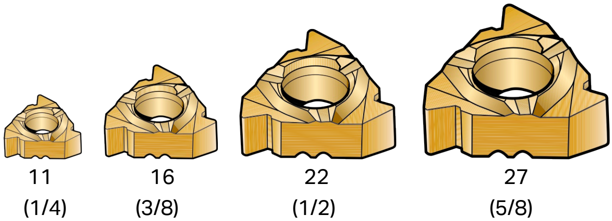
External thread turning assortment

Choose from an extensive program

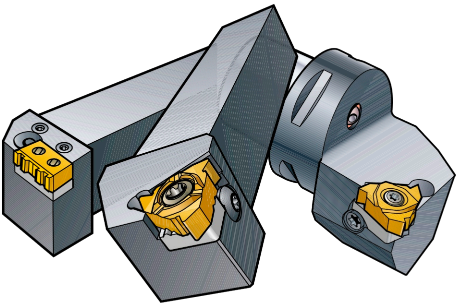
Inserts



- Four insert dimension (L) / sizes (IC):
  11, 16, 22, 27 mm
  (1/4, 3/8, 1/2, 5/8 inch)



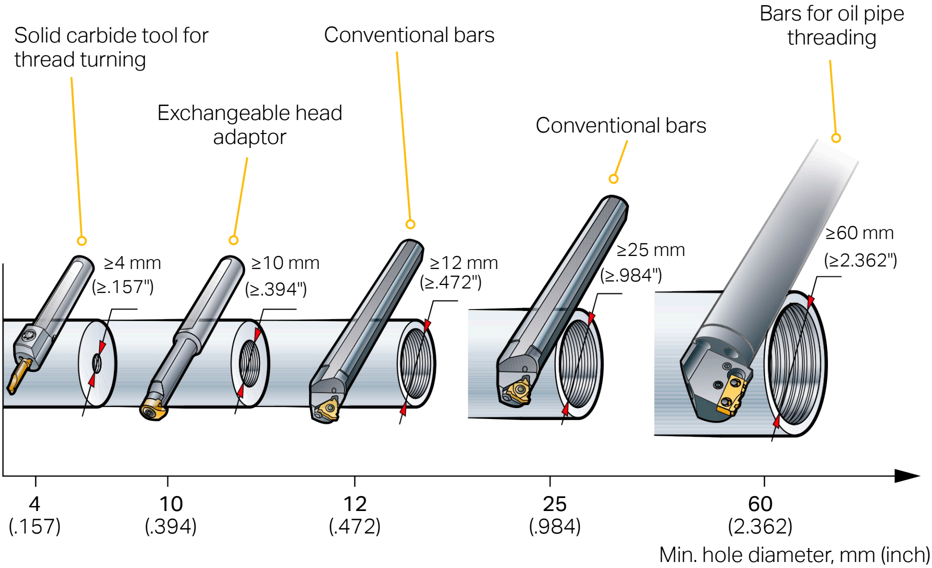
Tool holders



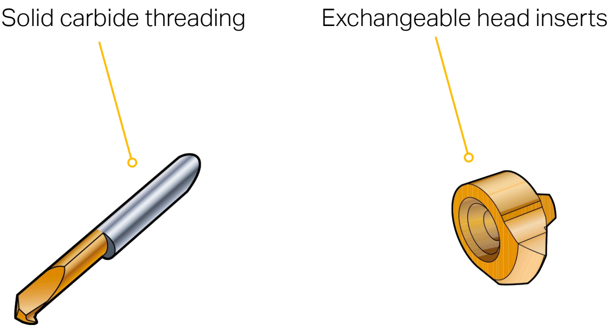
- Coromant Capto® cutting units
- QS-holders
- Shank tools
- Exchangable cutting heads
- Cartridges.

Internal thread turning assortment

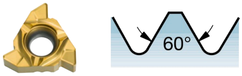
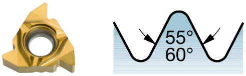
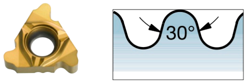
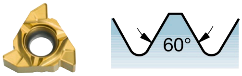
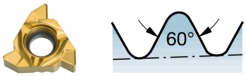
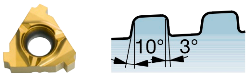
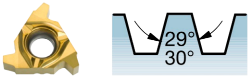
Choose from an extensive program and several systems



For high precision, internal thread turning of small holes



Thread forms

Application	Insert/thread form	Thread type	Code
General use		ISO metric American UN	MM UN
Pipe thread		Whitworth, NPT British Standard (BSPT), NPTF American National Pipe Threads	WH, NT PT, NF
Food and fire		Round DIN405	RN
Aerospace		MJ UNJ	MJ NJ
Oil and gas		API Round API "V" form 60°	RD V38, 40, 50
Oil and gas		Buttress, VAM	BU
Motion General use		Trapezoidal ACME Stub ACME	TR AC SA

General usage
- Good balance between load bearing capacity and volume of material.

Pipe Threads
- Ability to bear loads.
- Able to form leak-proof connections (threads are often conical).

Food & Fire
- Same as for pipe threads but round, for easy cleaning for food.
- Easily repeated connecting/disconnecting for fire.

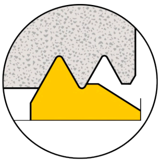
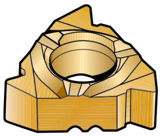
Aerospace
- High precision and minimized risk for stress concentration and breakage.

Oil & Gas
- Extreme load bearing and leak proof requirements, with limitations of thin wall thickness of pipe.

Motion
- Symmetrical form.
- Large contact surface.
- Sturdy form.

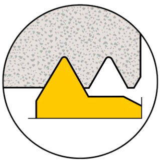
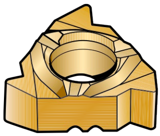
Insert types

Three different types of thread turning inserts

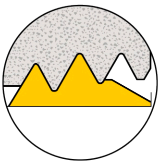
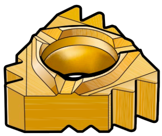
Full profile inserts

• For high productivity in threading.

V-profile inserts - 60° and 55°

• For threading with minimum tool inventory.

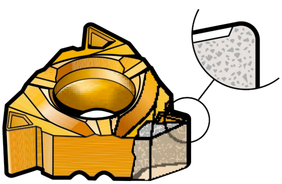
Multi-point inserts

• For highly productive, economic thread turning in mass production.

Three different geometries

A-geometry

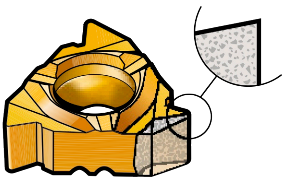
First choice in most operations.



Good chip forming in a wide range of materials.

F-geometry

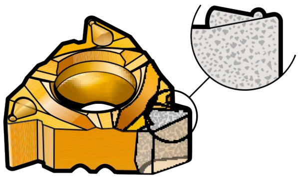
Sharp geometry.



Gives clean cuts in sticky and work hardening materials.

C-geometry

Chip breaking geometry.

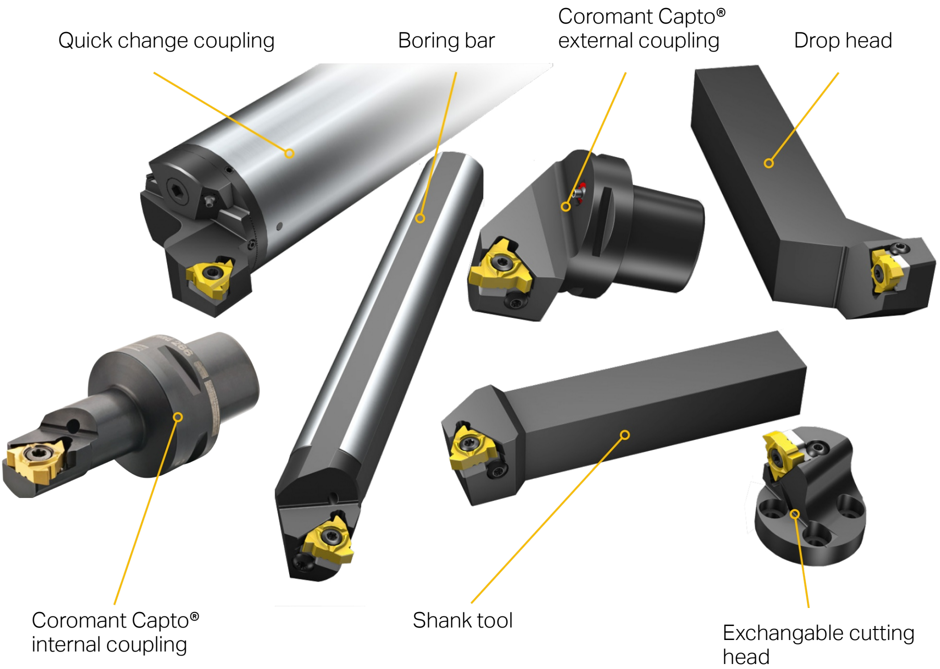


Optimized geometry for low carbon, low alloy and easily machined stainless steel.

Threading solutions

- Ultra-rigid threading with fixed position inserts.
- The insert locates in the correct position with guidance of the rail.
- The screw forces the insert on the rail back to a radial stop at one contact face in the insert seat. (The red contact faces).
- A secure insert interface ensures better tool life and thread quality.

A variety of tool holder solutions



How to apply

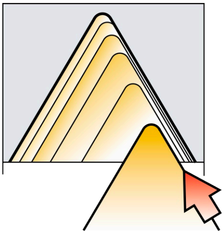
Three different types of infeed

The infeed method can have a significant impact on the thread machining process. It influences:

- chip control
- insert wear
- thread quality
- tool life.

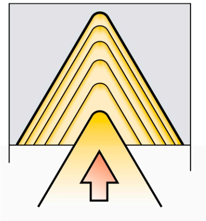
In practice, the machine tool, insert geometry, workpiece material and thread pitch influence the choice of infeed method.

Modified flank infeed



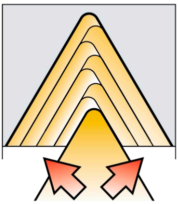
- Most newer CNC machines can be programmed for modified flank.
- Used with C-geometry as the chip breaker will not function with radial infeed.
- Axially directed cutting forces reduce the risk of vibrations.
- Controlled chip direction.
- Used for all insert geometries.
- C-geometry, designed only for modified flank infeed.

Radial infeed



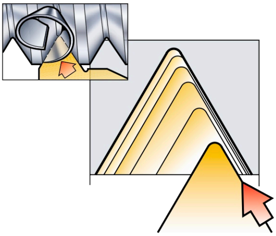
- Used by all manual machines and most canned CNC programs.
- First choice for work hardening materials and suitable for fine pitches.

Incremental infeed



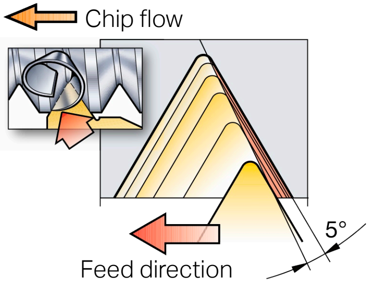
- Normally used with very large profiles and pitches, long work threading cycles where tool life needs to match the length of the thread.
- Requires special programming.

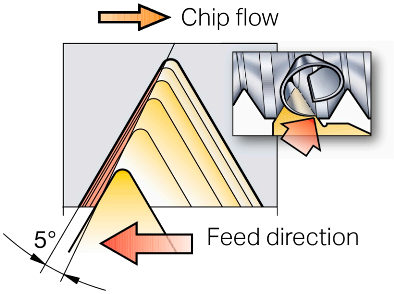
Modified flank infeed



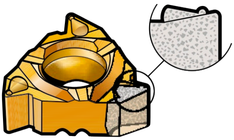
- Most CNC machines have a programmed cycle using this infeed.
- Chip is similar to that in conventional turning - easier to form and guide.
- Axially directed cutting forces reduce the risk of vibrations.
- Chip is thicker, but has contact with only one side of the insert.
- Less heat is transferred to the insert.
- First choice for most threading operations.

