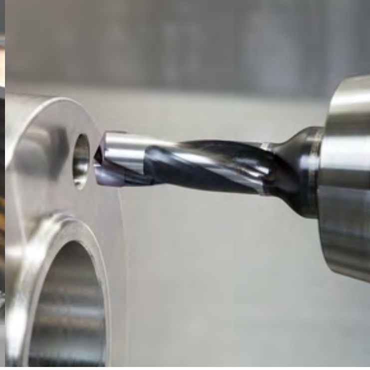
Training Handbook

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.


Turning

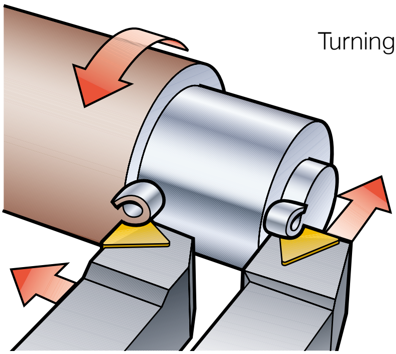
Turning generates cylindrical and rounded forms with a single-point tool. In most cases the tool is stationary with the workpiece rotating.

General turning operations

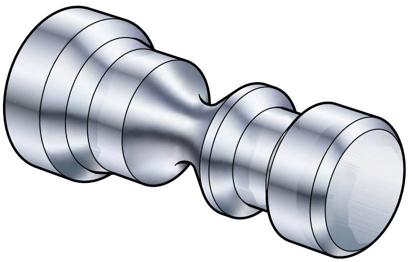
Turning is the combination of two movements – rotation of the workpiece and feed movement of the tool.

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.

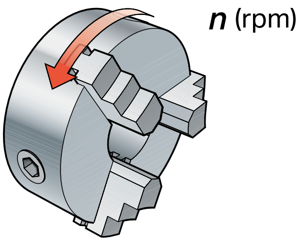


Three common turning operations:

- Longitudinal turning
- Facing
- Profiling

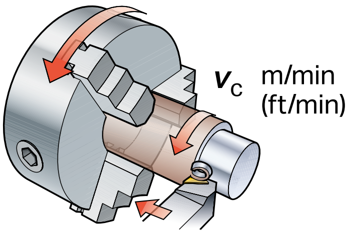
Definitions of terms

Spindle speed

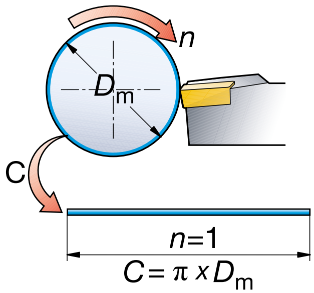


The spindle speed rpm (revolution per minute) is the rotation of the chuck and workpiece.

Cutting speed



The cutting speed is the surface speed, m/min (ft/min), at which the tool moves along the workpiece 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

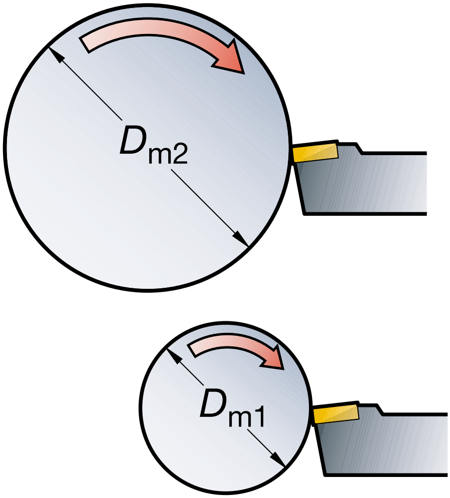
Metric

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

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

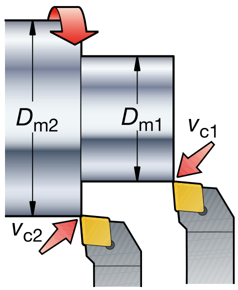
C = 3.14 x 3.937
= 12.362 inch

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

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

Inch

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

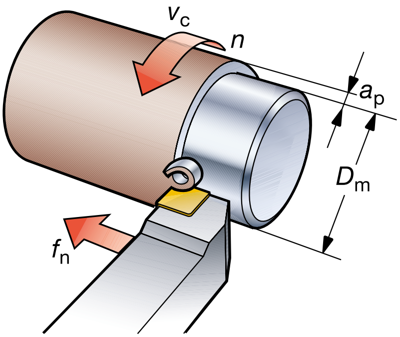
$$v_{c1} = \frac{3.14 \times 50 \times 2000}{1000} = 314 \text{ m/min}$$

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

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

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

Definitions of terms



n = spindle speed (rpm)

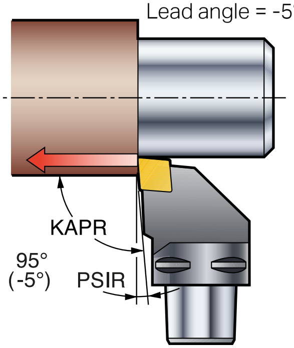
v_c = cutting speed m/min (ft/min)

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

a_p = depth of cut mm (inch)

KAPR = entering angle

PSIR = lead angle



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.

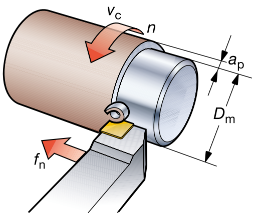
Entering angle KAPR, lead angle PSIR

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

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

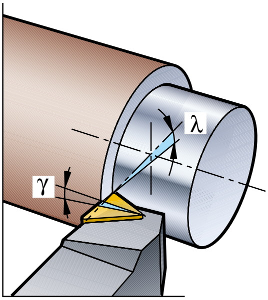
$$n = \frac{400 \times 1000}{3.14 \times 100} = 1274 \text{ r/min}$$

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



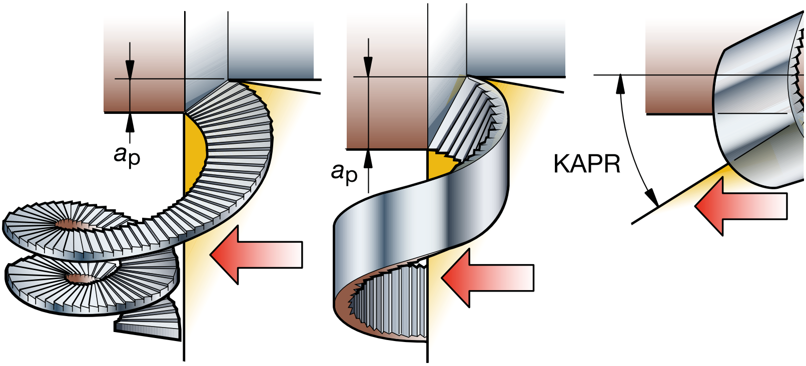
Rake angle

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.

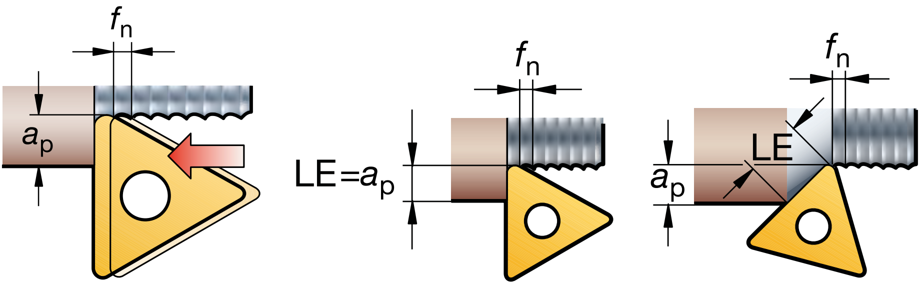
Cutting depth and chip formation



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

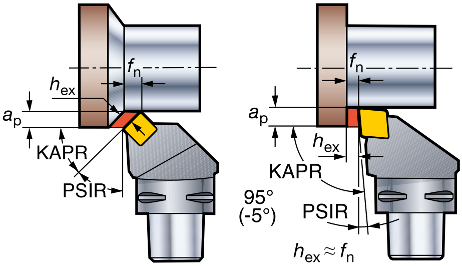
The effective cutting edge length (LE) relates to cutting depth and entering (lead) angle.

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°
$h_{ex} \approx f_n \times 0.71$

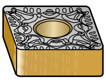
Entering angle KAPR (Lead angle PSIR)

- 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).

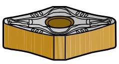
Possible entering (lead) angle positions for insert shapes



CNMG

Entering angle KAPR:
95°

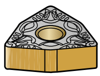
Lead angle PSIR:
-5°



DNMG

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

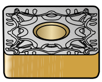
Lead angle PSIR:
-17.5°, -3°, 27.5°



WNMG

Entering angle KAPR:
95°

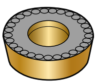
Lead angle PSIR:
-5°



SNMG

Entering angle KAPR:
45°, 75°

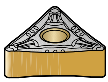
Lead angle PSIR:
45°, 15°



RCMT

Entering angle KAPR:
Variable

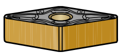
Lead angle PSIR:
Variable



TNMG

Entering angle KAPR:
93°, 91°, 60°

Lead angle PSIR:
-3°, -1°, 30°



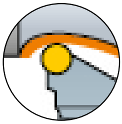
VNMG

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

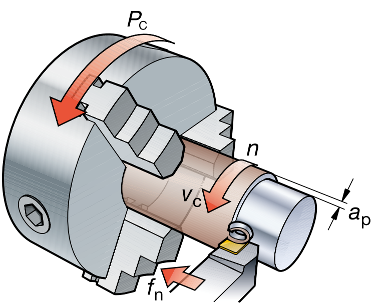
Maximum chip thickness h_{ex} reduces relative to the feed rate as the entering angle reduces (lead angle increases).

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

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

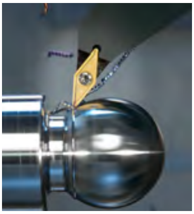
$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{33 \times 10^3} \text{ HP}$$

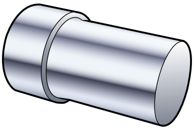
Selection procedure

Production planning process

1 **Component**

Dimension and type of operation





Workpiece material and quantity

2 **Machine**

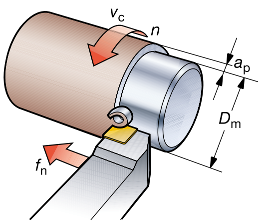
Machine parameters

3 **Choice of tool**

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

4 **How to apply**

Cutting data, tool path, etc.

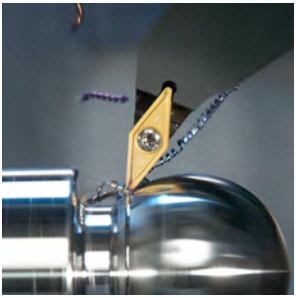


5 **Troubleshooting**

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.

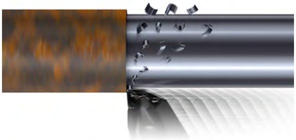
3. Choice of tools

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 compo-nents.

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

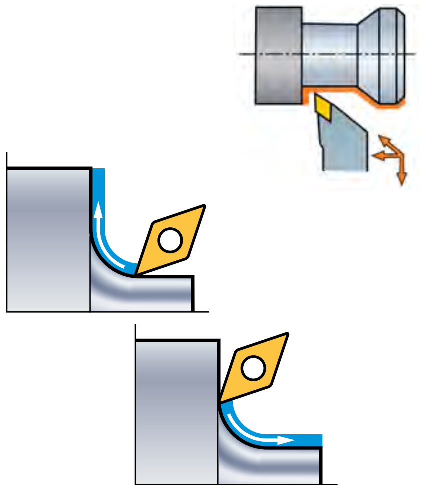
Advantages
- Increased cutting data in profiling
- Increased ability to hold tolerance.

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



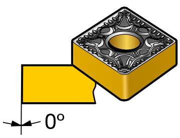
The tool path has a significant impact on the machining process.

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

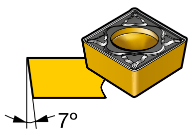
In practice, the tool holder, insert geometry, grade, workpiece material and tool path influences the cycle time and productivity considerably.

5. Troubleshooting

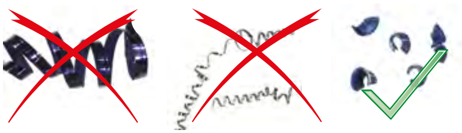
Some areas to consider

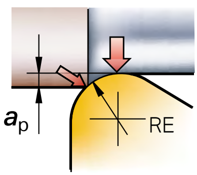


Negative style



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).

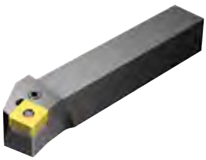
Insert wear
- Make sure that the flank wear does not exceed the general recommendation of 0.5 mm (.020 inch).

External Turning - negative inserts

1. Longitudinal turning
2. Profiling
3. Facing

Overview of tool holders

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



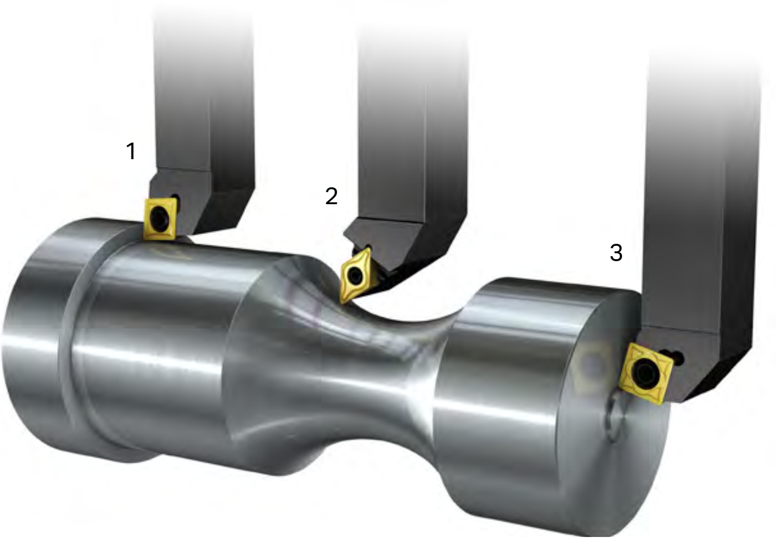
- Negative insert
- Lever clamping system
- Modular/shank tools.

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

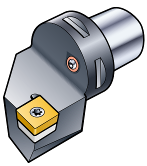
- Negative inserts
- Lever clamping system
- Precision coolant
- Modular/shank tools.

External Turning - positive inserts

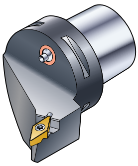
1. Longitudinal turning
2. Profiling
3. Facing



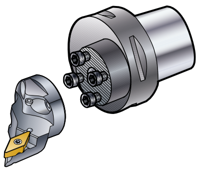
Overview of tool holders



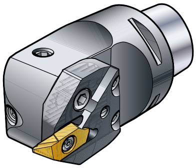
- Positive insert
- Screw clamping system
- Modular/shank tools
- Precision coolant.



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

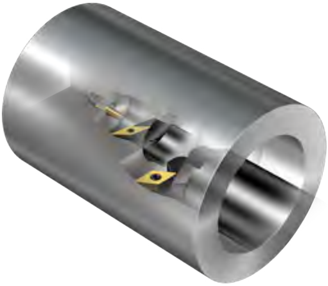


- Negative/positive insert
- All clamping systems
- Cutting heads
- Modular/shank tools.

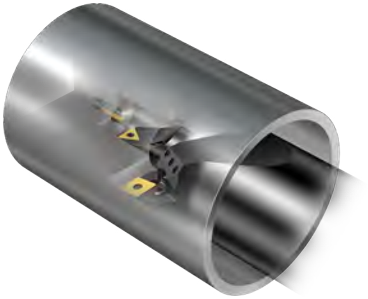


- Positive insert
- Screw clamping system
- Modular/shank tools.

Internal turning, negative/positive inserts

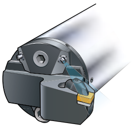


Positive inserts

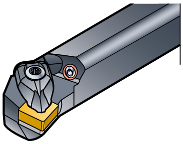


Negative inserts

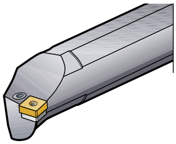
Overview of internal tool holders



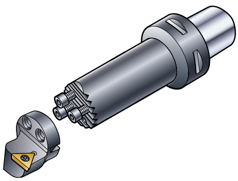
- Negative/positive inserts
- Dampened boring bars
- Boring bars



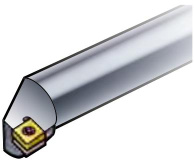
- Negative insert
- Rigid clamping system
- Modular/boring bars.



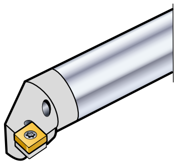
- Negative insert
- Lever clamping system
- Modular/boring bars.



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

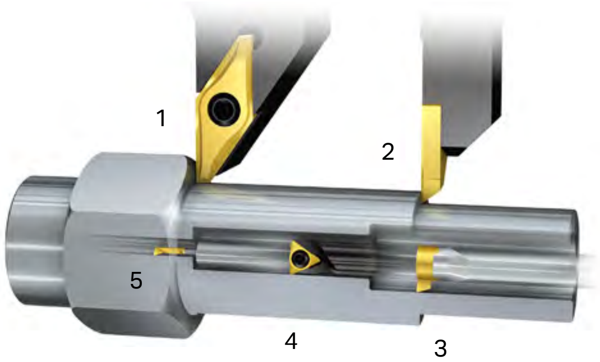


- Positive insert
- Screw clamping system
- Cutting heads
- Modular/boring bars.
- Precision coolant.