Infeed direction



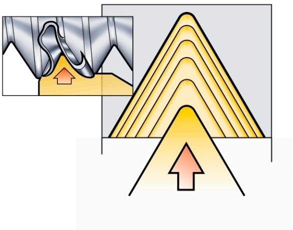


C-geometry insert



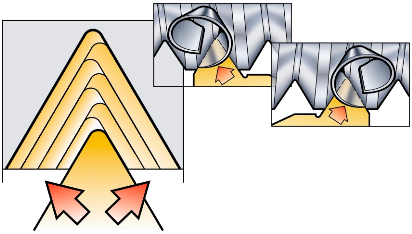
- Better chip control
- Better surfaces
- For C-geometry insert, modified flank infeed is the only suitable infeed.

Radial infeed



- Most commonly used method - and only method possible on older non-CNC lathes.
- Makes a stiff “V” chip.
- Even insert wear.
- Insert tip exposed to high temperatures, which restricts depth of infeed.
- Suitable for fine pitches.
- Vibration possible and poor chip control in coarse pitches.
- First choice for work hardening materials.

Incremental infeed

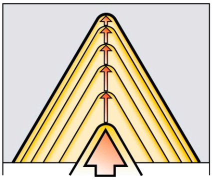


- Recommended for large profiles.
- Even insert wear and longest tool life in very coarse threads.
- Chips are directed both ways, making control difficult.

Programming methods

Ways of improving the machining result

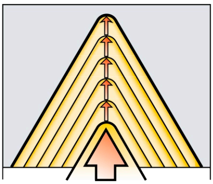
Decreasing depth per pass (Constant chip area)



Allows for constant chip area. This is the most common method in CNC programs.

- The deepest pass is the first pass
- Follows recommendation on infeed tables in catalog
- More “balanced” chip area
- Last pass actually around 0.07 mm (.0028").

Constant depth per pass



Each pass is of an equal depth, regardless of the number of passes.

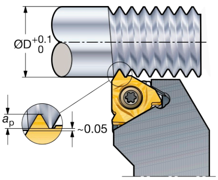
- Much more demanding on the insert
- Offers best chip control
- Should not be used for pitches larger than TP 1.5 mm or 16 TPI.

A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

Thread turning with full profile inserts

Use extra stock/material for topping the thread

For topping inserts, 0.03 – 0.07 mm (.001 – .003") material should be left from prior turning operations to allow for proper forming of the crest.



- The blank does not need to be turned to the exact diameter prior to the threading.
- Add extra stock/material on the workpiece diameter, 0.06 – 0.14 mm (.002 – .006") for topping the finish diameter of the thread.

Infeed values recommendations

Number of infeeds and total depth of thread.

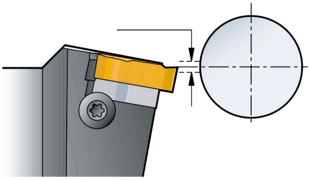
ISO metric and inch, external

No. of infeeds (nap)	Pitch, mm ← Reduce cutting speed														
	0.5	0.75	1.0	1.25	1.5	1.75	2.0	2.5	3.0	3.5	4.0	4.5	5.0	5.5	6.0
	Radial infeed per pass, mm														
1	0.11	0.17	0.19	0.20	0.22	0.22	0.25	0.27	0.28	0.34	0.34	0.37	0.41	0.43	0.46
2	0.09	0.15	0.16	0.17	0.21	0.21	o.24	0.24	0.26	0.31	0.32	0.34	0.39	0.40	0.43
3	0.07	0.11	0.13	0.14	0.17	0.17	0.18	0.20	0.21	0.25	0.25	0.28	0.32	0.32	0.35
4	0.07	0.07	0.11	0.11	0.14	0.14	0.16	0.17	0.18	0.21	0.22	0.24	0.27	0.27	0.30
5	0.34	0.50	0.08	0.10	0.12	0.12	0.14	0.15	0.16	0.18	0.19	0.22	0.24	0.24	0.27
6			0.67	0.08	0.08	0.10	0.12	0.13	0.14	0.17	0.17	0.20	0.22	0.22	0.24
7				0.80	0.94	0.10	0.11	0.12	0.13	0.15	0.16	0.18	0.20	0.20	0.22
8						0.08	0.08	0.11	0.12	0.14	0.15	0.17	0.19	0.19	0.21
9						1.14	1.28	0.11	0.12	0.14	0.14	0.16	0.18	0.18	0.20
10								0.08	0.11	0.12	0.13	0.15	0.17	0.17	0.19
11								1.58	0.10	0.11	0.12	0.14	0.16	0.16	0.18
12									0.08	0.08	0.12	0.13	0.15	0.15	0.16
13									1.89	2.20	0.11	0.12	0.12	0.13	0.15
14											0.08	0.10	0.10	0.13	0.14
14											2.50	2.80	3.12	0.12	0.12
16														0.10	0.10
														3.41	3.72

No. of infeeds (nap)	Pitch, TPI																	
	32	28	24	20	18	16	14	13	12	11	10	9	8	7	6	5	4.5	4
	Radial infeed per pass,																	
1	.007	.006	.007	.007	.008	.007	.007	.008	.009	.008	.008	.008	.009	.010	.009	.012	.011	.013
2	.006	.005	.006	.007	.007	.007	.007	.007	.008	.008	.008	.008	.008	.009	.009	.011	.011	.012
3	.005	.005	.006	.006	.007	.007	.007	.007	.008	.008	.008	.008	.008	.009	.009	.011	.011	.012
4	.003	.004	.005	.006	.006	.006	.006	.007	.007	.007	.007	.007	.008	.009	.009	.011	.010	.012
5		.003	.003	.005	.005	.006	.006	.006	.007	.007	.008	.007	.007	.008	.008	.010	.010	.011
6				.003	.003	.005	.005	.006	.006	.006	.006	.007	.007	.008	.008	.010	.010	.011
7						.003	.005	.005	.005	.006	.006	.006	.007	.008	.008	.010	.010	.011
8							.003	.003	.003	.005	.006	.006	.006	.007	.008	.009	.009	.010
9										.003	.005	.005	.006	.007	.007	.009	.009	.010
10											.003	.005	.005	.006	.007	.008	.008	.010
11												.003	.005	.005	.007	.008	.008	.009
12													.003	.003	.006	.007	.008	.008
13															.005	.006	.007	.008
14															.004	.004	.007	.007
14																	.006	.006
16																	.004	.004
																	3.72	3.72

Positioning of the tool

Max ± 0.1 mm (±.004 inch)



Use maximum deviation of ±0.1 mm (±.004") from centerline.

Too high cutting edge

- Clearance will decrease.
- Cutting edge will rub (break).

Too low cutting edge

- The thread profile can be incorrect.

Method of thread turning

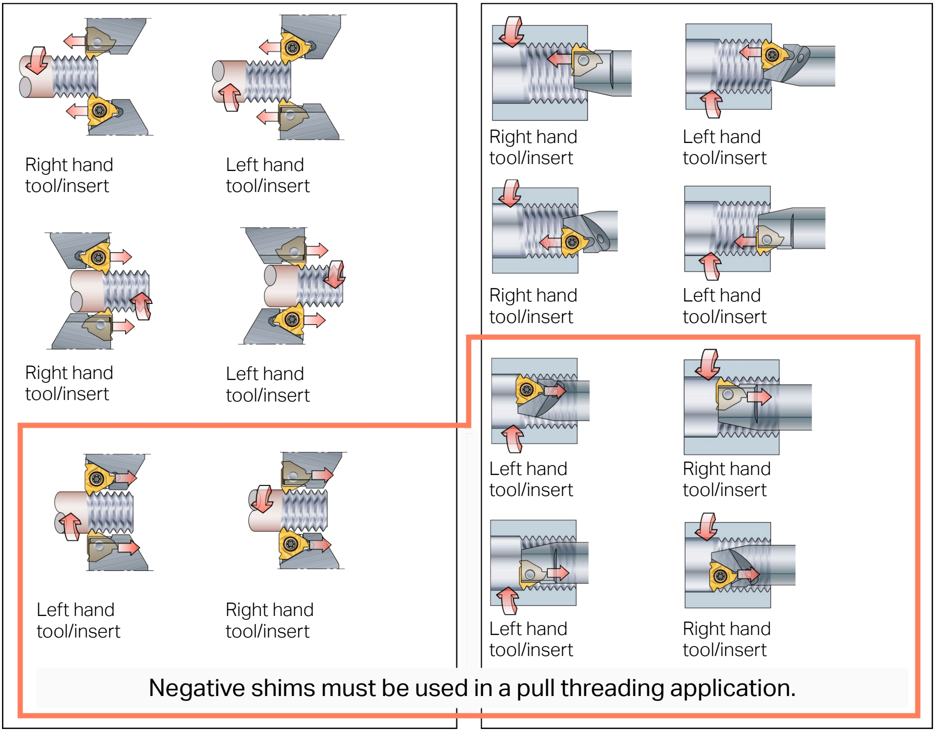
Right and left hand threads and inserts

External

Right hand threads | Left hand threads

Internal

Right hand threads | Left hand threads



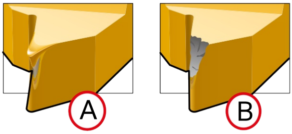
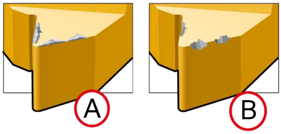
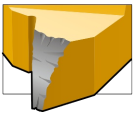
Negative shims must be used in a pull threading application.

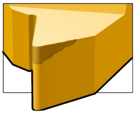
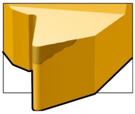
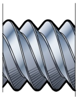
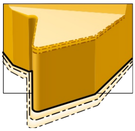
Thread turning application hints

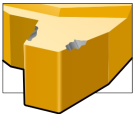
Some vital factors to consider to achieve success

- Check the workpiece diameter for correct working allowance before thread-turning, add 0.14 mm (.006") as crest allowance.
- Position the tool accurately in the machine.
- Check the setting of the cutting edge in relation to pitch diameter.
- Make sure the correct insert geometry is used (A, F, or C).
- Ensure there is sufficient and even clearance (insert-inclination shims) to achieve correct flank clearance by selecting the appropriate shim.
- If threads are rejected, check entire setup, including machine tool.
- Check the available CNC program for thread turning.
- Optimize infeed method, number and size of passes.
- Ensure the correct cutting speed for the demands of the application.
- In case of pitch error on component thread, check to see if machine pitch is correct.
- It is recommended that the tool should start a minimum distance of 3 times the thread pitch before engaging the workpiece.
- Precision coolant can improve tool life and chip control.
- A quick change system allows for quick and easy setup.
- For best productivity and tool life, first choice - multi-point insert, second choice - full profile single point insert, third choice - V-profile insert.

Troubleshooting

Problem	Cause	Solution
Plastic deformation		
 (A) Starts as plastic deformation, (B) which leads to edge chipping.	1. Excessive temperature in cutting zone. 2. Inadequate supply of coolant. 3. Wrong grade.	1. Reduce the cutting speed, increase the number of infeeds. Reduce the largest infeed depth, check the diameter before threading. 2. Improve coolant supply. 3. Choose a grade with better resistance to plastic deformation.
Built-up edge (BUE)		
 BUE (A) and edge chipping (B) often occur in combination. Accumulated BUE is then ripped away together with small amounts of insert material, which leads to chipping.	1. Often occurs in stainless steel and low carbon steel materials. 2. Unsuitable grade or cutting edge temperature too low.	1. Increase cutting speed. 2. Choose an insert with good toughness, preferably PVD coated.
Insert breakage		
	1. Wrong turned diameter prior to threading. 2. Infeed series too tough. 3. Wrong grade. 4. Poor chip control. 5. Center height incorrect.	1. Turn to correct diameter before threading operation, 0.03 – 0.07 mm (.001 – .003") radially larger than max. diameter for thread. 2. Increase number of infeeds. Reduce size of the largest infeeds. 3. Choose a tougher grade. 4. Change to C-geometry and use modified flank infeed. 5. Correct center height.