- Dampened boring bars
- Boring bars.

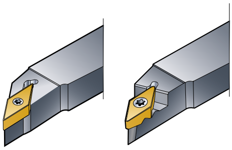
Tools for small part machining



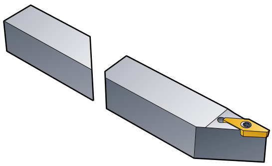
1. External turning
2. External turning (Sliding head machines)
3. Internal turning (Exchangeable inserts)
4. Internal turning
5. Internal turning (Carbide rods)

Overview of tool holders

External tools



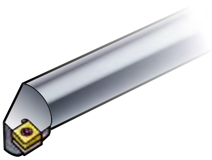
- Positive insert
- Screw clamping system
- Shank tools
- Precision coolant.



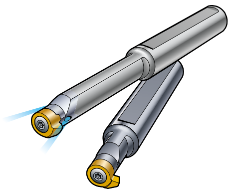
- Quick change tools
- Positive insert
- Screw clamping system.

- Positive insert
- Screw clamping system.

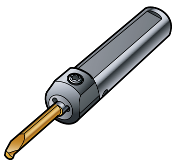
Internal tools



- Positive insert
- Screw clamping system
- Precision coolant.



- Positive insert
- Screw clamping system.

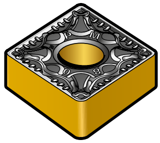


- Positive insert
- Carbide rods
- Machine adapted bars.

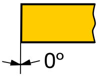
Overview of insert clamping systems

Clamping of negative basic-shape inserts

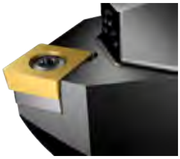
Rigid clamping system



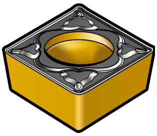
Lever clamping system

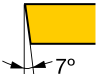


Clamping of positive basic-shape inserts

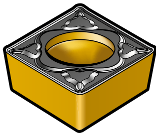


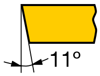
Screw clamping system





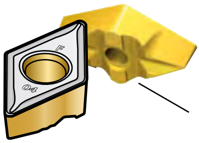
Screw clamping system



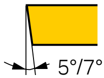


Clamping of positive iLock™ inserts

Screw clamping system

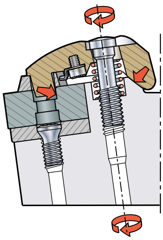


iLock™



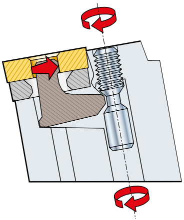
Modern insert clamping for turning tools

Rigid clamping



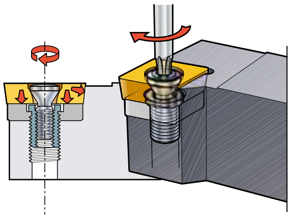
- Negative inserts
- Excellent clamping
- Easy indexing.

Lever clamping



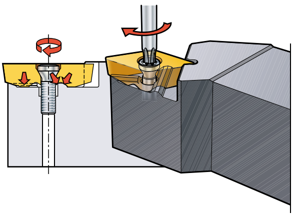
- Negative inserts
- Free chip flow
- Easy indexing.

Screw clamping

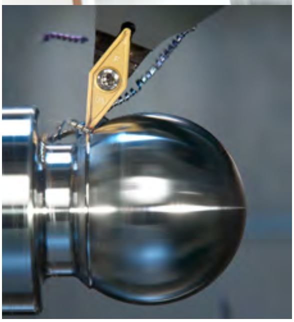


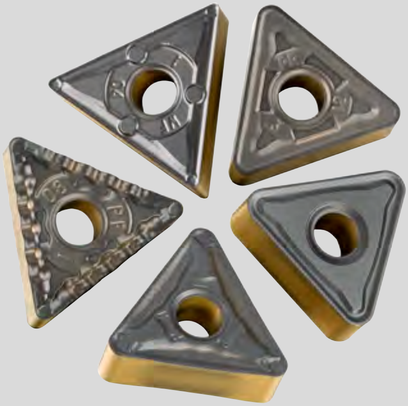
- Positive inserts
- Secure clamping of the insert
- Free chip flow.

Screw clamping system, iLock™



- Positive inserts
- Very secure clamping
- High accuracy.

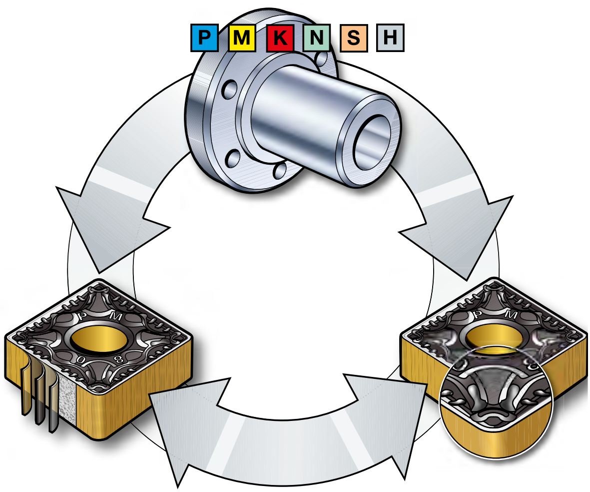




Choice of inserts

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.

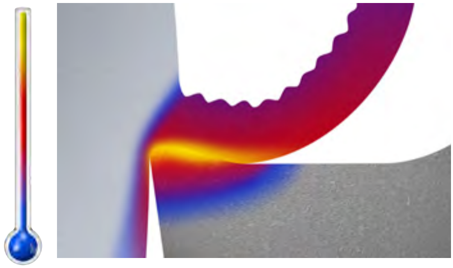
Cutting forces

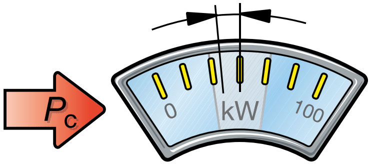
Another expression of the differences in the six material groups is through the force (F_T) needed to shear off a specific chip cross-section in certain conditions.

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).

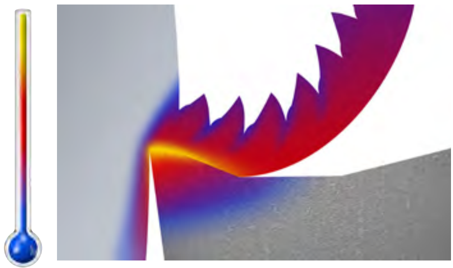
P Steel

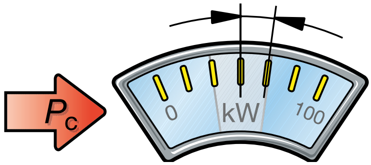




- P materials have a k_{c1} variation of: 1500-3100 N/mm^2 (217,500-449,500 lbs/inch^2)

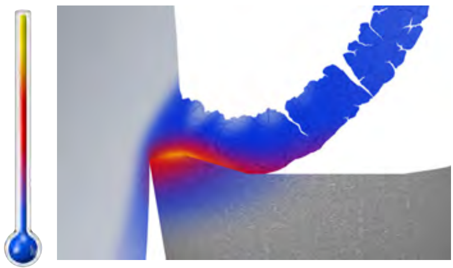
M Stainless steel

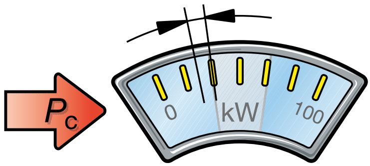




- M materials have *a* k_{c1} variation of: 1800-2850 N/mm^2 (261,000-413,250 lbs/inch^2)

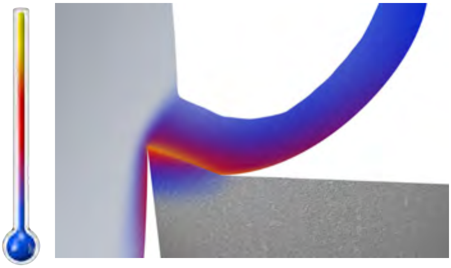
K Cast iron

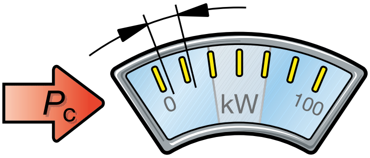




- K materials have a k_{c1} variation of: 790-1350 N/mm^2. (114,550-195,750 lbs/inch^2)

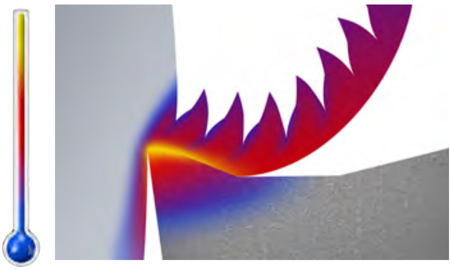
N Aluminum

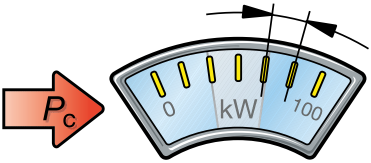




- N materials have a k_{c1} variation of:
  350-1350 N/mm²
  (50,750-195,750 lbs/inch²)

S Heat resistant super alloys

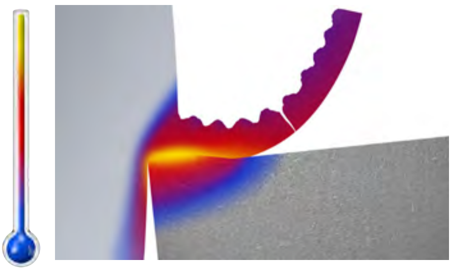


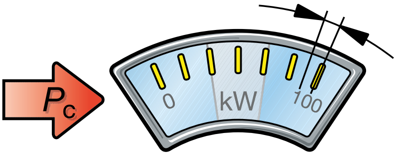


- S materials have a k_{c1} variation of:
  2400-3100 N/mm²
  (348,000-449,500 lbs/inch²) for HRSA

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

H Hardened steel

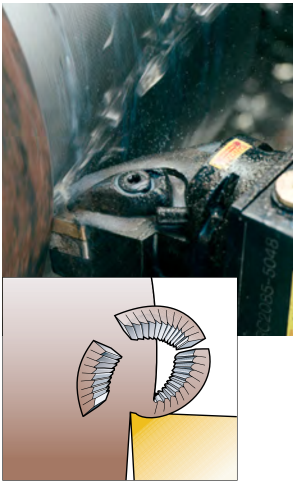
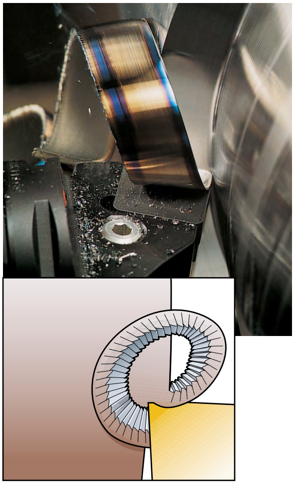
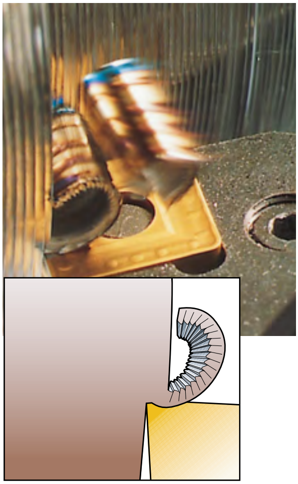




- H materials have a k_{c1} variation of:
  2550 – 4870 N/mm²
  (369,750-706,150 lbs/inch²)

Chip formation

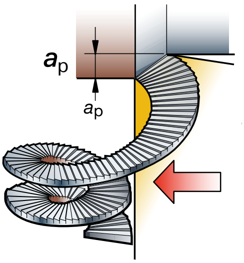
There are three patterns for a chip to break after it has been cut.

Self-breaking	Against the tool	Against the workpiece
		
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.	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.	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.

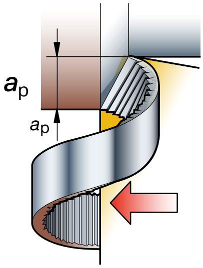
Chip formation varies with different parameters

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

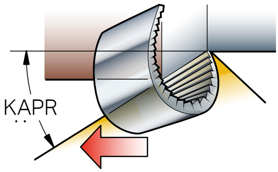
Self-breaking



Against the tool



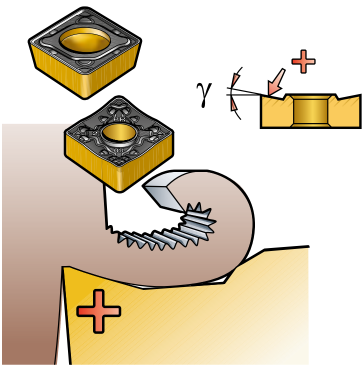
Against the workpiece



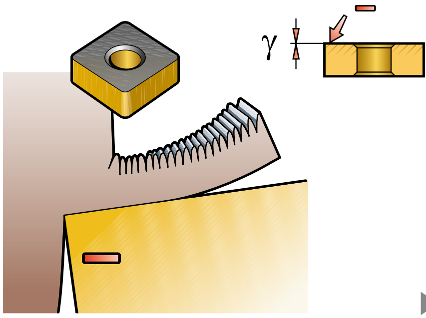
Insert rake angle

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

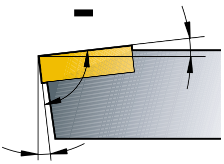


Insert rake angle

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.

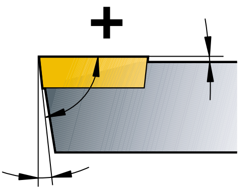
Negative style



- 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.

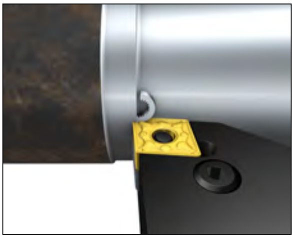
Positive style



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

Insert geometries

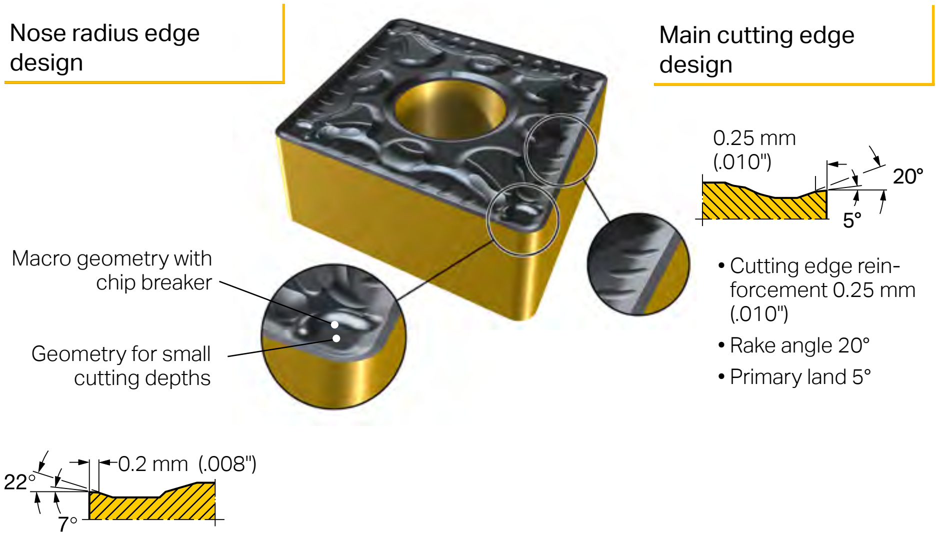
Metal cutting is very much the science of removing chips from the workpiece material in the right way. Chips have to be shaped and broken off into lengths that are manageable in the machine.



- 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.

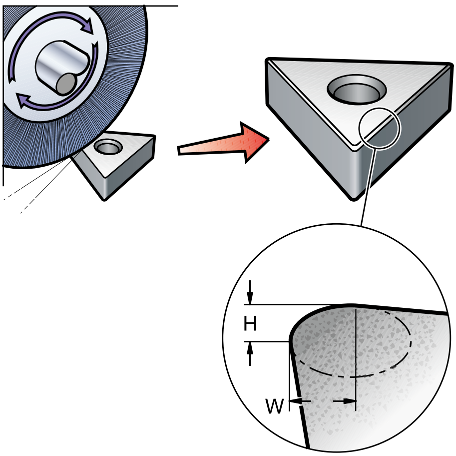
The design of a modern insert

Definitions of terms and geometry design



The reinforcement of the cutting edge

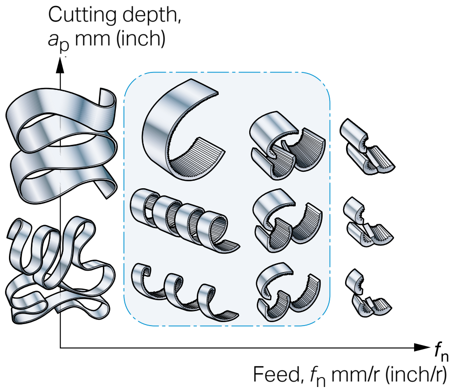
The Edge Roundness (ER) treatment gives the cutting edge the final micro-geometry.