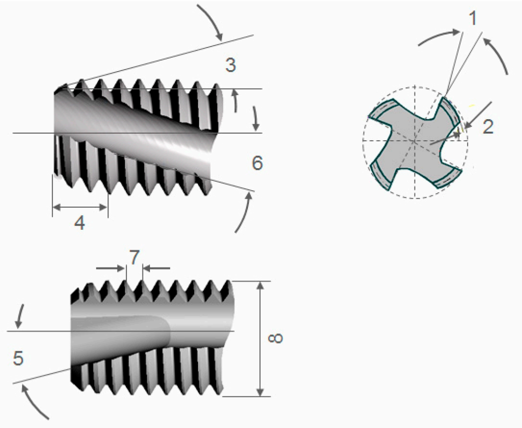
Problem	Cause	Solution
Rapid flank wear 	1. Highly abrasive material. 2. Cutting speed too high. 3. Infeed depths too shallow. 4. Insert is above center line.	1. Wrong grade. Choose a more wear resistant grade. 2. Reduce cutting speed. 3. Reduce number of infeeds. 4. Correct center height.
Abnormal flank wear   Poor surface on one flank of thread.	1. Incorrect method for flank infeed. 2. Insert inclination angle does not agree with the lead angle of the thread.	1. Change method of flank infeed for F-geometry and A-geometry; 3 - 5° from flank, for C-geometry; 1° from flank. 2. Change shim to obtain correct angle of inclination.
Vibration 	1. Incorrect clamping of the workpiece. 2. Incorrect setup of the tool. 3. Incorrect cutting data. 4. Incorrect center height.	1. Use soft jaws. 2. When using tail stock, optimize centering hole of component and check pressure of tail stock/face drive. Minimize overhang of tool. Check that the clamping sleeve for bars is not worn. Use 570-3 anti-vibration bars. 3. Increase cutting speed; if this does not help, lower the speed dramatically and try F-geometry. 4. Adjust center height.

Problem	Cause	Solution
Poor surface finish	1. Cutting speed too low. 2. The insert is above the center height. 3. Uncontrolled chips.	1. Increase cutting speed. 2. Adjust center height. 3. Use C-geometry and modified flank infeed.
Poor chip control	1. Incorrect method of infeed. 2. Incorrect thread geometry.	1. Modified flank infeed 3 - 5°. 2. Use C-geometry with modified flank infeed 1°.
Shallow profile	1. Wrong center height. 2. Insert breakage. Excessive wear.	1. Adjust center height. 2. Change cutting edge.
Incorrect thread profile	1. Unsuitable thread profile (angle of thread and nose radius) external inserts used for internal operation or vice versa. 2. Wrong center height. 3. Holder not 90° to center line. 4. Pitch error in machine.	1. Correct tool, shim and insert combination. 2. Adjust center height. 3. Adjust to 90°. 4. Correct the machine.
Excessive edge pressure 	1. Work hardening material in combination with infeed depths which are too shallow for the geometry. 2. Excessive pressure on cutting edge can cause chipping. 3. Profile with too small thread profile angle.	1. Reduce the number of infeeds. Change to F-geometry. 2. Change to a tougher grade. 3. Use modified flank infeed.

Tapping

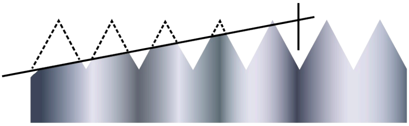
Tapping theory

Definitions of terms



1. Rake angle
2. Relief (clearence)
3. Chamfer angle
4. Chamfer (length)
5. Spiral point angle
6. Spiral angle
7. Pitch
8. Outer diameter

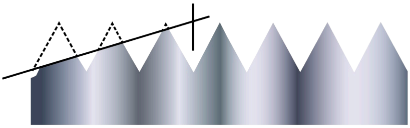
Long chamfer



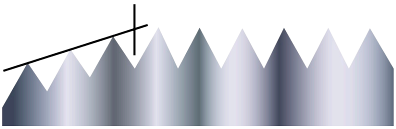
- High torque
- Best surface quality
- Thin chips
- Low pressure at the chamfer
- Longer tool life
- Most common for spiral point tap.

Medium chamfer

Cutting tap

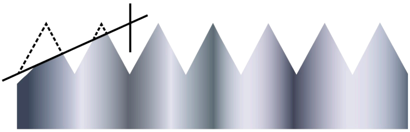


Forming tap

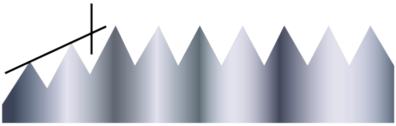


Short chamfer

Cutting tap



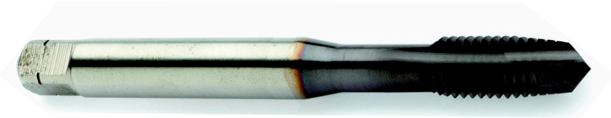
Forming tap



Different standards

- ISO
- ANSI

ISO and ANSI have quite short OAL (overall length) and are rather similar.
Except for the shank diameter which is in inch for ANSI and metric for ISO.

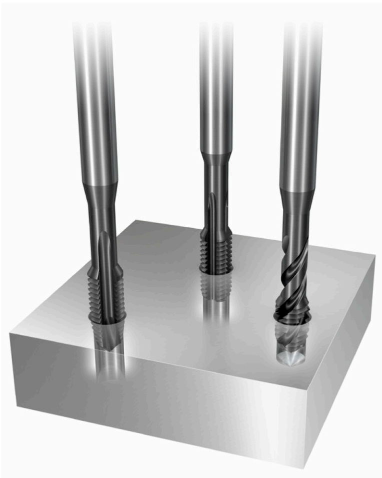


- DIN
- DIN/ANSI

DIN is a long version and metric.
DIN ANSI is a mix of both, with shank diameter from ANSI and OAL from DIN.

Tapping process

Different types of tapping processes



Geometries for different types of holes

Spiral point tap for through holes

- The strongest tap style
- Suited for tough conditions
- Pushes the chips forward through the hole
- Tap for through hole.

Spiral flute tap for blind holes

- The most common tap style
- Drives the chips up along the shank
- Tap for blind holes.

Straight flute tap for all holes

- For short chipped material like cast iron
- Often used in automotive industry, e.g. pumps and valves
- Can be used for all types of holes and depths.

Forming tap - a chip free tap solution

- A chip-free tap solution
- For soft steel, stainless steel and aluminum
- Can be used for all types of holes and depths
- Increases the strength of the thread in some materials, e.g. aluminum.

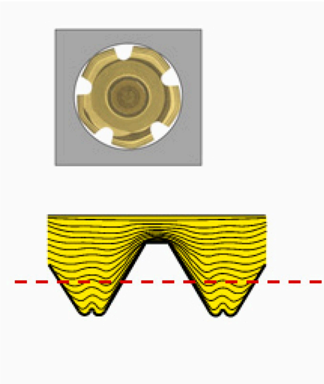
A Turning B Parting and grooving C Threading D Milling E Drilling F Boring G Tool holding H Machinability Other information

Forming and tapping processes

Forming tap

The thread is formed by deforming the material.

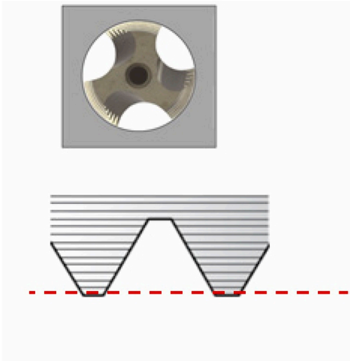
No chips are generated.



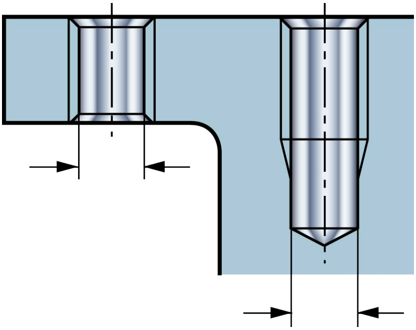
Cutting tap

The tap cuts the material.

Chips are generated.



Hole size and tolerances



Basic calculation of hole size, cutting taps

D = TD - TP

D = Hole diameter (mm, inch)

TD = Nominal thread diameter (mm, inch)

TP = Thread pitch (mm, inch)

Hole size for M10 x 1.5 cutting tap = 8.5 mm (8.5 = 10 - (1.5)

Hole size for 1/4" - 20 cutting tap = .2008" (.2008" = 1/4 - (20).

Basic calculation of hole size, forming taps

D = TD - (TP/2)

D = Hole diameter (mm, inch)

TD = Nominal thread diameter (mm, inch)

TP = Thread pitch (mm, inch)

Drill size for M10 x 1.5 forming tap = Hole size for 1/4" - 20 cutting tap = .2008" (.2008" = 1/4 - (20) mm (9.3 = 10 - (1.5/2))

Drill size for 1/4" - 20 forming tap = .2264" (.2264" = 1/4 - (20/2)).

Coolant

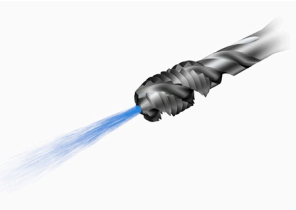
Important for successful performance

Coolant supply is essential in tapping and influences

- Chip evacuation
- Thread quality
- Tool life.

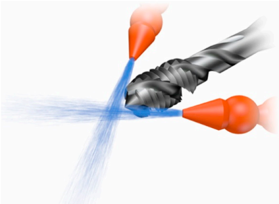
Coolant supply

Internal or external coolant supply



- Always to be preferred to improve chip evacuation, especially in long chipping materials and when threading deeper holes (2-3 x D)
- To be preferred for hole depths above 3 times the diameter.

External coolant supply



- The most common coolant method
- Can be used when chip formation is good
- To improve chip evacua-tion, at least one coolant nozzle (two if drill is stationary) should be directed closely to the tool axis.

Different cutting fluid/ emulsion

Three mail alternativ

- Mineral oil based
- Synthetic coolant
- Straight oil.

Two more options

- Vegetable oil based
- Semi synthetic.

Always be aware of

- Type of cutting fluid used in the machine
- Oil content.

Tool holding for tapping

Overview

Floating rubber collet chuck

Allows a certain amount of play to enable a proper path. Often used in manual and small tuning machines.

Coromant Capto®

Benefits and recommendations

- Rubber collets cover a wide clamping range
- Tension and compression to eliminate feed error.

Rigid ER collet chuck

With this approach there is no tension/compression play. That means motion of the spindle and axis movement has to be precisely synchronized. This requires a more sophisticated CNC controller.

Rigid tapping with ER collet chuck

Benefits and recommendations

- Rigid tapping is often faster
- Tooling cost is lower (rigid holders cost less than tension/compression holders)
- More compact and reliable than tension/compression holders
- Can result in a more accurate thread.

Note! Increased forces on the tap results in reduced tool life. Does not reverse quick enough at high speeds, say 6000 rpm.

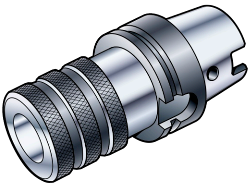
Quick change tapping chuck

First choice for standard tapping operations. All-round, lower volume production. Mainly for older, non-stable machines.

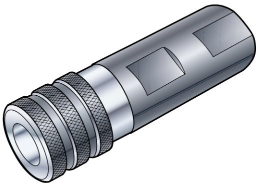
Benefits and recommendations

- Easy tap holding with quick change
- Tension and compression to eliminate feed error
- Adaptors with or without clutch.

Coromant Capto®



HSK solid holder



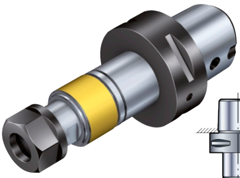
Weldon solid holder

Synchronous feed tap chucks

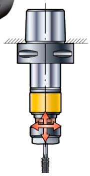
Rigid tap holder with micro float compensation for elimination of oversized threading. First choice for CNC machine tools and synchronized tapping operations.

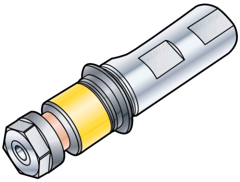
Benefits and recommendations

- High volume production / high precision
- Reduces thrust force on tap flanks
- Limited actual compensation provides accurate depths
- Designed for internal high pressure coolant.

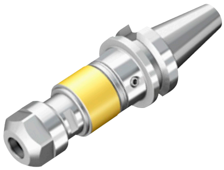


Coromant Capto®





Weldon shank holder



MAS-BT solid holder

Milling

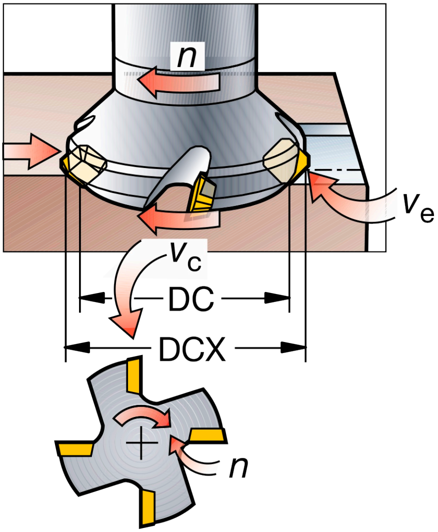
Milling is performed with a rotating, multi-edge cutting tool which performs programmed feed movements against a workpiece in almost any direction.

Milling is mostly applied to generate flat faces, but with the development of machines and software there are increasing demands to produce other forms and surfaces.

Milling theory

Definitions of terms

Spindle speed, cutting speed and cutter diameter



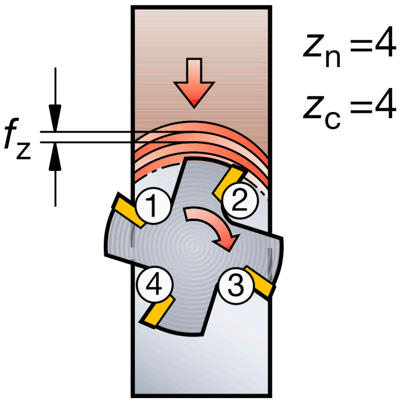
n	= Spindle speed, rpm (revolutions per minute)
v_c	= Cutting speed m/min (ft/min)
v_e	= Effective cutting speed m/min (ft/min)
DC	= Cutter diameter mm (inch)
DCX	= Maximum cutting diameter mm (inch)

Spindle speed (n) in rpm is the number of revolutions the milling tool on the spindle makes per minute.

Cutting speed (v_c) in m/min (ft/min) indicates the surface speed at which the cutting edge machines the workpiece.

Specified cutter diameter (DCX), having an effective cutting depth to diameter (DC), which is the basis for the cutting speed v_c or v_e.

Feed, number of teeth and spindle speed



f_z = Feed per tooth mm/tooth (inch/tooth)

v_f = Table feed mm/min (inch/min)

z_n = Number of cutter teeth (pcs)

z_c = Effective number of teeth (pcs) [in engagement]

f_n = Feed per revolution mm/rev (inch/rev) [f_z x z_c]

n = Spindle speed (rpm)

$$v_f = f_z \times z_c \times n \quad \text{mm/min (inch/min)}$$

Feed per tooth, f_z mm/tooth (inch/tooth), is a value in milling for calculating the table feed. The feed per tooth value is calculated from the recommended maximum chip thickness value.

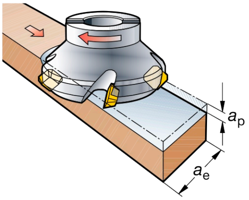
Feed per minute, v_f mm/min (inch/min), also known as the table feed, machine feed or feed speed is the feed of the tool in relation to the workpiece in distance per time-unit related to feed per tooth and number of teeth in the cutter.

The number of available cutter teeth in the tool (z_n) varies considerably and is used to determine the table feed while the effective number of teeth (z_c) is the number of effective teeth in cut.

Feed per revolution (f_n) in mm/rev (inch/rev) is a value used specifically for feed calculations and often to determine the finishing capability of a cutter.

Definitions of terms

Depth of cut



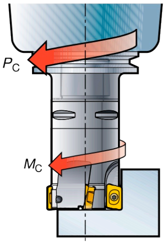
a_e = Radial depth of cut mm (inch) [working engagement]

a_p = Axial depth of cut mm (inch)

Axial depth of cut, a_p mm (inch), is what the tool removes in metal on the face of the workpiece. This is the distance the tool is set below the unmachined surface.

Radial cutting width, a_e mm (inch), is the width of the component engaged in cut by the diameter of the cutter. It is the distance across the surface being machined or, if the tool diameter is smaller, that is covered by the tool.

Net power, torque and specific cutting force



a_p = Axial depth of cut mm (inch)

a_e = Radial depth of cut mm (inch) [working engagement]

v_f = Table feed mm/min (inch/min)

k_c = Specific cutting force N/mm² (lbs/inch²)

P_c = Net power kW (Hp)

M_c = Torque Nm (lbf ft)

The net power (P_c) is the power the machine must be able to provide to the cutting edges in order to drive the cutting action. The efficiency of the machine must be taken into consideration when selecting cutting data.

The torque (M_c) is the torque value produced by the tool during cutting action, which the machine must be able to provide.

The specific cutting force value (k_{c1}) is a material constant, expressed in N/mm² (lbs/inch²). The values can be found in our main ordering catalog and technical guide.

Metric

$$P_c = \frac{a_p \times a_e \times v_f \times k_c}{60 \times 10^6} \quad \text{kW}$$

Inch

$$P_c = \frac{a_p \times a_e \times v_f \times k_c}{396 \times 10^3} \quad \text{Hp}$$

Metric

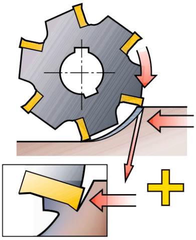
$$M_c = \frac{P_c \times 30 \times 10^3}{\pi \times n} \quad \text{Nm}$$

Inch

$$M_c = \frac{P_c \times 16501}{\pi \times n} \quad \text{lbf ft}$$

Climb or conventional milling

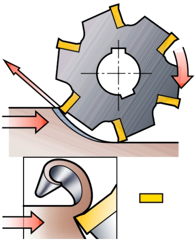
Climb milling – preferred method



Using climb milling (also referred to as down milling), the burnishing effect is avoided, resulting in less heat and minimal work-hardening tendency.

- In climb milling, the insert starts its cut with a large chip thickness.

Conventional milling



The feed direction of the workpiece is opposite to that of the cutter rotation at the area of cut.

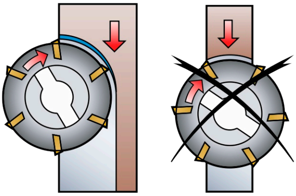
- In conventional milling (also referred to as up milling) the chip thickness starts at zero and increases to the end of the cut.

Always use climb milling for best cutting conditions.

Cutter diameter and position

The selection of milling cutter diameter is usually made on the basis of the workpiece width with the availability of the machine power also being taken into account.

The position of the cutter in relation to the workpiece engagement, and the contact which the cutter teeth have, are vital factors for a successful operation.

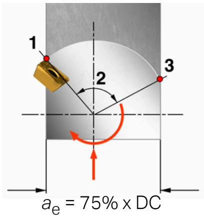


- Cutter diameter should be 20 – 40% larger than the width of cut.
- 2/3 rule (i.e., 150 mm (5.906 inch) cutter)
  - 2/3 in cut, 100 mm (3.937 inch)
  - 1/3 out of cut, 50 mm (1.969 inch).
- By moving the milling cutter off the center, a more constant and favorable direction of cutting forces will be obtained.

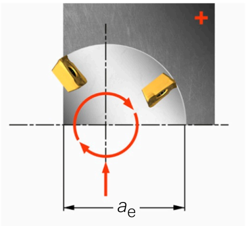
Chip formation through cutter position

The cutting edge in a radial direction engages with the workpiece in three different phases:

1. Entrance into cut
2. Arc of engagement in cut
3. Exit from cut.

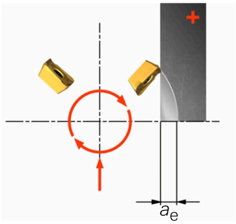


DC = Cutter diameter
a_e = working engagement



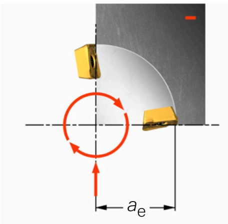
The centerline of the cutter is well inside the workpiece width, a_e >75% of DC.

- Most favorable cutting conditions and optimized use of the cutter diameter.
- The initial impact at the entry of cut is taken up further along the cutting edge, away from the sensitive tip.
- The insert leaves the cut gradually.



The centerline of the cutter is well outside the workpiece width, a_e <25% of DC.

- The angle of entry is positive
- The impact at the entry is taken up by the outermost tip of the insert and the load is gradually taken up by the the tool.



The centerline of the cutter is in line with the workpiece edge, a_e = 50% of DC.

- Not recommended.
- The shock loads at the cutting edge are very high at entry.

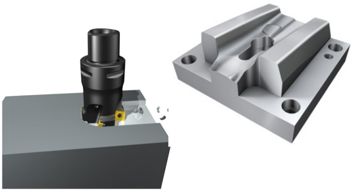
+ = Recommended cutter position.

– = Not recommended cutter position.

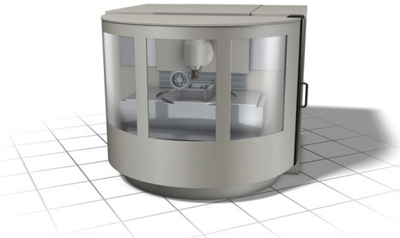
Selection procedure

Production planning process

1 Component — Type of operation and method

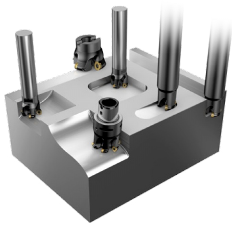


Workpiece material and quantity



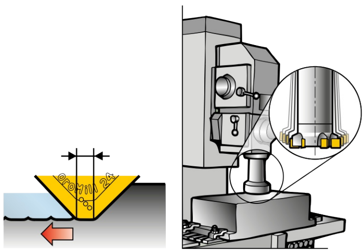
2 Machine — Machine parameters

3 Choice of tool — Select type of cutter



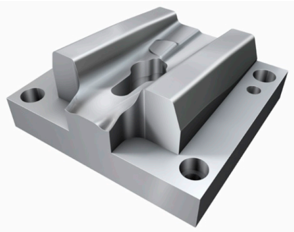
4 How to apply — Cutting data, method etc.

5 Troubleshooting — Remedies and solutions



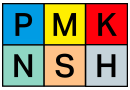
1. Component and the workpiece material

Parameters to be considered



Geometric shape

- Flat surface
- Deep cavities
- Thin walls/bases
- Slots.



Material

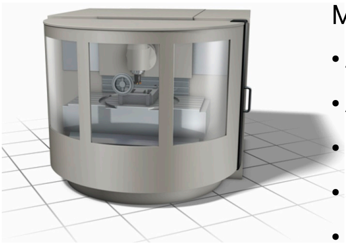
- Machinability
- Cast or pre-machined
- Chip forming
- Hardness
- Alloy elements.

Tolerances

- Dimensional accuracy
- Surface finish
- Part distortion
- Surface integrity.

2. Machine parameters

Condition of the machine and setup



Machine

- Available power
- Age/condition – stability
- Horizontal/vertical
- Spindle type and size
- Number of axes/ configuration
- Workpiece clamping.

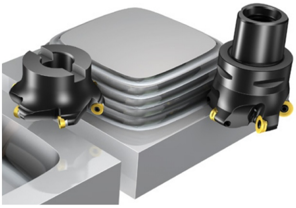
Tool holding

- Long overhang
- Poor holding
- Axial/radial runout.