- 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

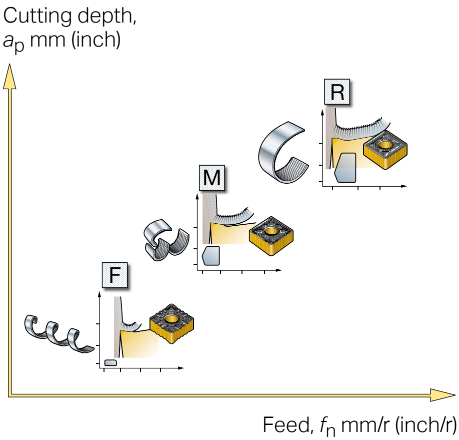
A chip breaking diagram for an insert geometry is defined by acceptable chip breaking for feed and depth of cut.



- 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

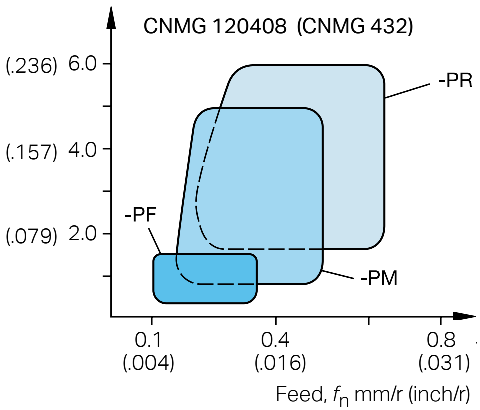
- Most applications – general purpose
- Medium operations to light roughing
- Wide range of cutting depth and feed rate combinations.

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.

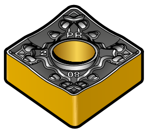
Medium – M
Medium operations to light roughing. Wide range of depth of cut and feed rate combinations.

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

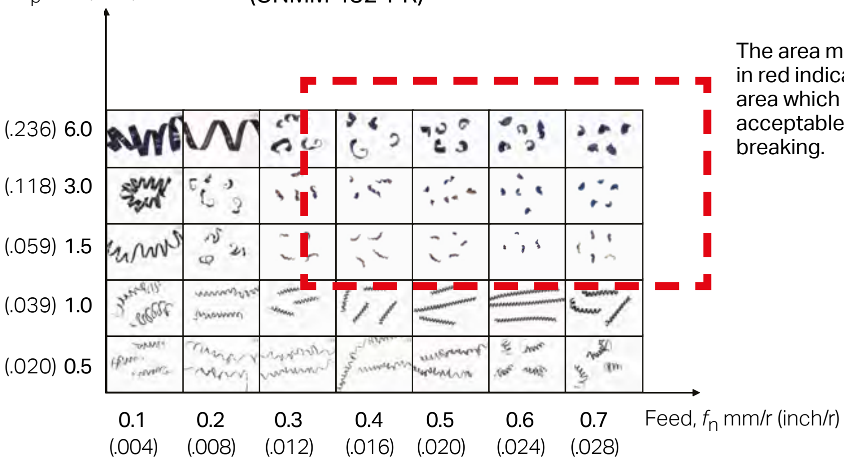


CNMM 120412-PR
(CNMM 432-PR)

Chip breaking area:

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

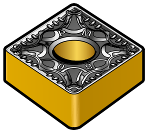
a_p = .197 (.039 - .295) inch
f_n = .020 (.010 - .028) inch/r



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

Medium machining of steel CMC 02.1

P M



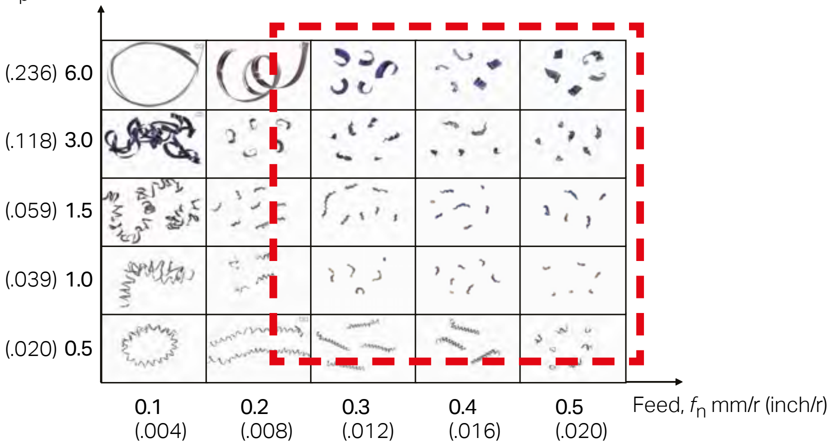
CNMG 120408-PM
(CNMG 432-PM)

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

Cutting depth,
a_p mm (inch)



Finishing of steel CMC 02.1

P F

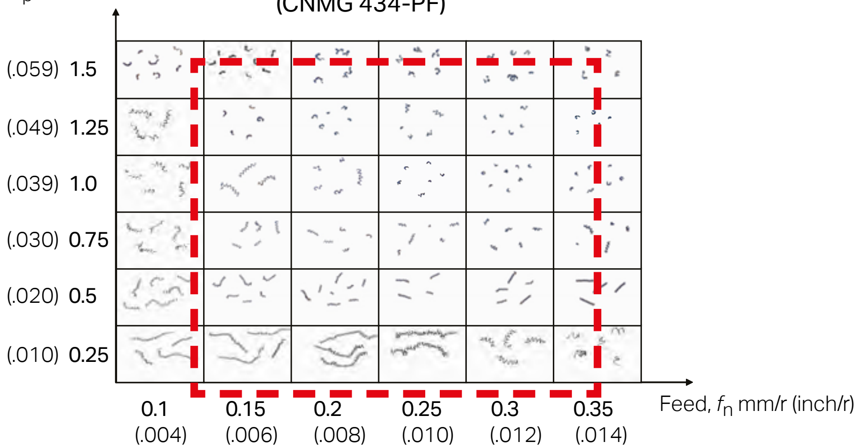
CNMG 120404-PF
(CNMG 434-PF)

Chip breaking area:

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

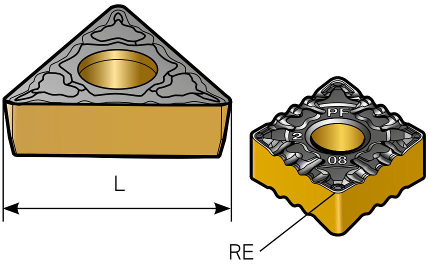
a_p = .016 (.010 - .059) inch
f_n = .006 (.003 - .012) inch/r

Cutting depth,
a_p mm (inch)



Selection of inserts

Considerations when selecting inserts



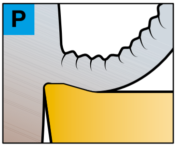
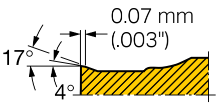
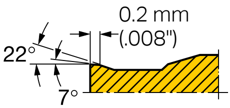
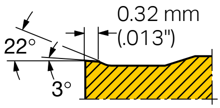
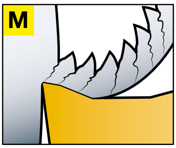
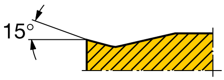
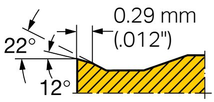
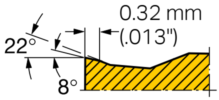
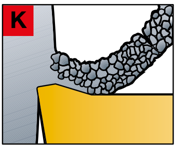
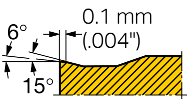
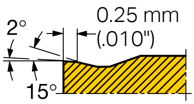
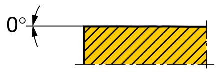
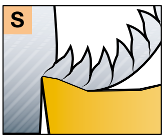
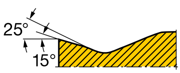
L= cutting edge length (insert size)
RE = nose radius

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
			
			
			
			

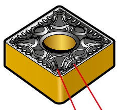
Geometry description

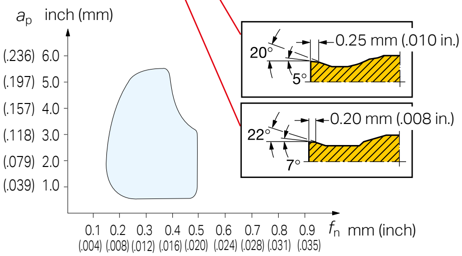
Every insert has a working area with optimized chip control.

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.

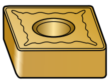
General recommendations: Combine with a wear resistant grade for best productivity.

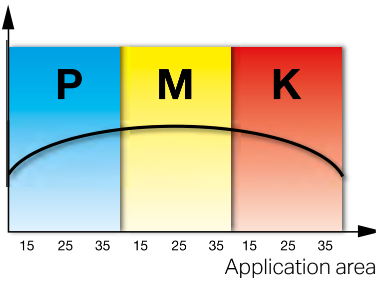
Possible optimization: geometry WMX.

From universal to optimized turning inserts

Universal inserts

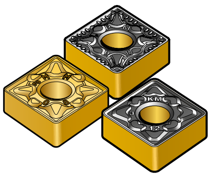
- Universal geometry
- Optimizing with grades
- Performance compromised.

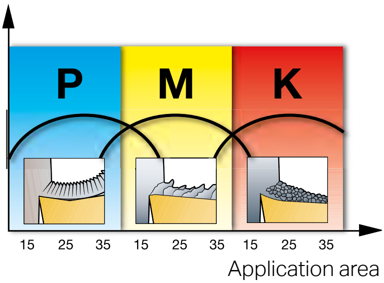




Optimized inserts

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



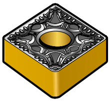


Inserts for general turning

The choice of different insert concepts

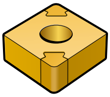
Negative, double/single-sided inserts

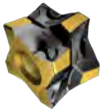
Plain inserts


Double sided


Single sided

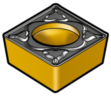

Without hole

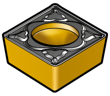

With hole


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

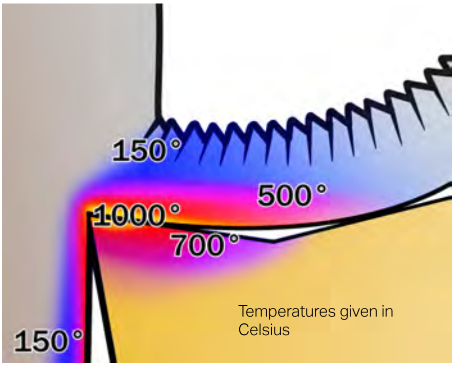

Concept iLock™ clamping


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

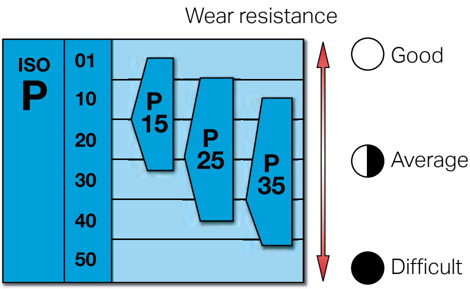
The most common cutting tool materials are divided into the following main groups:

- **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.

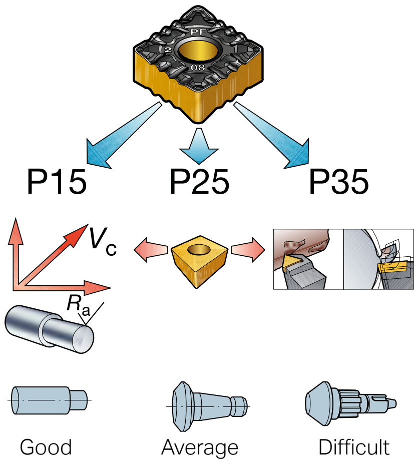
How to select insert geometry and grade

Select the geometry and grade according to the application.

Build up of a grade chart

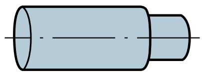


Machining conditions

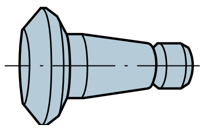


Machining conditions

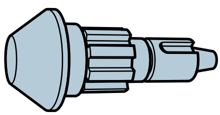
Good conditions
- Continuous cuts
- High speeds
- Pre-machined workpiece
- Excellent component clamping
- Small overhangs.



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.



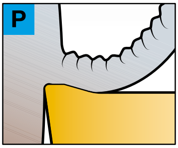
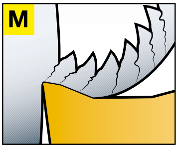
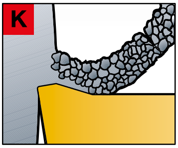
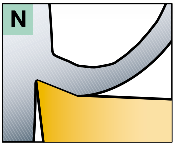
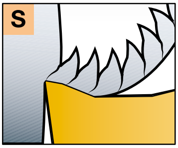
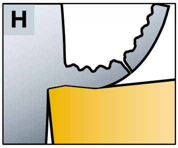
A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

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
		

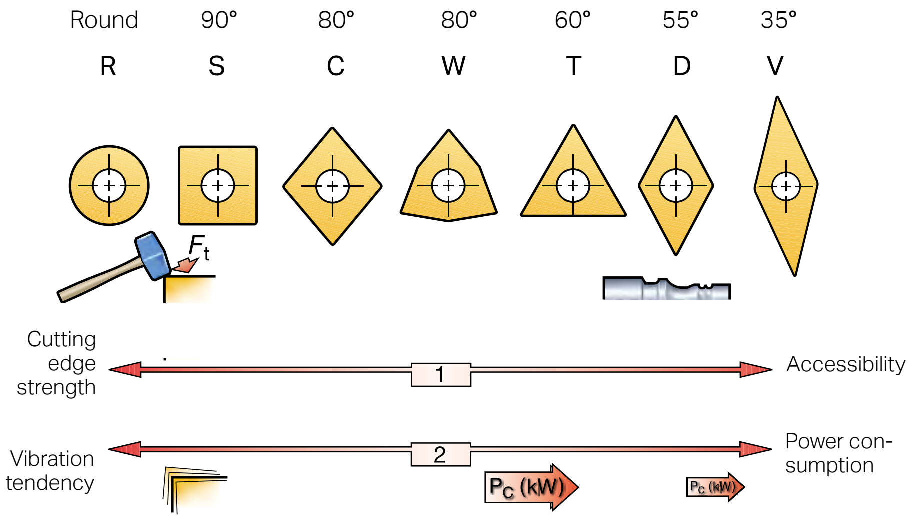
Selection of the insert shape

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.

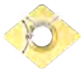
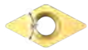
Factors affecting choice of insert shape

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

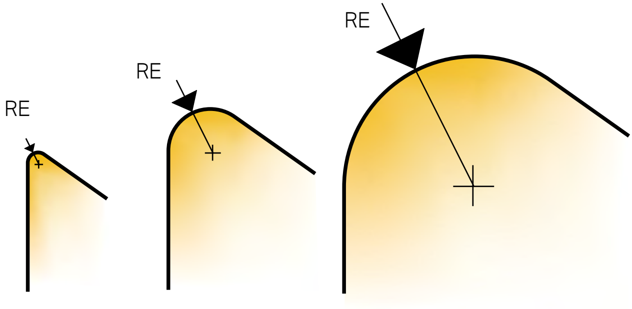
Number of cutting edges

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

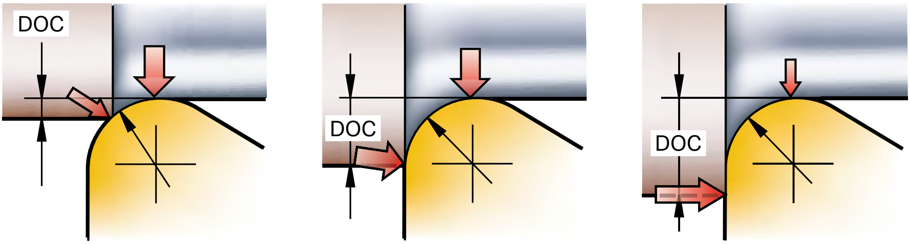
- Heavy feed rates
- Large depths of cut
- Strong edge security
- Increased radial pressures.

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.

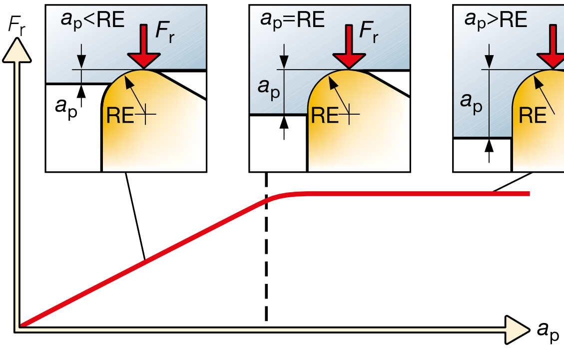


- The relationship between nose radius and DOC (depth of cut) affects vibration tendencies. It is often an advantage to choose a nose radius which is smaller than the DOC.

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.

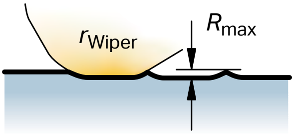
However with a round insert, radial pressure will never stabilize because the theoretical nose radius is half the insert diameter (IC).