3. Choice of tools

Different ways to optimize milling

Cutters with round inserts



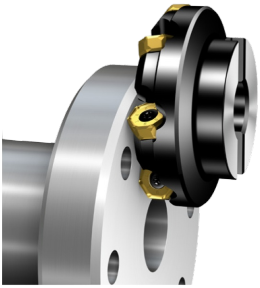
Advantages

- Robust milling cutters
- Very flexible for face milling and profiling
- High performance multi-purpose cutters.

Disadvantages

- Round inserts require more stable machines.

45° face mill



Advantages

- General choice for face milling
- Balanced radial and axial cutting forces
- Smooth entry into cut.

Disadvantages

- Max cutting depth 6-10 mm (.236-.394 inch).

90° square shoulder face mill

Advantages

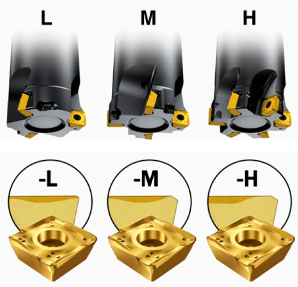
- Great versatility
- Large depth of cut
- Low axial cutting forces (thin workpieces)
- Light-cutting inserts with true four edges.

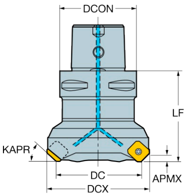
Disadvantages

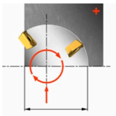
- Feed per tooth is relatively low while $f_z = h_{ex}$.

4. How to apply

Important application considerations







Number of cutting edges/pitch

- Selecting the right number of edges or pitch is very important.
- It affects both productivity and stability.

Insert geometry

- Select between a geometry for Light, Medium or Heavy machining.

Stability

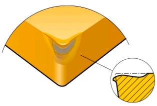
- Choose largest possible spindle size or outer diameter.

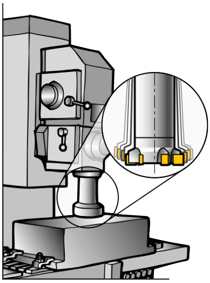
Chip formation through cutter positioning

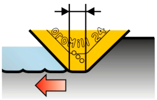
- Always use climb milling
- Move the cutter off the center
- Use a cutter with a diameter 20–50% larger than the cut.

5. Troubleshooting

Some areas to consider







Insert wear and tool life

- Check the wear pattern and if necessary adjust the cutting data accordingly.

Vibration

- Weak fixture
- Long tool overhang
- Weak workpiece
- Size of spindle taper.

Unsatisfactory surface finish

- Check spindle runout
- Use wiper inserts
- Decrease feed per tooth.

System overview

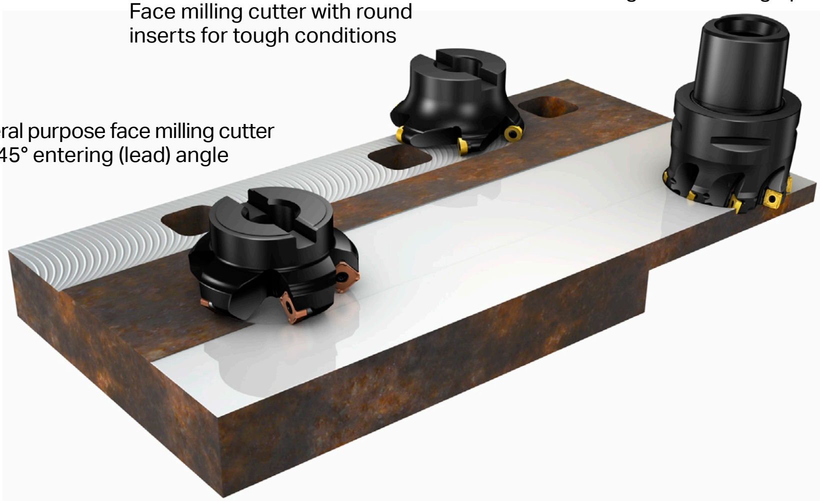
Face milling

Cutters for general use

Face and square shoulder milling cutter for light face milling operations

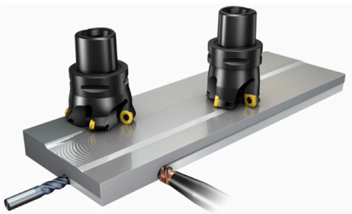
Face milling cutter with round inserts for tough conditions

General purpose face milling cutter with 45° entering (lead) angle



Dedicated cutters

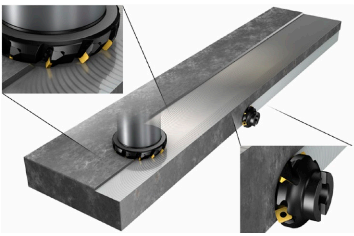
High feed face milling



Face milling cutters for cast iron machining

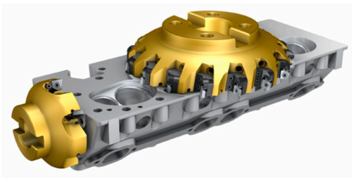
K

Heavy duty face milling



Face milling cutters for aluminum machining

N

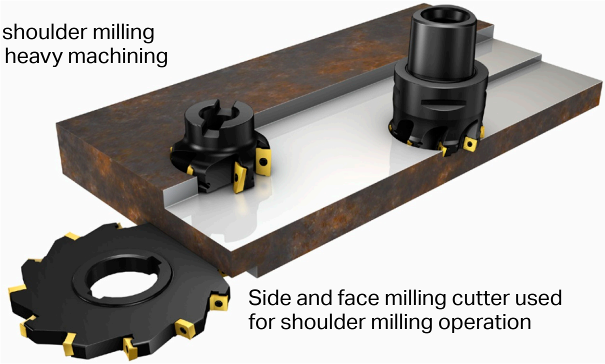


Shoulder milling

Cutters for general use

Face and shoulder milling for light shoulder milling operations

Face and shoulder milling cutter for heavy machining

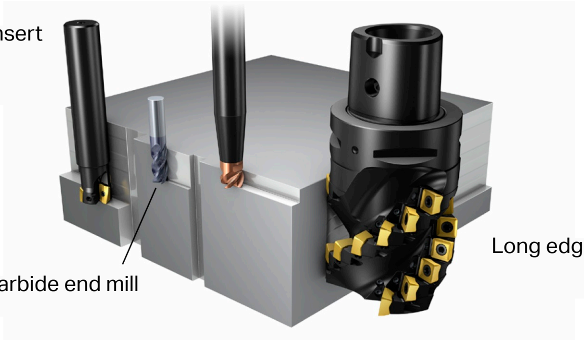


Side and face milling cutter used for shoulder milling operation

Dedicated end mills and long edge cutters

End mill with exchangeable, solid carbide head

Indexable insert end mill

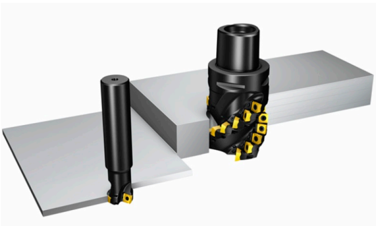


Solid carbide end mill

Long edge milling cutter

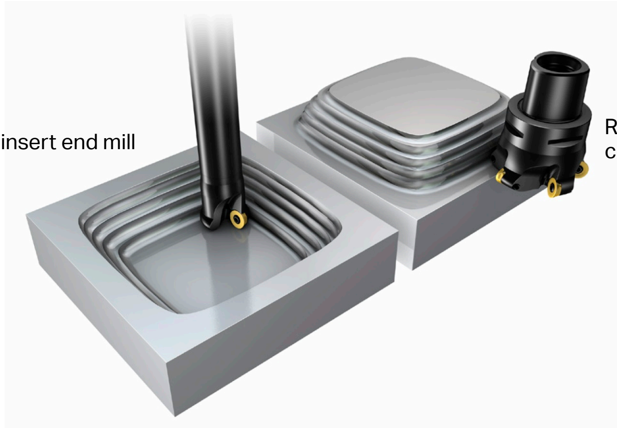
Deep shoulder milling

Edging with square shoulder milling cutters

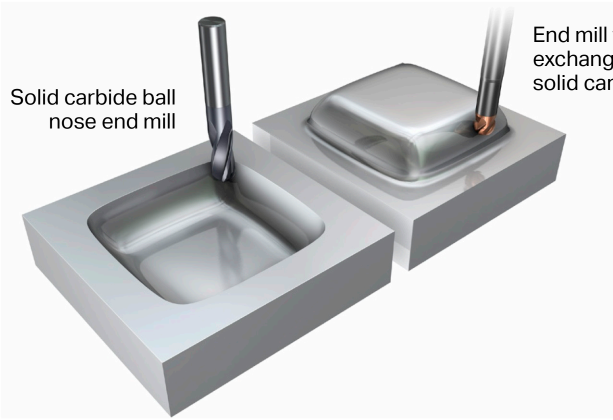


Profiling

Cutters for general use – roughing



Cutters for general use – finishing



Other methods

Turn milling

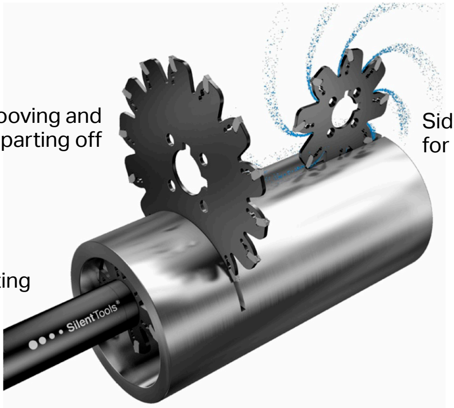
Blade milling

Groove milling

Cutters for general use – radial groove milling

Cutters for grooving and parting off

Internal grooving and slotting

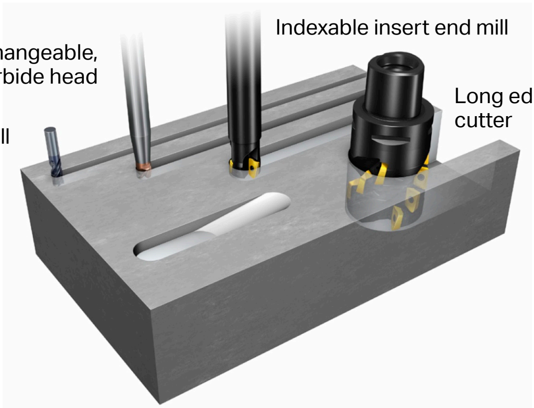


Side and face mill for groove milling

Cutters for general use – axial slot milling

End mill with exchangeable, solid carbide head

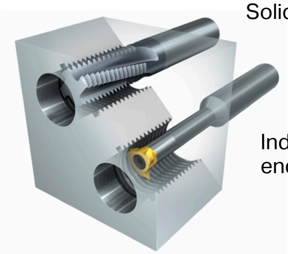
Solid carbide end mill



Indexable insert end mill

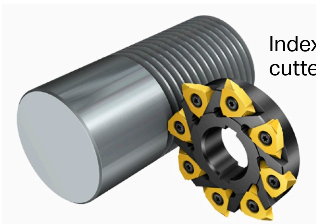
Long edge milling cutter

Thread milling and shallow grooving



Solid carbide end mill

Indexable insert end mill

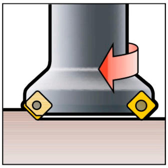
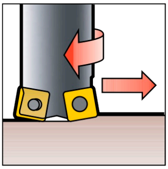
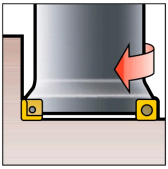
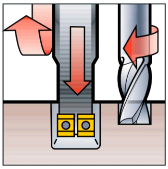
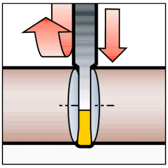
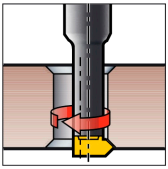
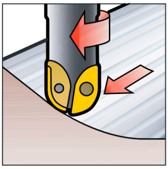
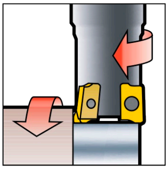
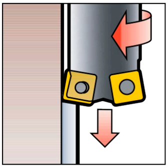
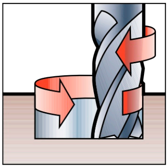
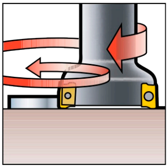
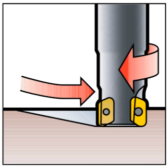
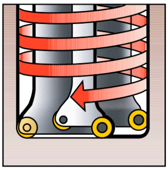
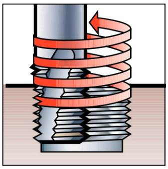


Indexable insert cutter

Overview of milling operations

Modern milling is a very universal machining method. During the past few years, hand-in-hand with machine tool developments, milling has evolved into a method that machines a very broad range of configurations. The choice of methods in multi-axis machinery makes milling a strong contender for producing holes, cavities, surfaces that used to be turned, threads, etc.