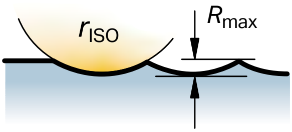
High feed turning with wiper inserts

Wiper – General information

Wiper insert



Conventional insert



Why use a wiper

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

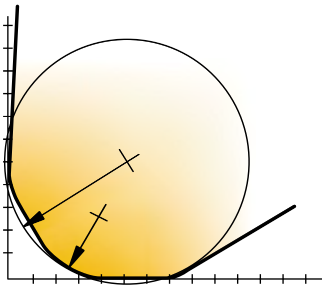
When to use wipers

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

Limitations

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

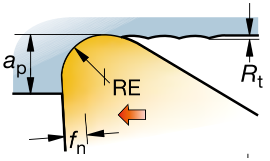
Wiper – Technical solution



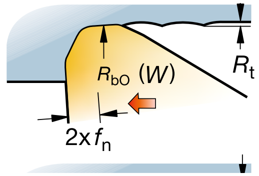
- One wiper cutting edge is based on 3-9 radii.
- Contact surface between insert and component is longer with wipers.
- Longer contact surface makes a better surface finish.
- Longer contact surface increases cutting forces which makes a wiper insert more sensitive to vibration when machining unstable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

A conventional nose radius compared with a wiper nose radius.

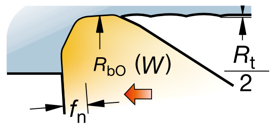
Wiper – Surface finish



Traditional insert



Wiper insert
Twice the feed, same R_a



Wiper insert
Same feed, half R_a

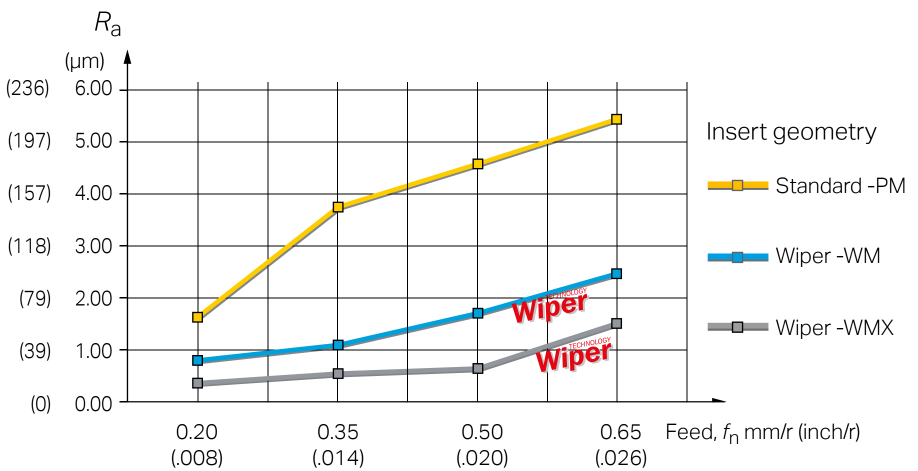
Rule of thumb

- 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.

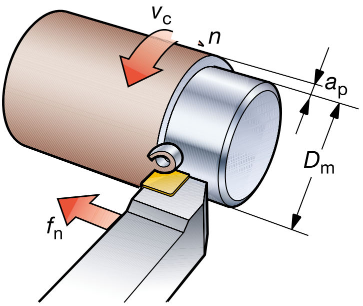
R_t = Maximum value peak-to-valley height

R_a = Arithmetic average height of the profile

Achieved surface – traditional ISO inserts and wipers

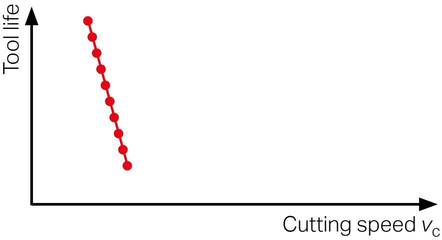


Cutting data parameters affect tool life



Use the potential of:

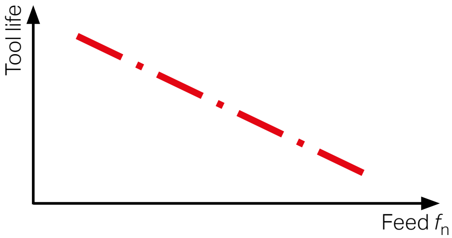
- a_p – to reduce number of cuts
- f_n – for shorter cutting time
- v_c – for best tool life



Cutting speed

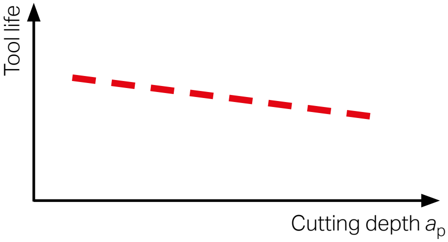
v_c – large effect on tool life.

Adjust v_c for best economy



Feed

f_n – less effect on tool life than v_c

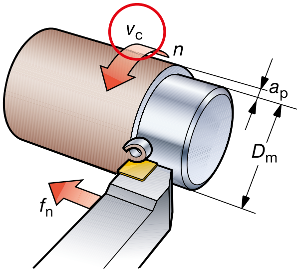


Cutting depth

a_p – little effect on tool life

Effects of cutting speed

The single largest factor determining tool life



Too high

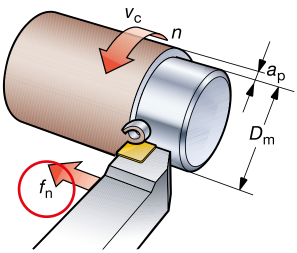
- Rapid flank wear
- Poor finish
- Rapid cratering
- Plastic deformation.

Too low

- Built-up edge
- Uneconomical.

Effects of feed rate

The single largest factor determining productivity



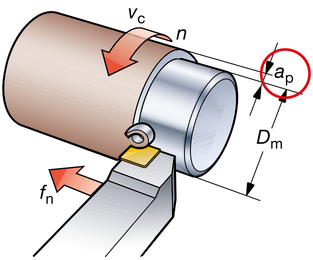
Too high

- 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

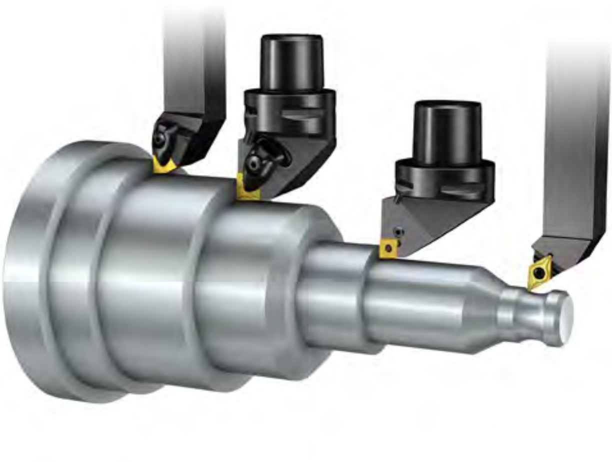
- High power consumption
- Insert breakage
- Increased cutting forces.

Too small

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

External turning

Tool selection and how to apply

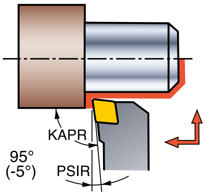


General guidelines

- 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

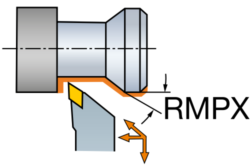
Longitudinal turning/facing



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°).

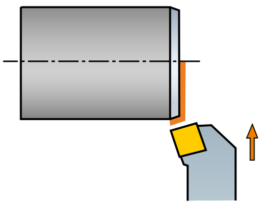
Profiling



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.

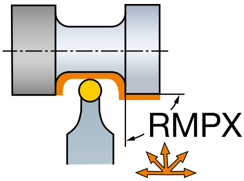
Facing



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.

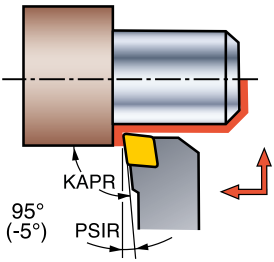
Pocketing



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.

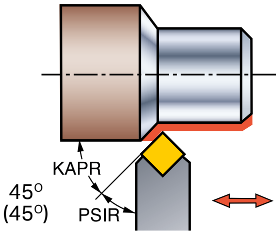
Large entering angle (small lead angle)



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)



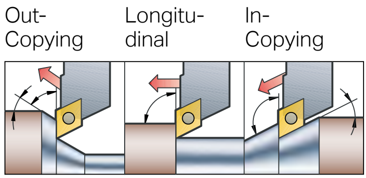
Features / Benefits

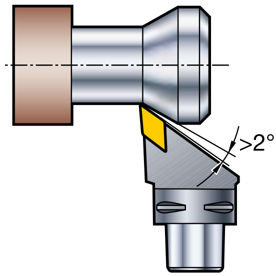
- Produces a thinner chip
  - Increased productivity
- Reduced notch wear
- Cannot turn against a shoulder.

The entering and copying angle

Important consideration in profile turning

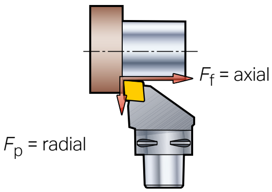
- In profile turning, the cut can vary with regard to cutting depth, chip thickness and speed.
- The largest suitable nose angle on the insert should be selected for strength and cost efficiency, but the insert nose angle also has to be considered in relation to accessibility for proper clearance between 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.





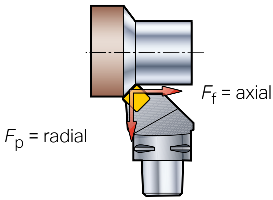
Axial and radial cutting forces

Large entering angle (small lead angle)



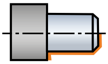
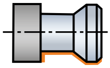
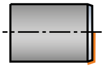
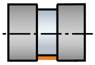
- Forces directed toward the chuck. Less tendency for vibration.
- Higher cutting forces especially at entrance and exit of cut.

Small entering angle (large lead angle)

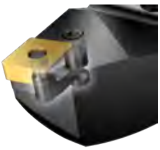
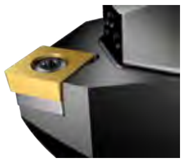


- Forces are directed both axially and radially.
- Reduced load on the cutting edge.
- Forces are directed both axially and radially - Vibration tendencies.

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

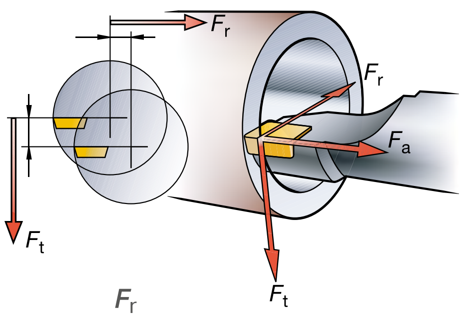
- Chip size
- Chip control
- Techniques
- Coolant.

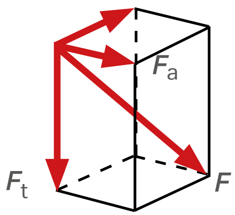
Tool requirements

- 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

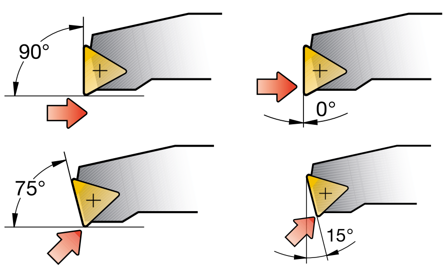
- Alters cutting depth and chip thickness
- Gives out of tolerance dimension and risk of vibration.

Feed force, F_a

- Directed along the feed of the tool.

Selecting entering (lead) angles

Entering (lead) angle and cutting forces

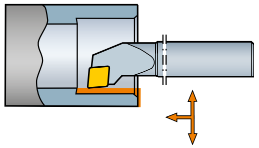


- Select an entering angle close to 90° (lead angle close to 0°).
- If possible, do not choose an entering angle less than 75° (lead angle not more than 15°), since this leads to a dramatic increase of the radial cutting force F_r.

  - Less force in radial direction = less deflection.

Four main application areas

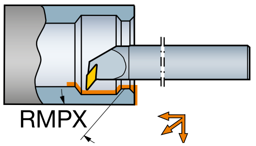
Longitudinal turning/facing



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.

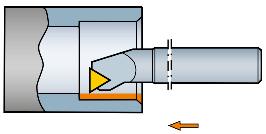
Profiling



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-copying angle between 22–27°, are commonly used.
- D-style 55° and V-style 35° inserts are frequently used.

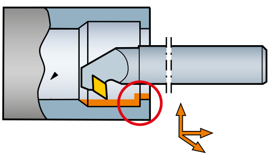
Longitudinal turning



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

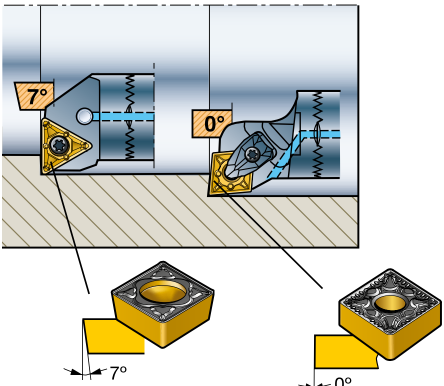


Back boring is a boring operation with reverse feed.

- 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

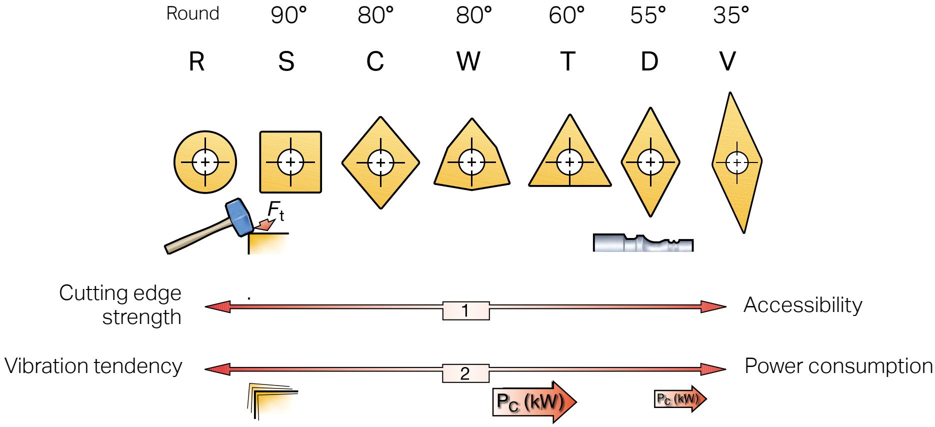
Insert shape ++ = Recommended + = Alternative	Longitudinal turning	Profiling	Facing
Rhombic 80°	+		++
Rhombic 55°	+	++	+
Round	+		+
Square	+		
Triangular	++		+
Trigon 80°	+		+
Rhombic 35°		+	

Insert point angle

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

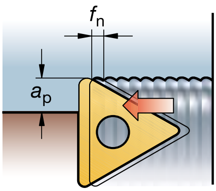
Small point angle:
- Increases accessibility
- Decreases vibration
- Decreases cutting forces

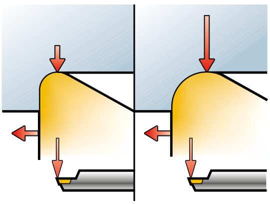
Use the smallest angle giving acceptable strength and economy.



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.

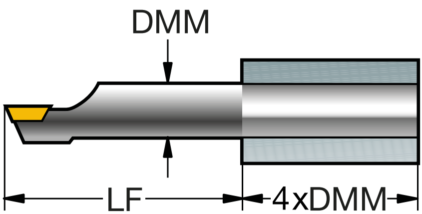
- The relationship between RE (nose radius) and a_p (depth of cut) affects vibration tendencies.
- Less force in radial direction = less deflection.

Rule of thumb!

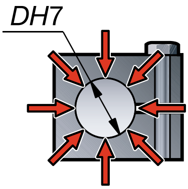
Choose a nose radius which is somewhat less than the cutting depth.

Clamping the boring bar

Critical stability factors for optimized performance

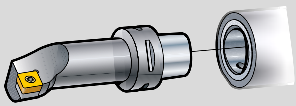


- Maximum contact between tool and tool holder (design, dimensional tolerance).
- Clamping length 3 to 4 times bar diameter (to balance cutting forces).
- Holder strength and stability.



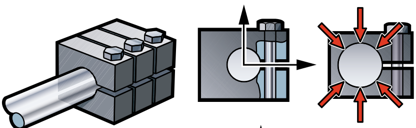
Tool requirements for clamping

Maximum contact between tool and tool holder

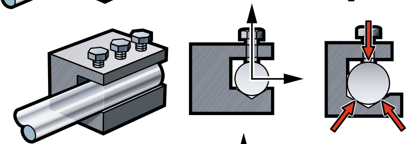


Coromant Capto® coupling

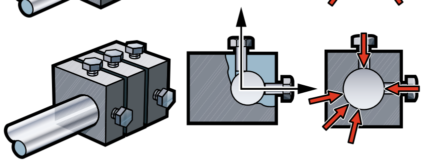
Recommended



Acceptable



Not recommended



Not recommended

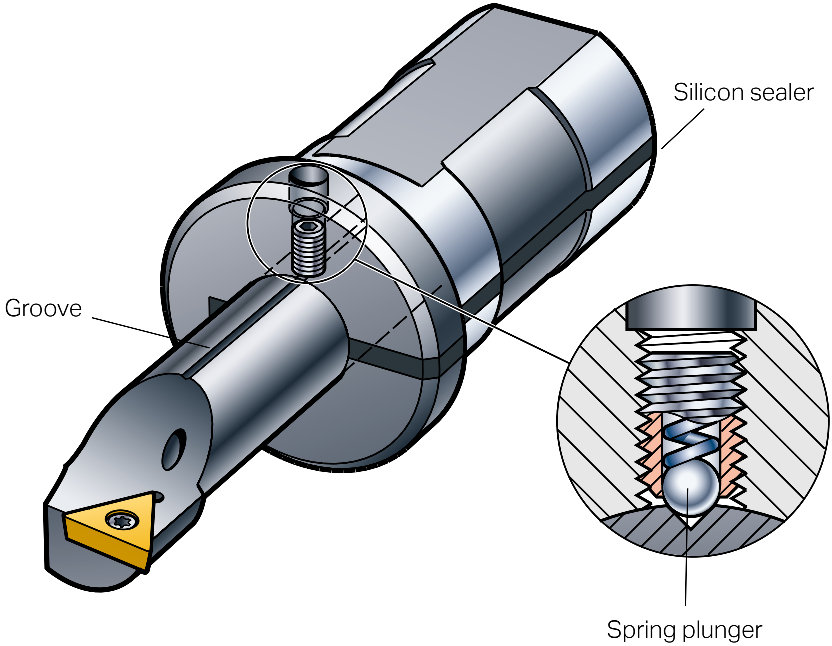
EasyFix sleeves

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

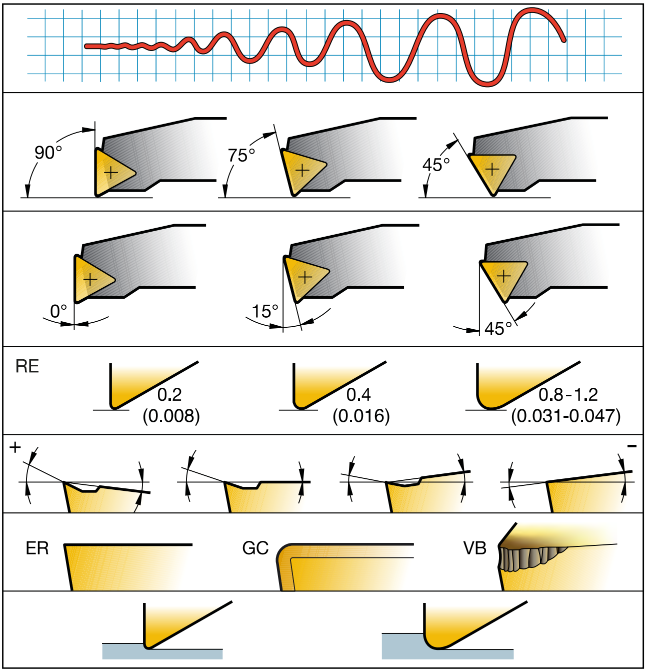
Lead angle

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

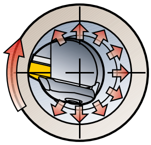
- Insert wear changes the clearance between the insert and the hole wall. This can affect the cutting action and lead to vibration.
- Inserts with thin coatings, or uncoated inserts, are to be preferred as they normally give lower cutting forces.

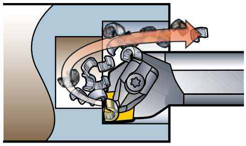
Depth of cut (DOC)

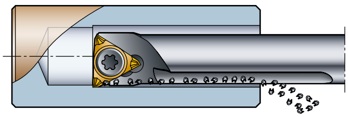
- Choose a corner radius which is somewhat smaller than the cutting depth.

Chip evacuation

Chip evacuation is a critical factor for successful boring

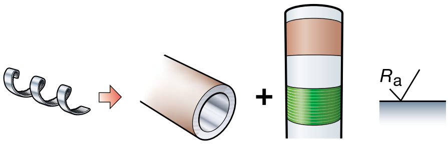






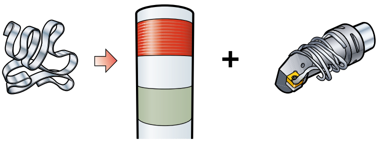
- Centrifugal force presses the chips to the inside wall of the bore.
- The chips can damage the inside of the bore.
  - Internal coolant can help with chip evacuation.
  - Boring upside down helps to keep chips away from the cutting edge.

Chip evacuation and chip control



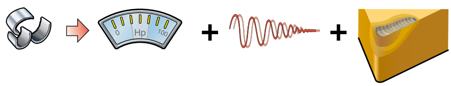
Short and spiral chips

- 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

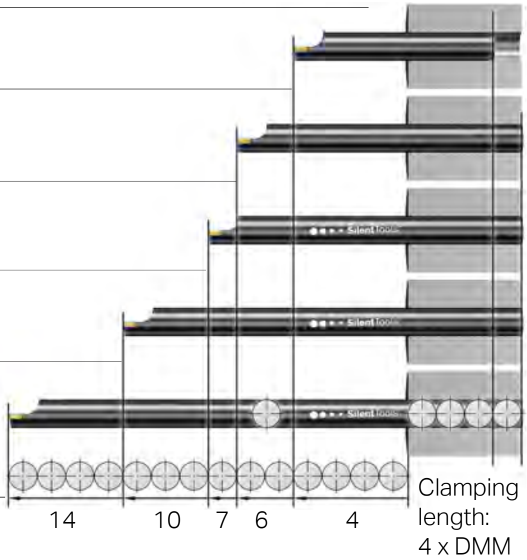
Carbide bar
– up to 6 x DMM).

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

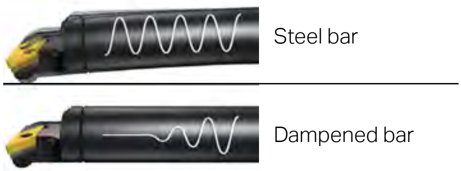


Eliminate vibrations

Internal machining with dampened boring bars

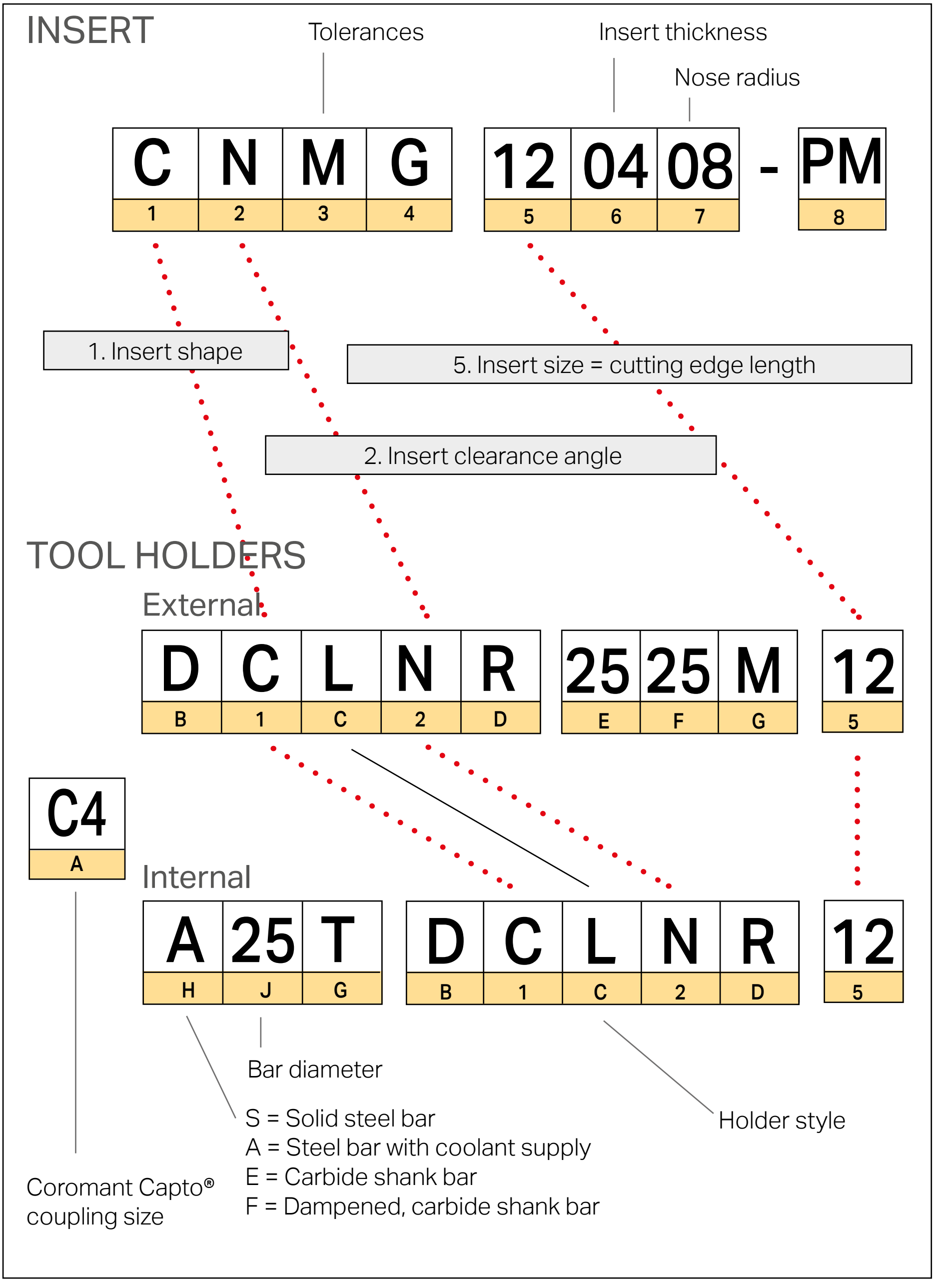
- Increase productivity in deep bores.
- Minimize vibration.
- Machining performance can be maintained or improved.
- Dampened boring bars are available in diameters from 10 mm (.394 inch).
- For max overhang 14 x DMM (carbide reinforced).

SilentTools®

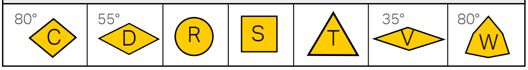


Code key for inserts and toolholders - METRIC

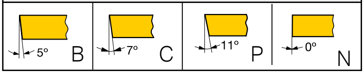
Extract from ISO 1832:1991



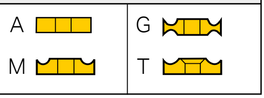
1. Insert shape



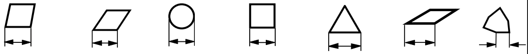
2. Insert clearance angle



4. Insert type

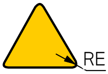


5. Insert size = Cutting edge length



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:

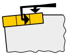
	T-MAX P	CoroTurn 107
Finishing	08	04
Medium	08	08
Roughing	12	08

8. Geometry — manufacturer's option

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

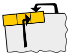
-PF = ISO P Finishing
-MR = ISO M Roughing

B. Clamping system



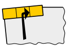
D

Rigid clamping (RC)



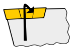
M

Top and hole clamping



P

Hole clamping

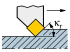


S

Screw clamping

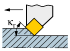
D. Hand of tool

R



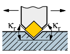
Right-hand style

L



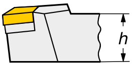
Left-hand style

N

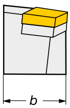


Neutral

E. Shank height

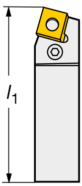


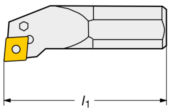
F. Shank width



G. Tool length

Tool length
= l_1 in mm

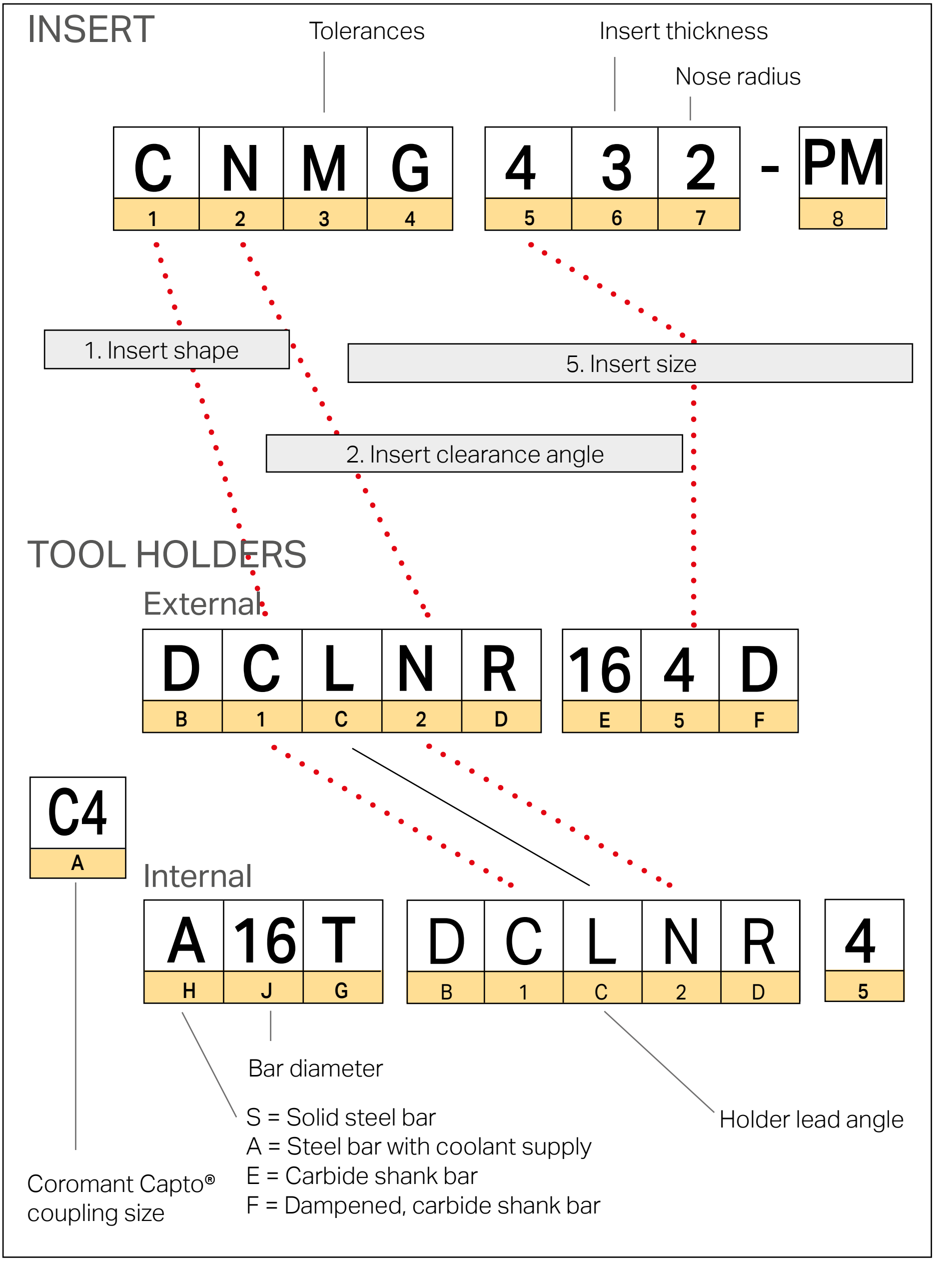




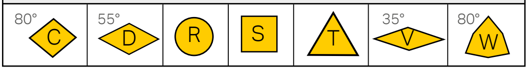
H = 100	S = 250
K = 125	T = 300
M = 150	U = 350
P = 170	V = 400
Q = 180	W = 450
R = 200	Y = 500

Code key for inserts and toolholders - INCH

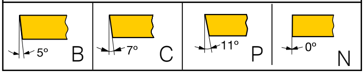
Extract from ANSI/ISO standards



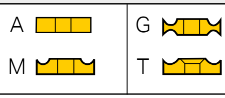
1. Insert shape



2. Insert clearance angle

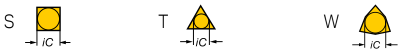


4. Insert type



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

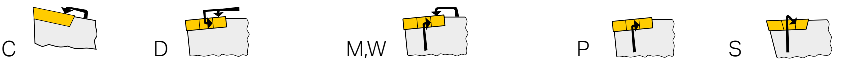
8. Geometry — manufacturer's option

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

-PF = ISO P Finishing

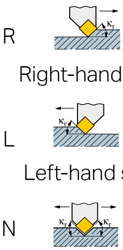
-MR = ISO M Roughing

B. Clamping system



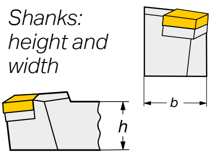
Top clamping | Rigid clamping (RC) | Top and hole clamping | Hole clamping | Screw clamping

D. Hand of tool

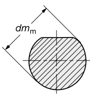


E. Shank or bar size

Shanks: height and width

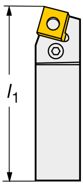


Bars:



G. Tool length

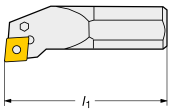
External, l_1 in inch



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

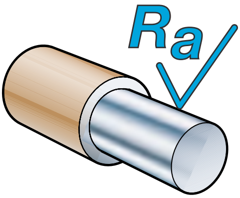
Troubleshooting

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.

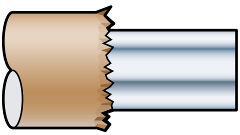
Surface finish

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.