Tooling developments have also contributed to the new possibilities, along with the gains in productivity, reliablity and quality consistency that have been made in indexable insert and solid carbide technology.

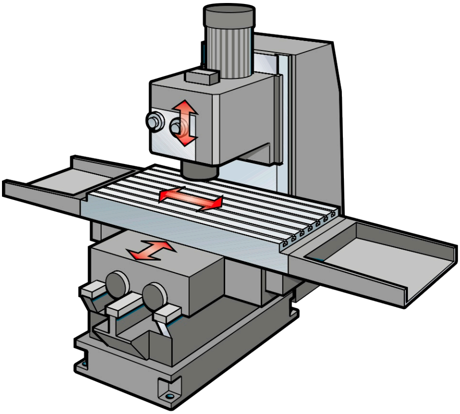
Face milling	High-feed milling	Shoulder milling	Groove milling
			
Parting off	Chamfering	Profile milling	Turn milling
			
Plunge milling	Trochoidal milling	Circular milling	Linear ramping
			
Helical interpolation	Thread milling		
			

Milling methods

Milling machines may be manually operated, mechanically automated, or digitally automated via computer numerical control (CNC).

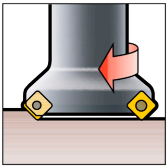
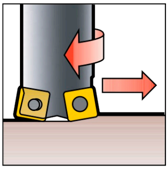
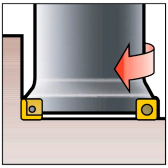
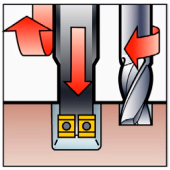
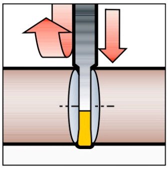
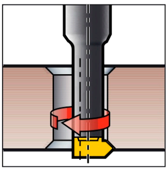
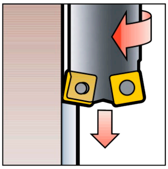
Conventional milling methods

Vertical milling machines



In conventional 3-axis machines, milling most frequently entails the generation of flat faces, shoulders and slots.

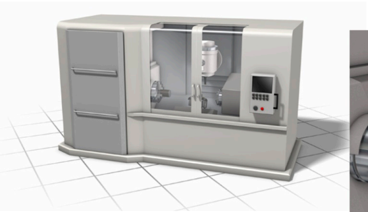
Surfaces and forms, other than those described below, are increasing steadily as the number of five-axis machining centers and multi-task machines grows.

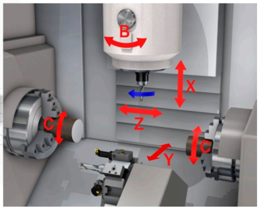
Face milling	High-feed milling	Shoulder milling	Groove milling
			
Parting off	Chamfering	Plunge milling	
			

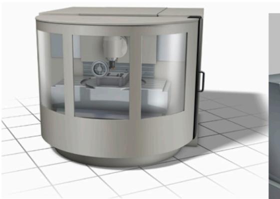
Advanced milling methods

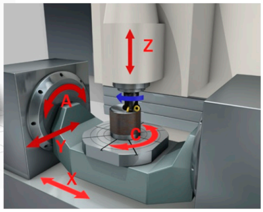
Modern 4 to 5 axis machines

Today, machines are developing in all directions. Turning centers now have milling capability through driven tools, and machining centers have turning capability via turnmill or mill-turn machines. CAM developments mean that 5-axis machines are increasing.



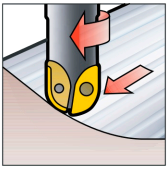
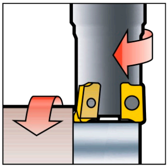
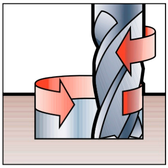
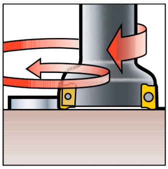
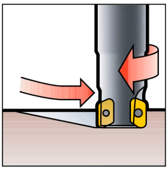
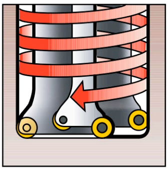
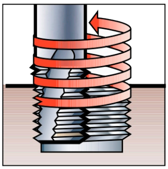




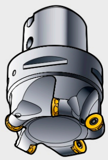
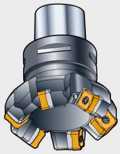
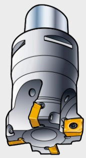


The results of these trends and the development of methods put new demands and opportunities on the tooling, such as:

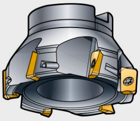
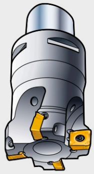
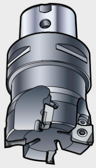
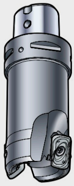
- Increased flexibility
- Fewer machines/setups to complete a component
- Reduced stability
- Longer tool lengths
- Lower depth of cut.

Profile milling	Turn milling	Trochoidal milling	Circular milling
			
Linear ramping	**Helical interpolation**	**Thread milling**	
			

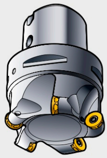
Positioning of cutters for face milling

Type of milling cutter				
Considerations	Round inserts	10-25°	45°	90°
Machine/spindle size	ISO 40, 50	ISO 40, 50	ISO 40, 50	ISO 30, 40, 50
Stability requirement	High	High	Medium	Low
Roughing	Very good	Good	Very good	Acceptable
Finishing	Acceptable	Acceptable	Very good	Good
Cutting depth a_p	Medium	Small	Medium	Large
Versatility	Very good	Good	Good	Very good
Productivity	Very good	Very good	Very good	Good

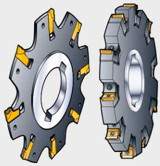
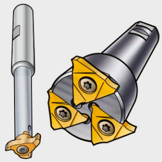
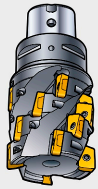
Positioning of cutters for shoulder milling

Type of milling cutter / Considerations				
	90°	90°	90°	90°
Machine/spindle size	ISO 40, 50	ISO 30, 40, 50	ISO 40, 50	ISO 30, 40, 50
Stability requirement	High	High	Medium	Low
Roughing	Very good	Good	Acceptable	Good
Finishing	Acceptable	Acceptable	Very good	Good
Cutting depth a_p	Large	Medium	Small	Large
Material	All	All	Aluminum	Aluminum
Versatility	Very good	Very good	Acceptable	Good

Positioning of cutters for profile milling

Type of milling cutter / Considerations	 Round inserts	 Ball nose indexable	 Ball nose exchangable	 Ball nose solid carbide
Machine/spindle size	ISO 40, 50	ISO 40, 50	ISO 30, 40	ISO 30, 40
Stability requirement	High	Medium	Medium	Low
Roughing	Very good	Good	Acceptable	Acceptable
Finishing	Acceptable	Acceptable	Very good	Very good
Cutting depth a_p	Medium	Medium	Small	Small
Versatility	Very good	Very good	Very good	Very good
Productivity	Very good	Good	Good	Good

Positioning of cutters for slots and grooves

Type of milling cutter / Considerations			
	Groove Side and face	Grooving	Long edge
Machine/spindle size	ISO 50	ISO 40, 50	ISO 40, 50
Groove open	Open	Open	Open
Groove closed	Closed	Closed	Closed
Cutting width	Small	Small	Large
Cutting depth a_p	Medium-Large	Small	Medium-Large
Versatility	Limited	Good	Good

Type of milling cutter / Considerations			
	Indexable insert end mill	Exchangable-head end mill	Solid carbide end mill
Machine/spindle size	ISO 30, 40, 50	ISO 30, 40, 50	ISO 30, 40, 50
Groove open	Open	Open	Open
Groove closed	Closed	Closed	Closed
Cutting width	Medium	Small	Small
Cutting depth a_p	Medium	Small	Large
Versatility	Very good	Very good	Very good

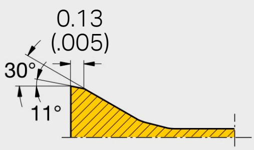
Choice of inserts and how to apply

Modern milling inserts for face milling operations.

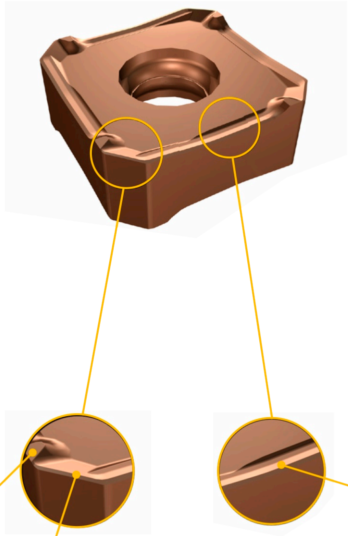
The design of a modern milling insert

Definitions of terms and geometry design

Corner design



- Cutting edge reinforcement 0.13 mm (.005 inch)
- Rake angle 30°
- Primary land 11°.

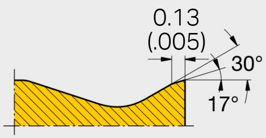


Corner reinforcement

Chip former

Main cutting edge design

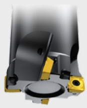
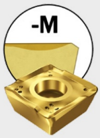
Main cutting edge design



- Cutting edge reinforcement 0.13 mm (.005 inch)
- Rake angle 30°
- Primary land 17°.

A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

Making the tool choice in milling

	First choice	
Low ←	Operation stability	→ High
	Cutter pitch	
 L	 M	 H
Coarse pitch (-L)	Close pitch (-M)	Extra close pitch (-H)
 -L	 -M	 -H
Light (-L)	Medium (-M)	Heavy (-H)
Wear resistant ←	Machining conditions/ Grades	→ Toughness
Good conditions	Average conditions	Difficult conditions

Type of application

a_p Depth of cut, mm (inch)

H

Heavy milling
- Operation for maximum stock removal and/or severe conditions.
- Larger depth of cut and feed rate.
- Operations requiring highest edge security.

M

Medium milling
- Most applications – general purpose milling.
- Medium operations to light roughing.
- Medium depth of cut and feed rate.

L

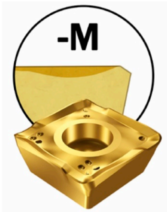
Light milling
- Operations at small depth of cut and low feed rates.
- Operations requiring low cutting forces.

f_n Feed, mm/tooth (inch/tooth)

Selecting the insert geometry

Light (-L)

- Extra positive
- Light machining
- Low cutting forces
- Low feed rates.



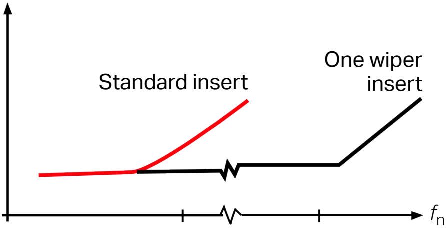
Medium (-M)

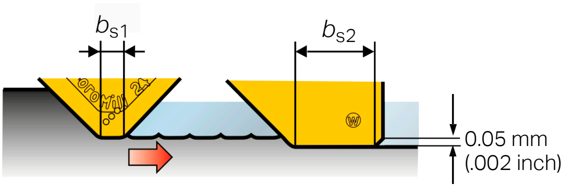
- General purpose geometry
- Medium feed rates
- Medium operations to light roughing.

Heavy (-H)

- Reinforced cutting edge
- Heavy machining
- Highest edge security
- High feed rates.

Achieving good surface finish in milling



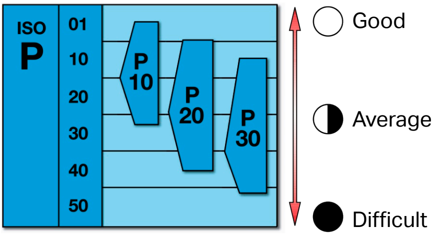


- Use wiper inserts for higher productivity and improved surface finish
- Limit the feed to 60% of the parallel land
- Mount the wiper inserts correctly
- Set the wiper inserts below other inserts.

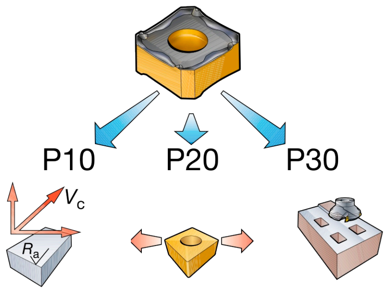
How to select insert grade

Select the geometry and grade according to the application.

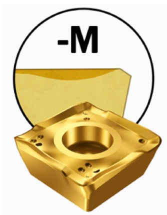
Build-up of a grade chart



Machining conditions



Define machining conditions



Good conditions

- Cutting depth 25% of max a_p or less
- Overhang under two times cutter diameter
- Continuous cuts
- Wet or dry machining.

Average conditions

- Cutting depth 50% of max a_p or more
- Overhang two to three times cutter diameter
- Interrupted cuts
- Wet or dry machining.

Difficult conditions

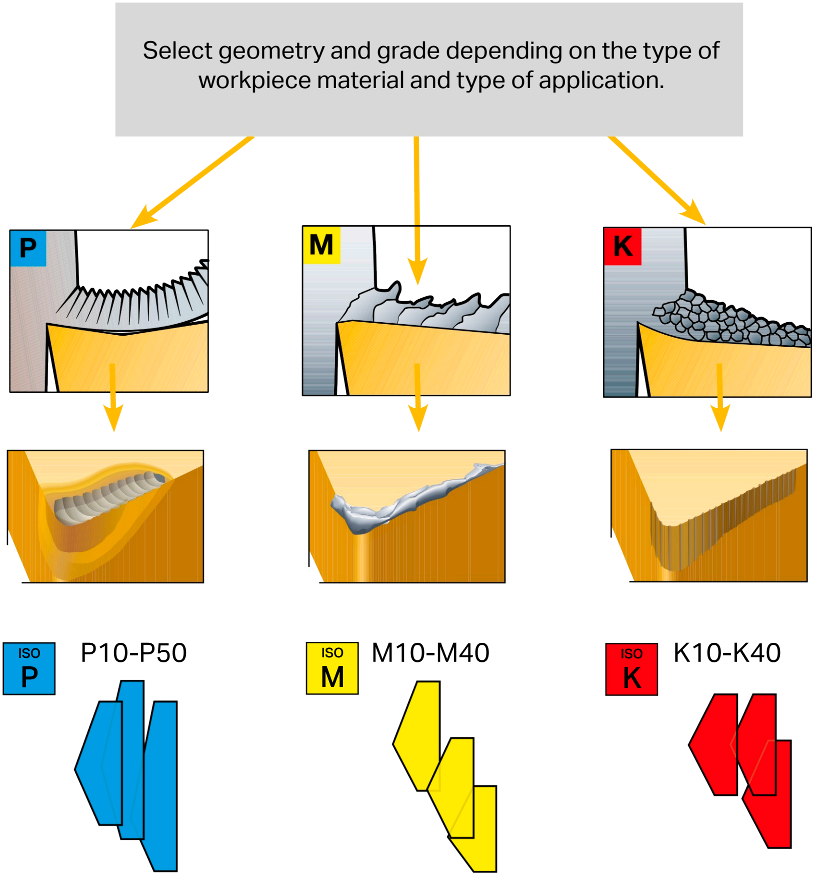
- Cutting depth 50% of max a_p or more
- Overhang over three times cutter diameter
- Interrupted cuts
- Wet or dry machining.

Dedicated grades for ISO P, M and K

Dedicated grades minimize tool wear development

The workpiece material influences the wear during the cutting action in different ways. Therefore dedicated grades have been developed to cope with the basic wear mechanisms, e.g.:

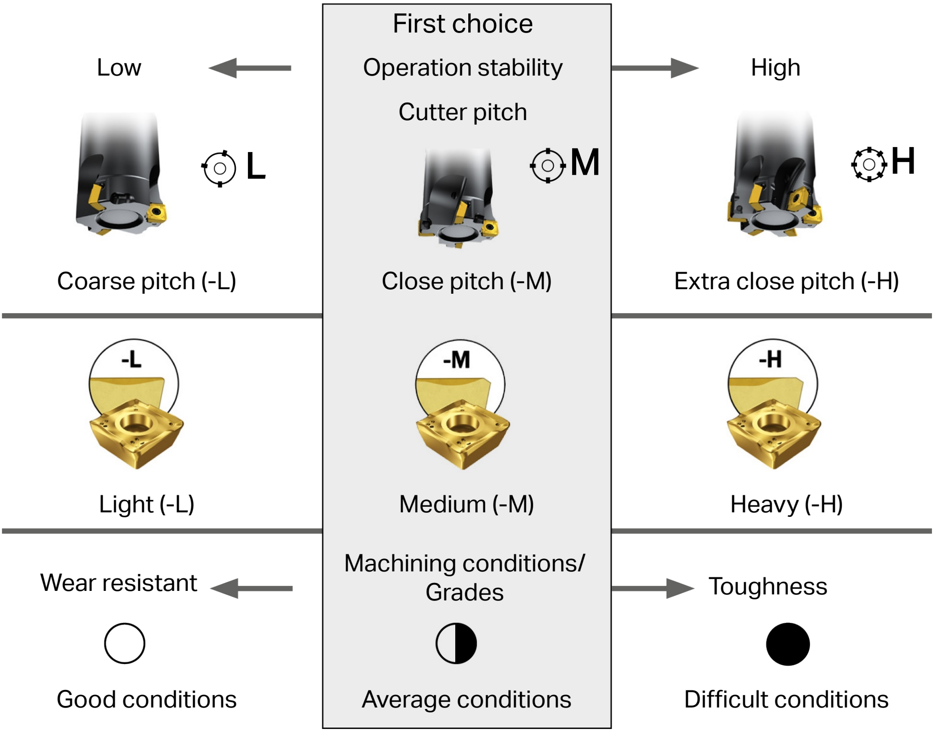
- Flank wear, crater wear and plastic deformation in steel
- Built-up edge and notch wear in stainless steel
- Flank wear and plastic deformation in cast iron.



Choice of cutter and how to apply

High performace face milling cutters for small to medium cutting depths.

Making the tool choice in milling



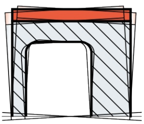
Selecting cutter pitches

Low ←	First choice Operation stability Cutter pitch	→ High

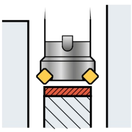
L

Coarse pitch (-L)

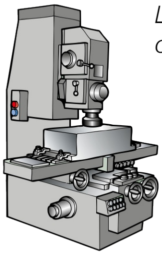
- Reduced number of inserts
- Limited stability
- Long overhang
- Small machines/limited horsepower
- Deep, full slotting operations
- Differential pitch.



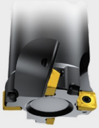
Limited stability



Long overhang



Limited horsepower



M

Close pitch (-M)

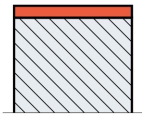
- General purpose
- Suitable for mixed production
- Small to medium machines
- Usually first choice.

First choice

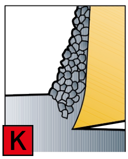
H

Extra close pitch (-H)

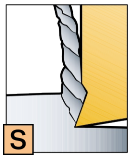
- High number of inserts for maximum productivity
- Stable conditions
- Short chipping materials
- Heat resistant materials.



Stable conditions



Cast iron (CMC 08)

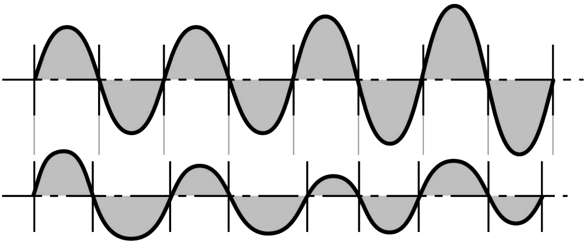


Heat resistant alloys (CMC 20)

Differential pitch

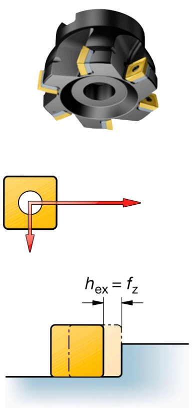
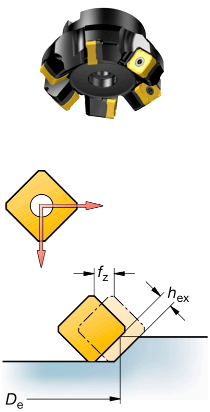
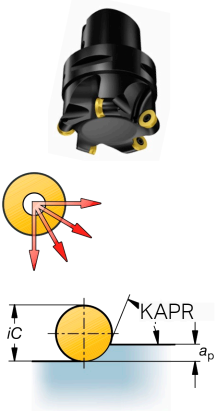
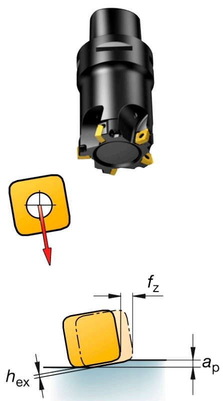
In general, the coarser the cutter pitch, the least chance of harmonic vibration. Sometimes, replacing a 16-tooth cutter with a 12-tooth tool ends chatter altogether. A differential-pitched cutter may be required in more difficult cases to eliminate troublesome harmonics.

Differential pitch cutters have uneven tooth spacing, which impacts the vibration amplitude of each tooth. Reducing the risk of vibration.



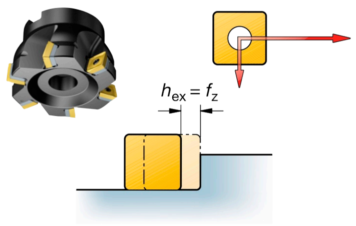
Differential pitch reduces the risk of vibration.

Cutting forces and entering angle

90° entering angle	45° entering angle	Round insert cutters	10° entering angle
			

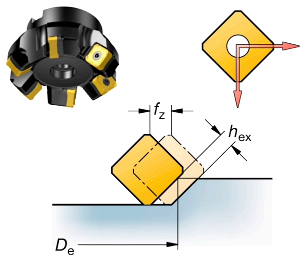
Axial and radial cutting forces

Effect of entering angle (90º)



- Thin-walled components
- Axially weak fixtured components
- Square shoulder
- $h_{ex} = f_z$ (In case a_e > 50% x DC).

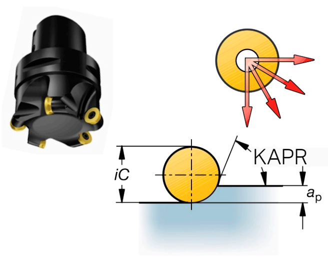
Effect of entering angle (45º)



- General purpose 1st choice
- Reduced vibration on long tool overhang
- Chip thinning effect allows increased productivity
- f_z = 1.41 x h_{ex} (Compensating for entering angle).