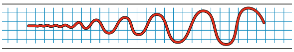
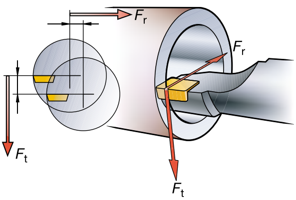


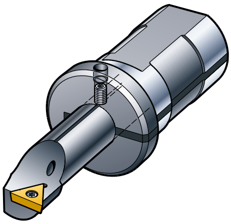
Burr formation

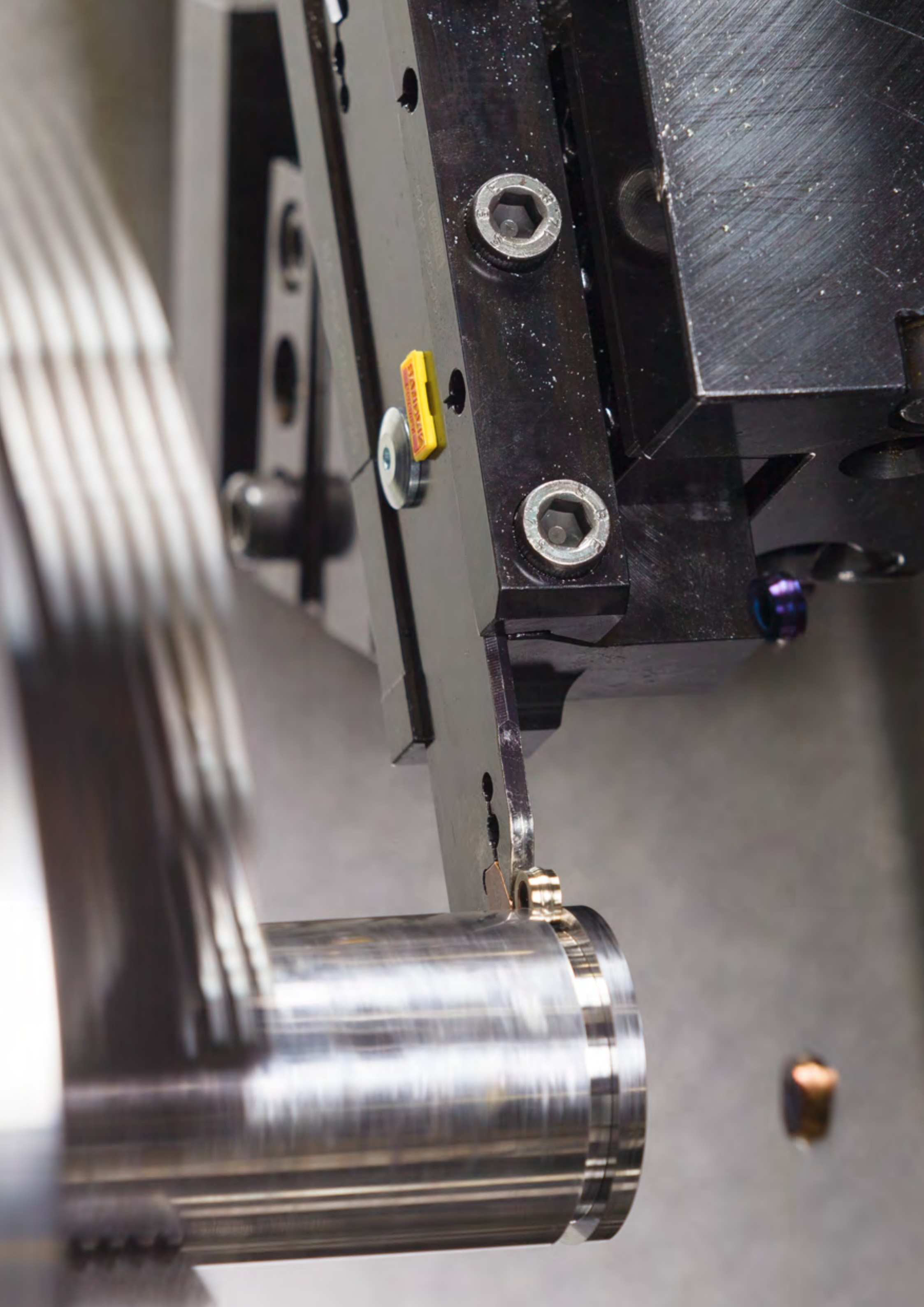
Problem	Cause	Solution
Burr formation at the end of the cut when the cutting edge is leaving the workpiece.	• The cutting edge is not sharp enough. • The feed is too low for the edge roundness.	• Use inserts with sharp edges: - PVD coated inserts. - ground inserts at small feed rates, < 0.1 mm/r (.004 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:	• Unsuitable entering/lead angle.	• Select as large as possible entering angle (KAPR = 90°) or as small as possible lead angle (PSIR = 0°).
 Vibrations or chatter marks which are caused by the tooling or the tool mounting. Typical for internal machining with boring bars.	• 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.

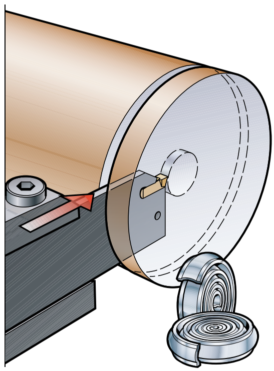
These tools can be used, to some extent, for general turning.

Parting & grooving theory

Parting off

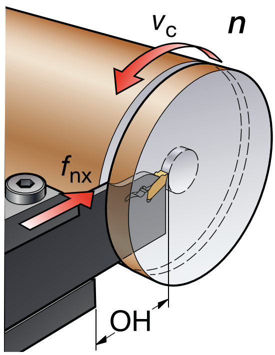
Chip evacuation is essential

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)

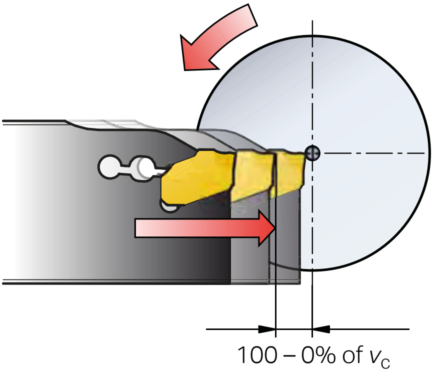
v_c = cutting speed m/min (ft/min)

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

OH = overhang recommended

Cutting speed value

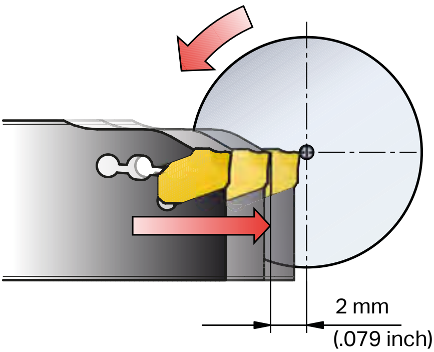
When parting off to center, the cutting speed will gradually be reduced to zero when the machine has reached its rpm limit.



- Cutting speed declines to zero at the center.

Feed reduction towards center

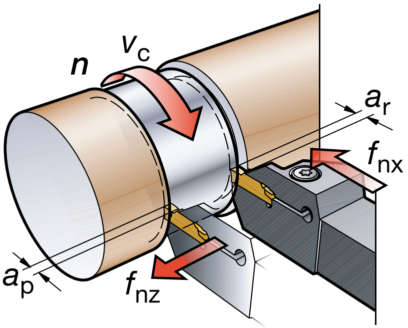
Cutting speed decreases toward the part center line, causing unbalance. Feed rate must be reduced to maintain cutting force balance during the part-off. The feed rate should be reduced to the minimum recommended or about 0.05 mm/rev (.002"/rev) at 2 mm (.079") before reaching centerline.



- 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)

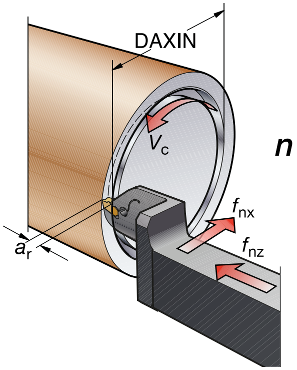
f_{nx} = radial cutting feed mm/r (inch/r)

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)

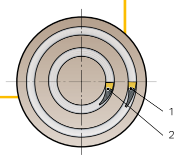
f_{nz} = axial cutting feed mm/r (inch/r)

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)

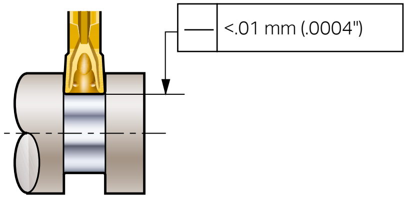


Tool selection procedure

Production planning process

1 Component

Dimension and quality of the groove or face



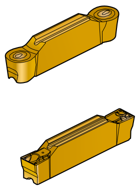
Workpiece material, chip evacuation

2 Machine

Machine parameters

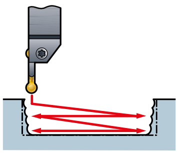
3 Choice of tool

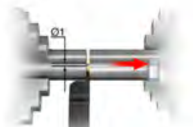
Type of tool:
- Spring-clamp
- Screw-clamp
- Insert type



4 How to apply

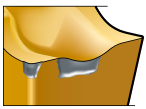
Cutting data, method, cutting fluid, etc

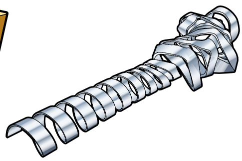




5 Troubleshooting

Remedies and solutions

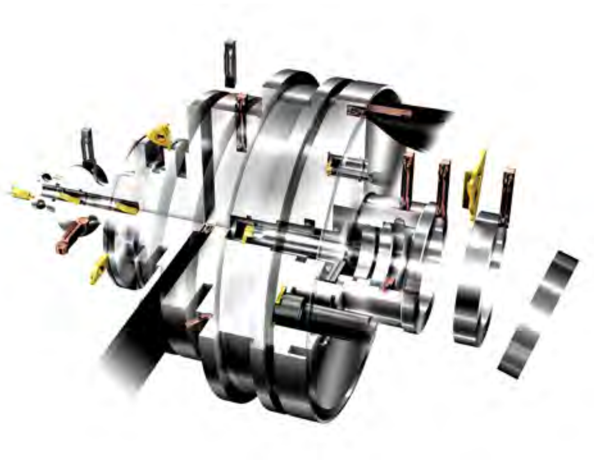




A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

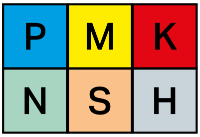
1. Component and the workpiece material

Parameters to be considered



Component

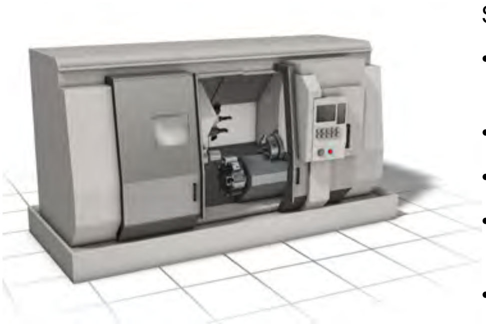
- Analyze the dimensions and quality demands of the groove or face to be machined
- Type of operation: parting, grooving
- Cutting depth
- Cutting width
- Corner radius.



Material

- Machinability
- Chip breaking
- Hardness
- Alloy elements.

2. Machine parameters



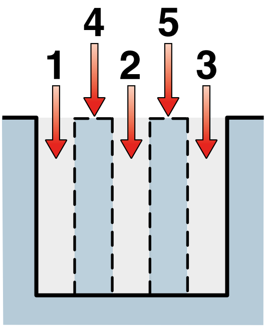
Some important machine considerations:

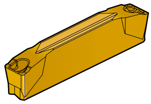
- Stability, power and torque especially for larger diameters
- Component clamping
- Turret interface
- Tool changing times/number of tools in turret
- Chip evacuation
- Cutting fluid and coolant.

3. Choice of tools

Example of different machining methods

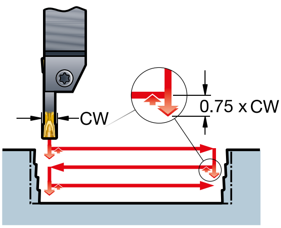
Multiple grooving

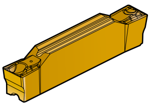




- Multiple grooving is the best method for rough grooving when the depth is bigger than the width.
- Make a “fork”. This will improve chip flow and increase tool life.

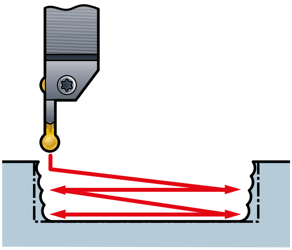
Plunge turning

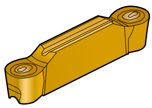




- Plunge turning is the best choice when machining steel and stainless steel and when the width of the groove is larger than the depth.
- Good chip control.

Ramping

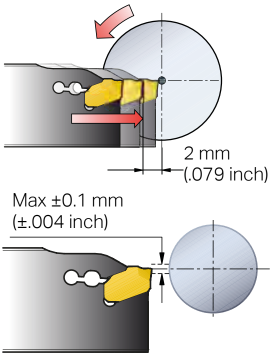




- Ramping avoids vibration and minimizes radial forces.
- Round inserts are the strongest inserts available.
- Double the number of cuts/passes.
- First choice in heat resistant super alloys (HRSA). Reduces notch wear.

4. How to apply

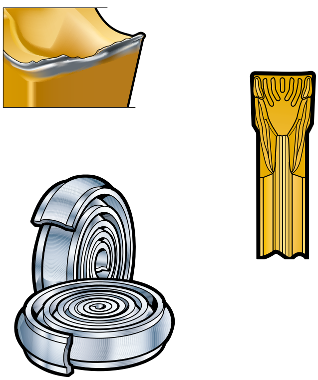
Application considerations



- 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.

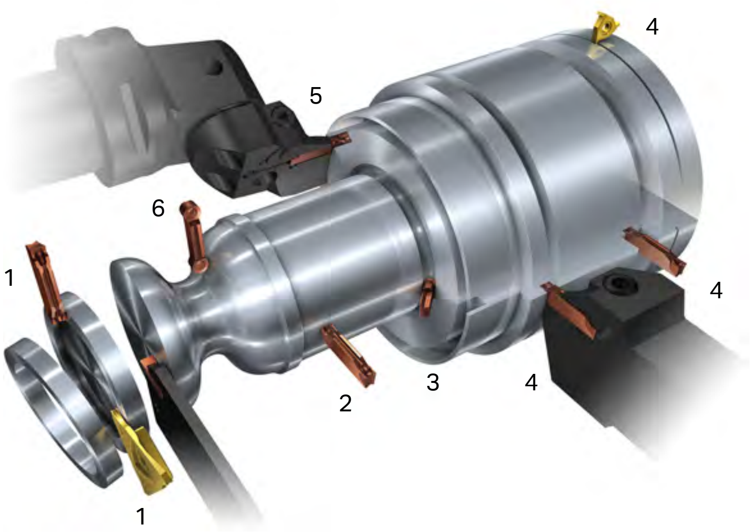
To improve chip formation & tool wear

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

System overview

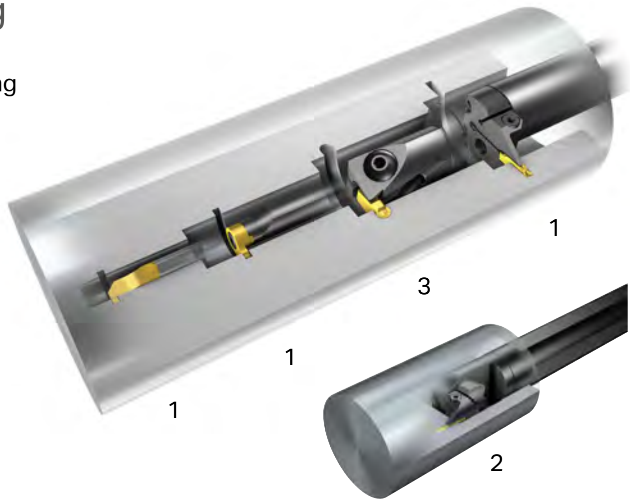
External parting and grooving

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

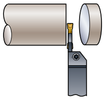
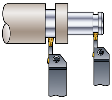
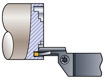
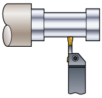
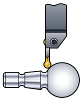
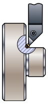
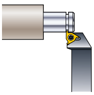


Internal grooving

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



Different systems

Application \ Insert type		 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

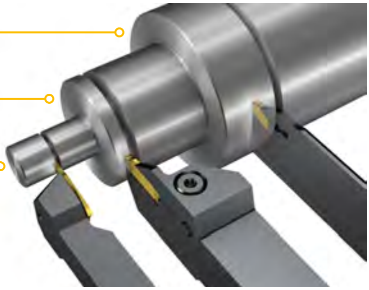
Different systems

External parting – diameter ranges

Deep parting – Ø < 160 mm (6.299")

Medium parting – Ø < 40 mm (1.575")

Shallow parting – Ø < 12 mm (.472")



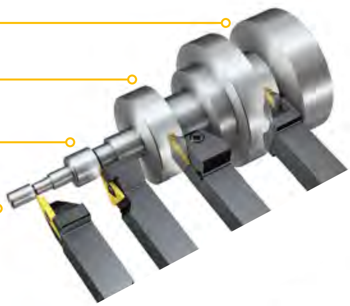
Grooving – depth ranges

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")

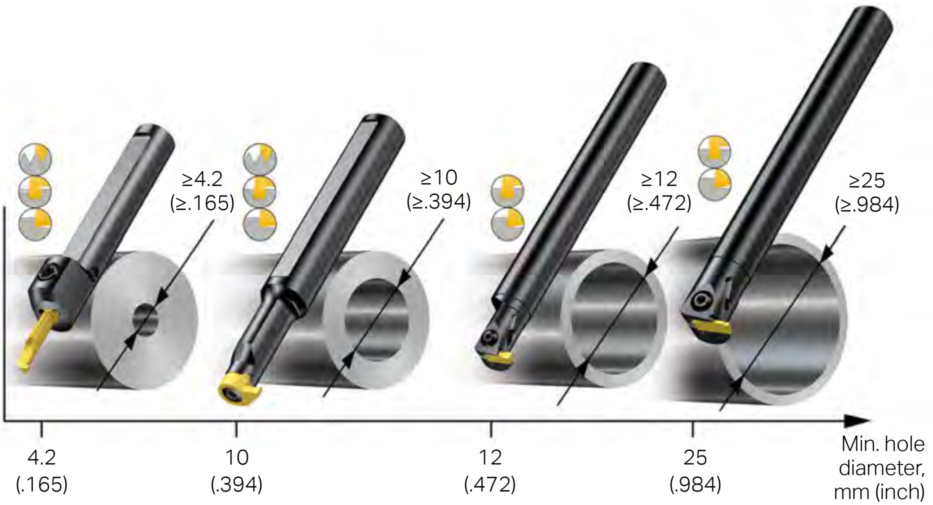
Small diameters > 6 mm (.236")

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

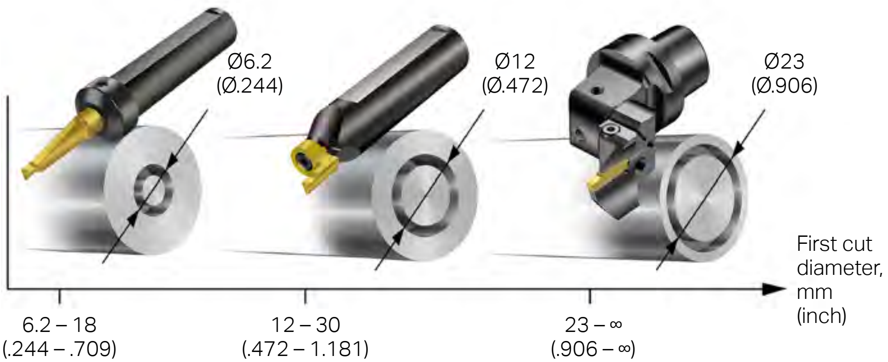
Internal parting and grooving

Different systems

Internal grooving – min hole diameter

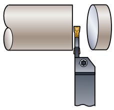
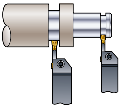
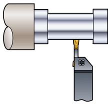
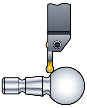
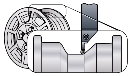
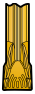
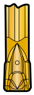
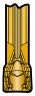
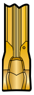
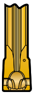
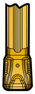
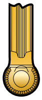
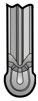
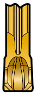
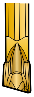
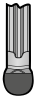
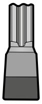
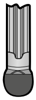


Face grooving – hole diameter range



Inserts

Geometry overview

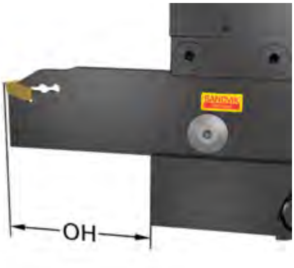
Application / Machining condition	 Parting (Cut off)	 Grooving	 Turning	  Profiling
Finishing	CF 	GF 	TF 	
Medium	CM 	GM 	TM 	RM  AM 
Roughing	CR 			
Optimizer				RO 
	CS 			RS 
		GE 		RE 

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

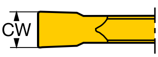


- Overhang (OH) should be as small as possible
- Largest seat size should be used

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.

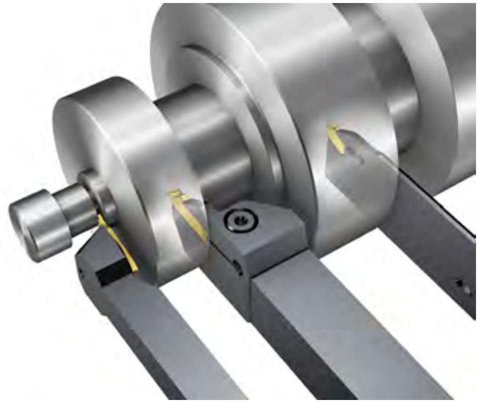


Inserts:

- 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

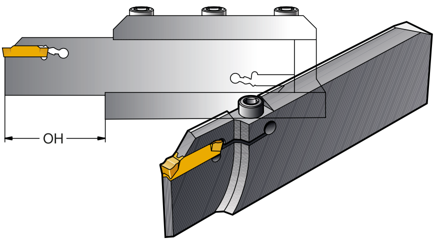
- First choice for medium parting are holders with 2-edge inserts.

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

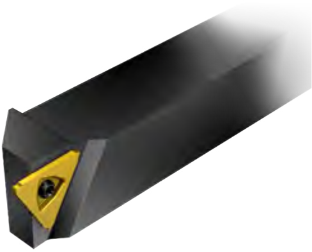
- Quicker insert change
- Cut off larger diameter
- Adjustability
- Deep grooving
- Double ended
- Radial feed only
- Precision coolant.

Screw-clamp and spring-lock design holders

Features/Benefits

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

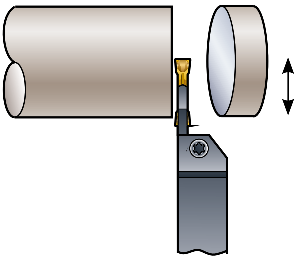
Screw-clamp design holders for 3-edge inserts



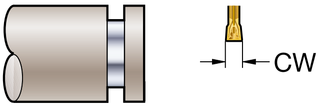
Features/Benefits

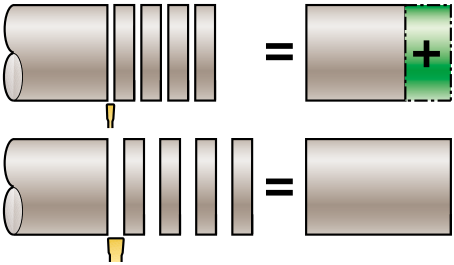
- Extremely small insert widths
  - grooving down to 0.5 mm (.020")
  - parting down to 1 mm (.039").
- Cutting depths up to 6 mm (.236").
- One holder for all insert widths.
- Very tight insert indexing tolerance.
- The productivity choice, 3 cutting edges.

Parting-off bars



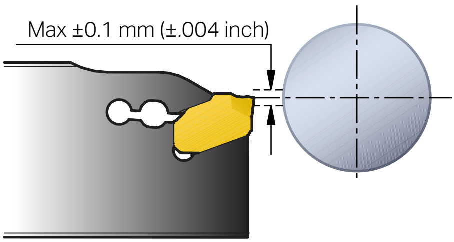
Use as narrow an insert as possible:
- To save material
- Minimize cutting force
- Minimize environmental pollution.





Material savings

Positioning of the tool

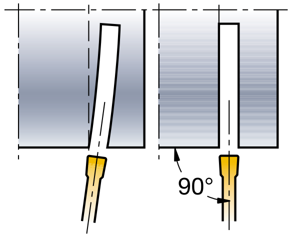


Use maximum deviation of ±0.1 mm (±.004 inch) from center line.

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

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

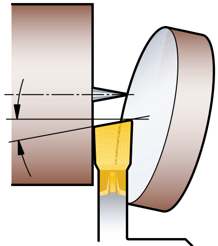
Positioning of the tool

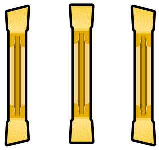


90º mounting of tool holder

- Perpendicular surface
- Reduce vibrations.

Hand of insert



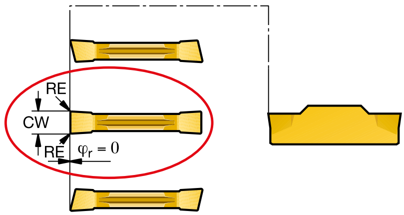


Three types of insert with different entering angles:

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

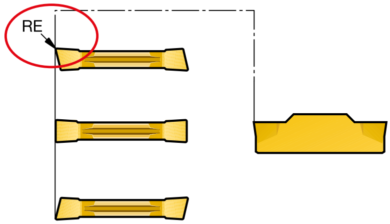
Insert geometry

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

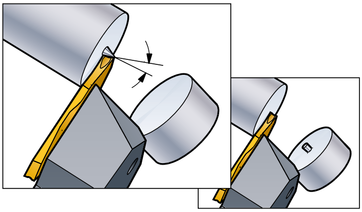
- Smaller pip
- Better chip control
- Lower feed rate.

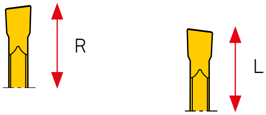
Large corner radius

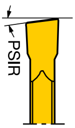
- Increased feed rate
- Longer tool life.

Parting off

Pip reduction by using different front angles





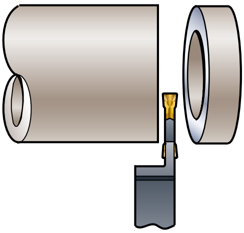


Example of front angles on 1-, 2- and 3-edge inserts:
KAPR = 95°, 98°, 100°, 102°, 105°, 110°
(PSIR = 5°, 8°, 10°, 12°, 15°, 20°)

- 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).

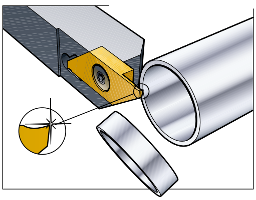
Note!
A front-angled insert will give reduced chip control due to the direction of the chip flow. (A neutral insert directs the chip straight out of the groove).

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:

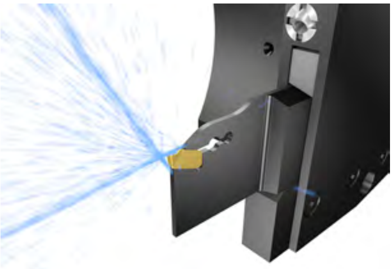
- Neutral inserts
- Smallest possible insert width
- Largest possible tool holder.

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.



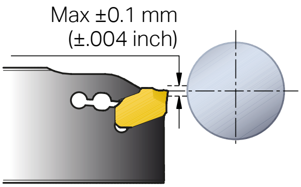
Apply:

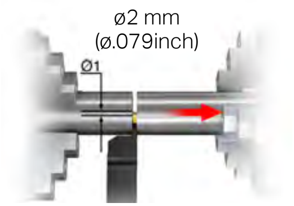
- Use large amounts
- Directly at the cutting edge
- Precision coolant.

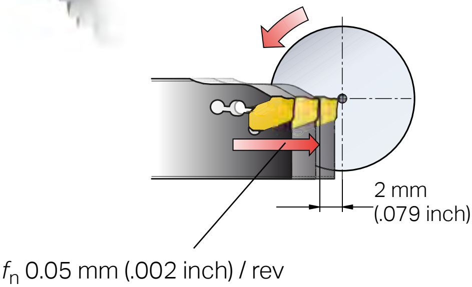
Result:

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

Practical hints



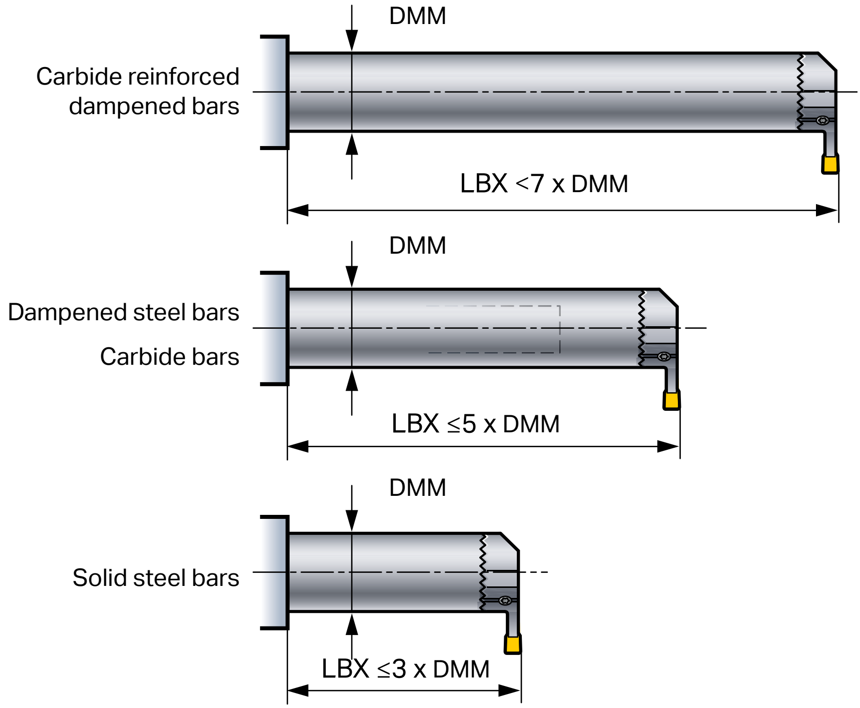




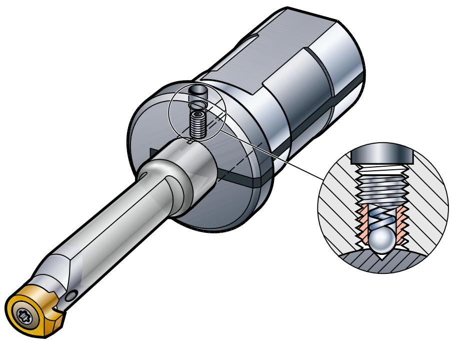
- Center height is important, ±0.1 mm (±.004 inch).
- If subspindle is used, pull away the component approximately 2 mm (.079 inch) before center.
- Recommended feed rate 0.05 (.002 inch) / per rev approximately 2 mm (.079 inch) before center – also for tube parting.

Recommendations for boring bar solutions

Recommended overhang

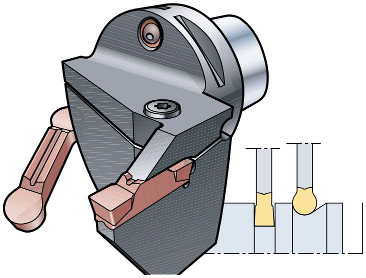


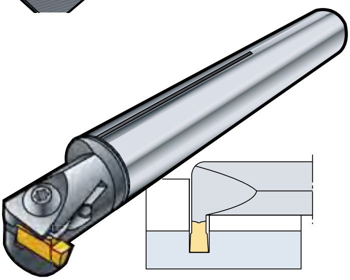
EasyFix sleeves



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

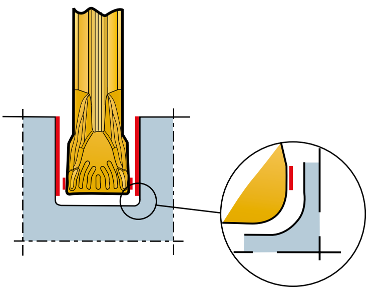
General grooving



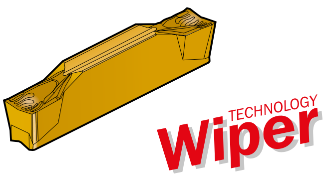


- Single cut grooving is the most economic and productive method to produce grooves.
- If the depth of the groove is bigger than the width, multiple grooving is the best method for rough grooving.
- A screw-clamp or spring-lock design holder should be selected for grooving operations.

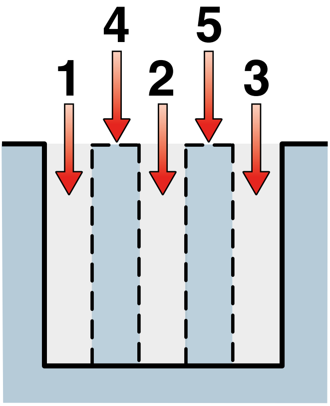
Single cut grooving



- 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

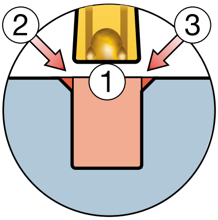


- The best method for rough grooving when depth is bigger than width.
- Use the insert width to produce full grooves and then remove the rings.

Practical hints

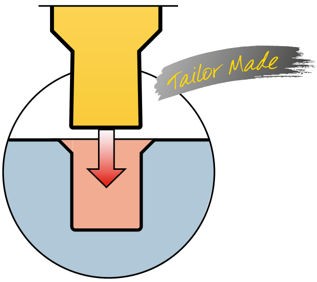
When producing high quality grooves, there is often a need for chamfered corners.

A



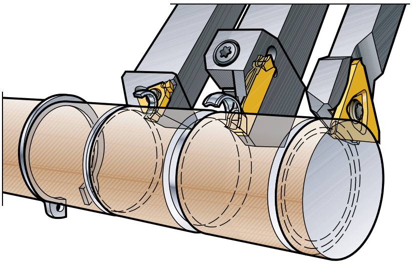
- One way is to use the corners on the insert, for example, of a finishing grooving insert, to chamfer; see illustration A.

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

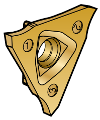


Circlips on shafts and axle components are very common.

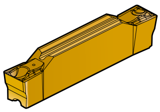
- Circlip grooving can be performed with three-edge inserts or two-edge grooving inserts.
- For internal grooving there is also a wide choice of inserts and boring bars.

Systems to choose from

3-edge inserts

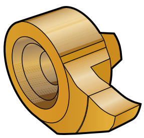


2-edge inserts

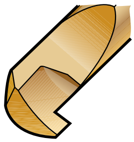


- For best economy, use 3-edge inserts in widths 1.00 - 3.18 mm (.039 - .125 inch).
- Or 2-edge inserts in widths 1.50 - 6.00 mm (.059 - .236 inch).

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.