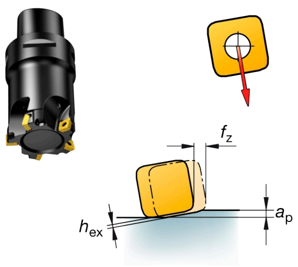
Effect of entering angle (Variable)

On round inserts, the chip load and entering angle vary with the depth of cut.



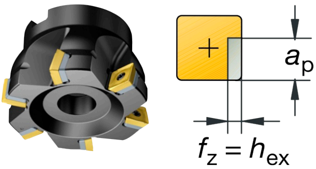
- Strongest cutting edge with multiple indexes
- General purpose cutter
- Increased chip thinning effect for heat resistant alloys
- h_{ex} = depends on a_p.

10° entering angle

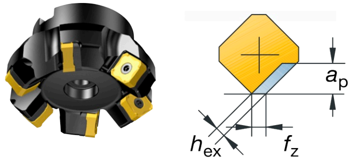


- High-feed milling cutters
- A thin chip is generated, allows very high feeds per tooth
- Axial cutting force is directed towards the spindle and stabilize it.

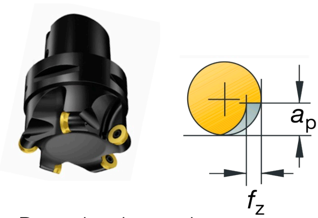
Feed compensation for different entering angles



90° = (f_z or h_{ex}) × 1.0



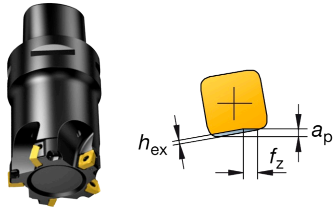
45° = (f_z or h_{ex}) × 1.41



Round = depends on a_p

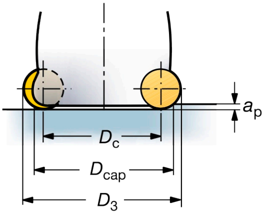
$$\sqrt{\frac{iC}{a_p}}$$

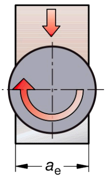
Formula for compensation in turning



10° = (f_z or h_{ex}) x 5.76

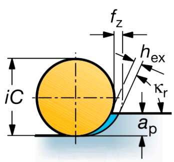
Formulas for cutters with round inserts

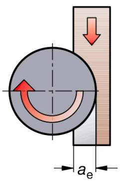




Max. cutting diameter at a specific depth (inch).

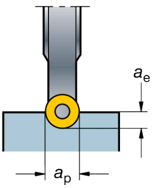
$$D_{cap} = DC + \sqrt{iC^2 - (iC - 2 \times a_p)^2}$$





Facemilling round insert (a_p<iC/2) (inch).

$$f_z = \frac{h_{ex} \times iC}{2 \times \sqrt{a_p \times iC - a_p^2}}$$



Side milling (a_e<D_{cap}/2) and round insert (a_p<iC/2) (inch).

$$f_z = \frac{h_{ex} \times iC \times D_{cap}}{4 \times \sqrt{a_p \times iC - a_p^2} \times \sqrt{D_{cap} \times a_e - a_e^2}}$$

A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

Calculating cutting data

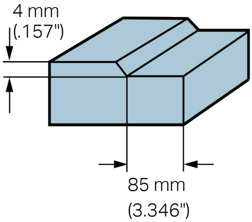
Example in face milling

Given:

- Cutting speed, v_c = 225 m/min (738 ft/min)
- Feed per tooth, f_z = 0.21 mm (.0082 inch)
- Number of cutter teeth, z_n = 5
- Cutter diameter, DC = 125 mm (4.921 inch)
- Cutting depth, a_p = 4 mm (.157 inch)
- Working engagement, a_e = 85 mm (3.346 inch)

Need:

- Spindle speed, n (rpm)
- Table feed, v_f (mm/min (inch/min)
- Metal removal rate, Q cm^3/min (inch^3/min)
- Power consumption kW (Hp)



Spindle speed

Given: v_c = 225 m/min (738 ft/min)

Metric

$$n = \frac{v_c \times 1000}{\pi \times DC} \quad \text{(rpm)}$$

$$n = \frac{225 \times 1000}{3.14 \times 125} = 575 \text{ rpm}$$

Inch

$$n = \frac{v_c \times 12}{\pi \times DC} \quad \text{(rpm)}$$

$$n = \frac{738 \times 12}{3.14 \times 4.921} = 575 \text{ rpm}$$

Table feed

Given: n = 575 rpm

Metric

$$v_f = n \times f_z \times z_n \quad \text{(mm/min)}$$

$$v_f = 575 \times 0.21 \times 5 = 600 \text{ mm/min}$$

Inch

$$v_f = n \times f_z \times z_n \quad \text{(inch/min)}$$

$$v_f = 575 \times .0082 \times 5 = 23.6 \text{ inch/min}$$

Metal removal rate

Given v_f = 600 mm/min (23.6 inch/min)

Metric

$$Q = \frac{a_p \times a_e \times v_f}{1000} \quad \text{(cm}^3\text{/min)}$$

$$Q = \frac{4 \times 85 \times 600}{1000} = 204 \text{ cm}^3\text{/min}$$

Inch

$$Q = a_p \times a_e \times v_f \quad \text{(inch}^3\text{/min)}$$

$$Q = .157 \times 3.346 \times 23.6 = 12.4 \text{ inch}^3\text{/min}$$

Net power consumption

Given: Material CMC 02.1

Metric

$$P_c = \frac{a_e \times a_p \times v_f \times k_c}{60 \times 10^6} \quad \text{(kW)}$$

Inch

$$P_c = \frac{a_e \times a_p \times v_f \times k_c}{396 \times 10^3} \quad \text{(Hp)}$$

Milling with large engagement

ISO	CMC No.	Material	Specific cutting force kc 1 N/mm²	Hardness Brinell HB	mc	CT530 Max chip thickne 0.1 – 0.15 – 0.2 Cutting speed v_c,
P		**Steel**				
		Unalloyed				
	01.1	C = 0.10 – 0.25%	1500	125	0.25	430–390–50
	01.2	C = 0.25 – 0.55%	1600	150	0.25	385–350–15
	01.3	C = 0.55 – 0.80%	1700	170	0.25	365–330–00
	01.4		1800	210	0.25	315–290–60
	01.5		2000	300	0.25	235–210–95
		Low alloyed (alloying elements 5%)				
	02.1	Non-hardened	1700	175	0.25	300–275–45
	02.2	Hardened and tempered	1900	300	0.25	195–180–60
		High alloyed (alloying elements > 5%)				
	03.11	Annealed	1950	200	0.25	230–205–85
	03.13	Hardened tool steel	2150	200	0.25	190–170–55
	03.21		2900	300	0.25	165–150–35
	03.22		3100	380	0.25	105–95–85
		Castings				
	06.1	Unalloyed	1400	150	0.25	305–280–50
	06.2	Low alloyed (alloying elements 5%)	1600	200	0.25	245–220–00
	06.3	High alloyed (alloying elements > 5%)	1950	200	0.25	180–160–45

	Specific cutting force k_c .016 lbs/in²	Hardness Brinell HB	
	216,500	125	0
	233,000	150	0
	247,000	170	0
	260,500	210	0
	291,500	300	0
	246,500	175	0
	278,500	300	0
	282,000	200	0
	311,000	200	0
	420,000	300	0
	448,500	380	0
	204,000	150	0
	230,500	200	0
	283,500	200	0

$$P_c = \frac{85 \times 4 \times 600 \times 1700}{60 \times 10^6} = 5.8 \text{ kW}$$

$$P_c = \frac{3.346 \times .157 \times 23.6 \times 246500}{396 \times 10^3} = 7.7 \text{ Hp}$$

The calculation above is approximate and valid for an maximum chip thickness (h_{ex}) of 0.1 mm (.0039 inch) .
For a more accurate value of power consumption (P_c) the k_c value should be calculated accordingly.

Metric

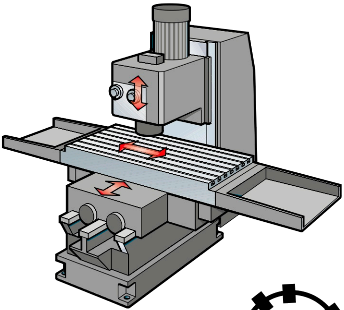
$$k_c = k_{c1} \times h_m^{-mc} \times \left(1 - \frac{\gamma_o}{100}\right) \quad \text{(N/mm}^2\text{)}$$

Inch

$$k_c = k_{c1} \times h_m^{-mc} \times \left(1 - \frac{\gamma_o}{100}\right) \quad \text{(lbs/inch}^2\text{)}$$

h_m = Average chip thickness
γ_o = Insert rake angle
m_c = Chip thickness compensation factor
k_c = Specific cutting force
k_{c1} = Specific cutting force for average chip thickness 0.1 mm (.0039 inch).

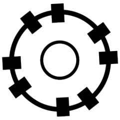
Application hints for milling



Power capacity
- Check power capability and machine rigidity, making sure that the machine can handle the cutter diameter required.

Stability of work piece
- Condition and considerations of component clamping.

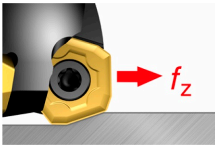
Overhang
- Machine with the shortest possible tool overhang on the spindle.



Select correct cutter pitch
- Use the correct cutter pitch for the operation to ensure that there are not too many inserts engaged in cut, as this may cause vibration.

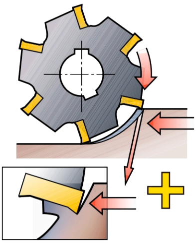
Cutting engagement
- Ensure there is sufficient insert engagement with narrow workpieces or when milling over voids.

Choice of insert geometry
- Use positive geometry indexable inserts whenever possible for smooth cutting action and lowest power consumption.



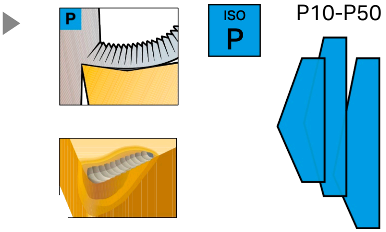
Up to 0.50 mm (.020 inch)

Use correct feed
- Ensure that the right feed per insert is used to achieve the right cutting action by use of the recommended maximum chip thickness.



Cutting direction
- Use climb (down) milling whenever possible.

Component consideration
- Work piece material and configuration. Also quality demands on the surface to be machined.

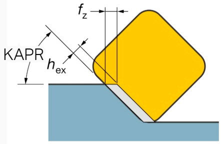


Choice of insert grade

- Select grade depending on the type of work-piece material and type of application.

Dampened milling tools

- For longer overhang of more than 4 times the tool diameter, vibration tendencies can become more apparent, and dampened cutters can improve the productivity radically.

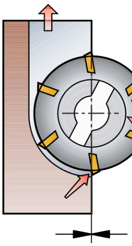


Entering angle

- Select the most suitable entering angle.

Cutter diameter

- Select the right diameter in relation to the workpiece width.

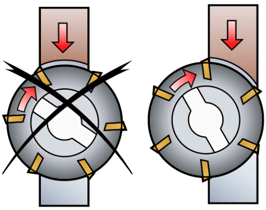


Cutter position

- Position the milling cutter correctly.

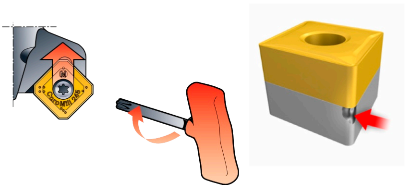
Cutter entrance and exit

- It can be seen that by rolling into cut, the chip thickness on exit is always zero, allowing higher feed and longer tool life.



Coolant

- Only use coolant if considered necessary. Milling is generally performed better without.



Maintenance

- Follow tool maintenance recommendations and monitor tool wear.