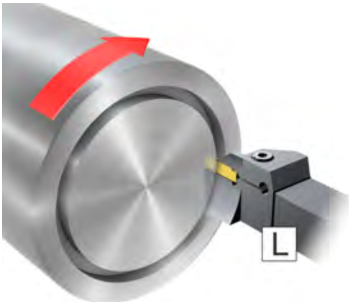
Tools for face grooving

- 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



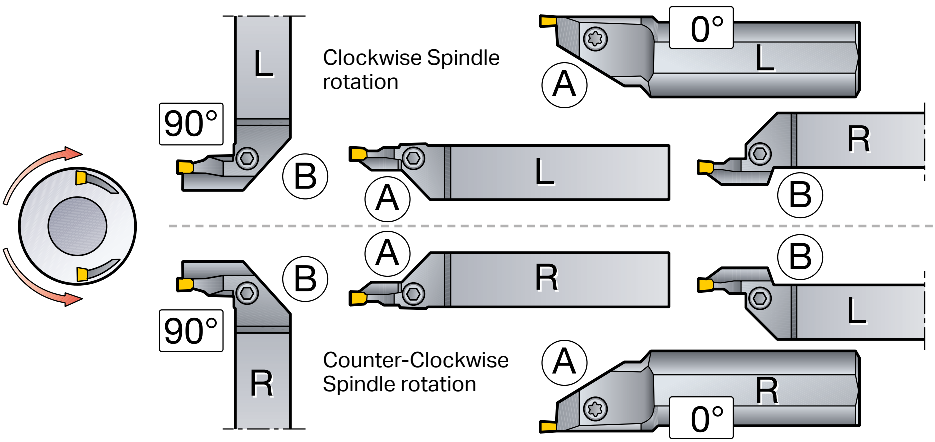
Left hand (L) tool

Right hand (R) tool

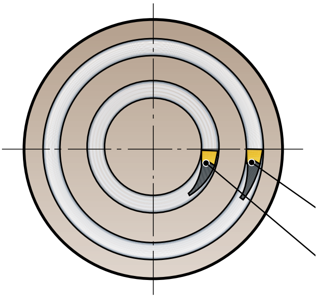
- Tool is fed axially towards the end surface of the part.
- Tool must be adapted to the bending radius of the groove.
- Machine largest diameter and work inwards for best chip control.

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.



First cut consederation

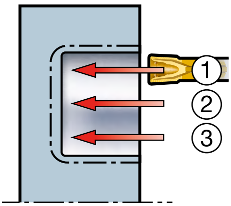


1 **If the insert support rubs workpiece inside dia:**
- maybe the dia. range is wrong
- tool is not parallel to axis
- check center height
- lower the tool below center line.

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

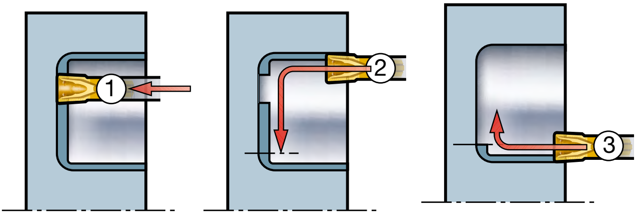
Roughing



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.

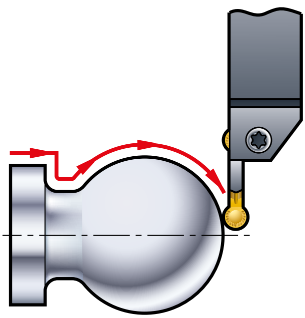
Cut two (2) finishes the diameter. Always start outside and turn inwards.

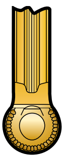
Finally, cut three (3) finishes the inner diameter to the correct dimensions.

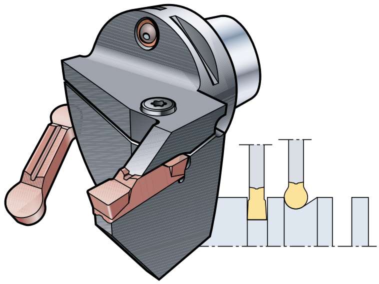
Profiling

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

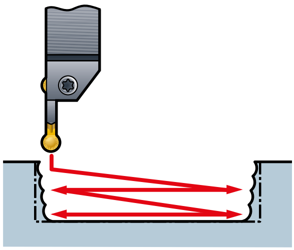
- Modern parting and grooving tool systems can also perform turning.
- A screw-clamp tool holder should be selected for turning and profiling operations in view of achieving maximum stablility.
- A neutral tool holder is suitable for both opening up or completing a recess.
- The round shape inserts have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







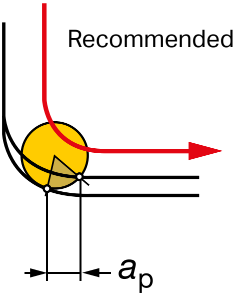
Ramping



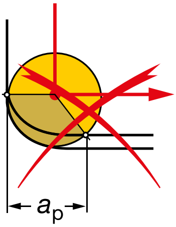
- Use round inserts for outstanding chip control and good surface finish.
- In unstable setups, use ramping to avoid vibrations.

Profile turning

Insert radius < component radius



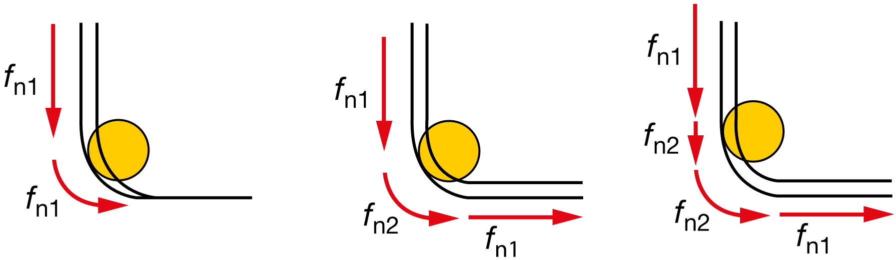
- Large area of insert creates high cutting force so feed should be reduced.
- If possible, use an insert radius that is smaller than the component radius.
- If you must have the same insert radius as the component radius, use micro-stops to make the chip short and avoid vibrations.



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.

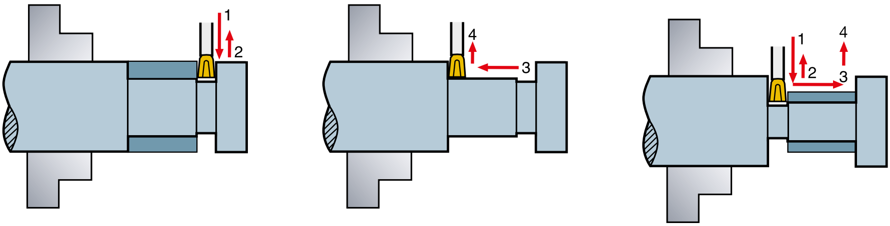


Turning

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 overhang, 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.

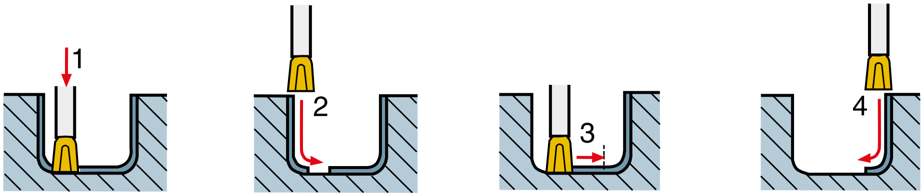
Roughing



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

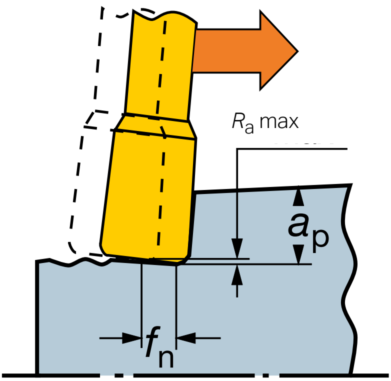
As the insert contours around the radius, most of the movement is in the Z direction. This produces an extremely thin chip along the front cutting edge which can result in rubbing and hence vibration.



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

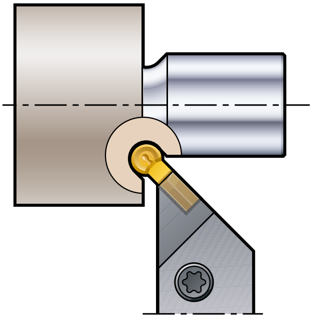
Axial turning

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.

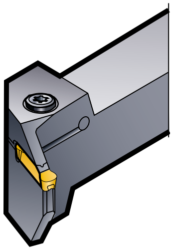
Undercutting



- 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).

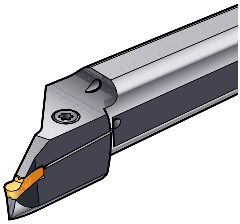
Tools for undercutting

Angled
7° ,45° and 70°



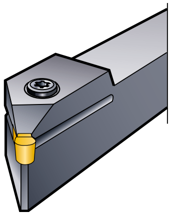
- Holder for external undercutting. Insert with two cutting edges.

Angled 20°



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

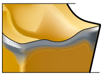
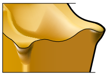
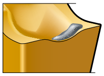
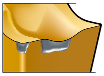
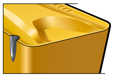
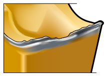
Angled 45°



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

Troubleshooting

Tool wear

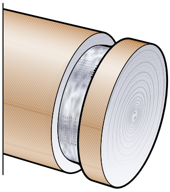
Problem / Solution	 Flank wear	 Plastic deforma-tion	 Crater wear	 Chipping	 Fracture	 Built-up edge
More positive geometry						++
Tougher grade				++		
More wear resistant grade	++	+	+			
Increase cutting speed						+
Decrease cutting speed	+	+	++			
Reduce feed rate		++		+	+	
Choose stronger geometry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

+ + = Best possible remedy

+ = Possible remedy

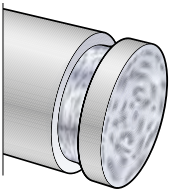
Problem	Solution

Bad surface



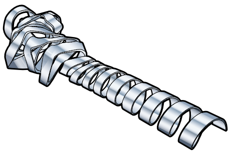
- Use a short and stable tool.
- Take away the chips, use geometry with good chip control.
- Use tools with precision coolant.
- Check speed/feed guidelines.
- Use wiper geometry.
- Check tool setup.

Bad surface on aluminum

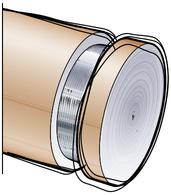
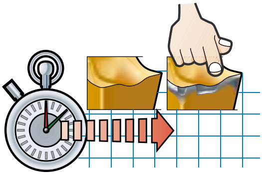


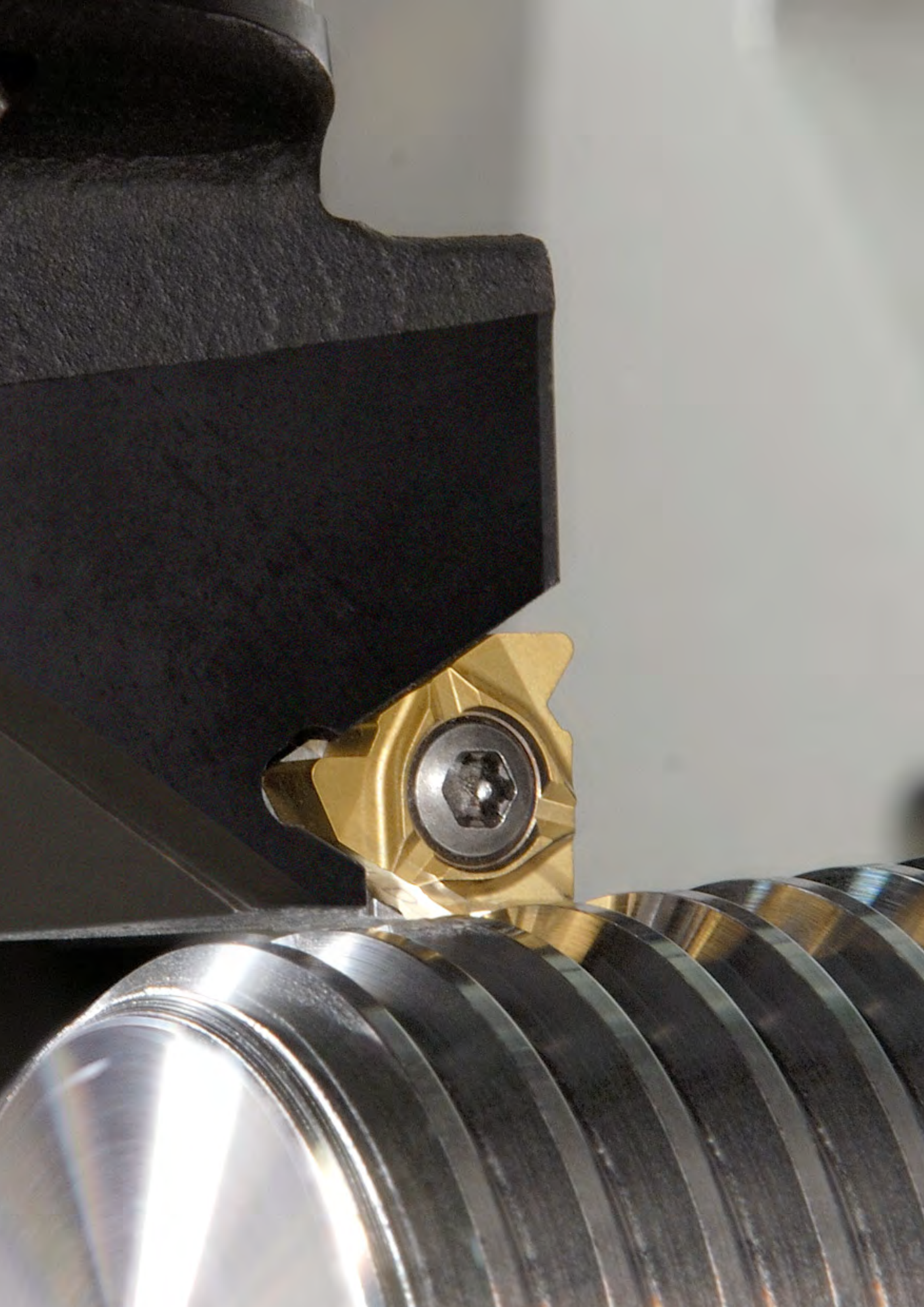
- Select the sharpest geometry.
- Use geometry with good chip control.
- Select a special soluble oil for the material.
- Use tools with precision coolant.

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 guide-lines. • Use shorter tool and compo-nent 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

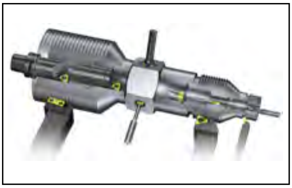
The prime functions of a thread are:

- 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.

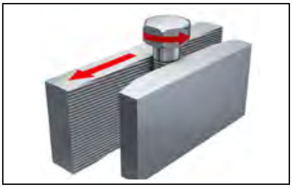
Different ways of making threads

Molding

Metal cutting



Rolling

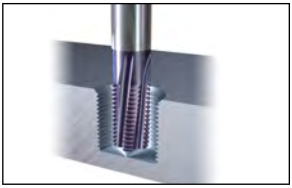


Metal cutting threading methods

Thread turning

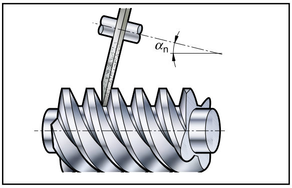
Tapping

Thread milling

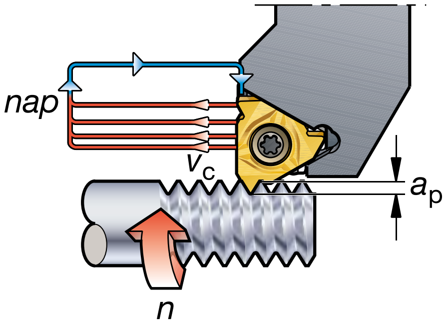


Thread whirling

Grinding



Definitions of terms

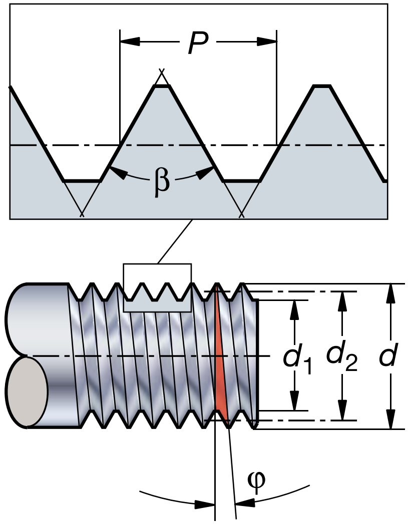


v_c = cutting speed m/min (ft/min)

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

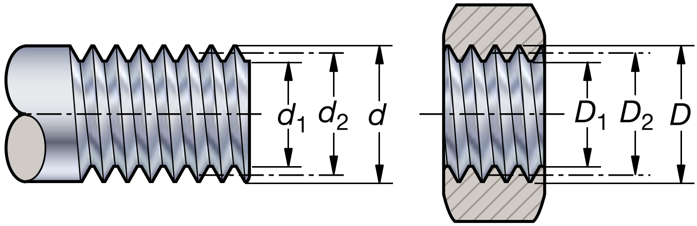
d_2 = pitch diameter external

D_2 = pitch diameter internal

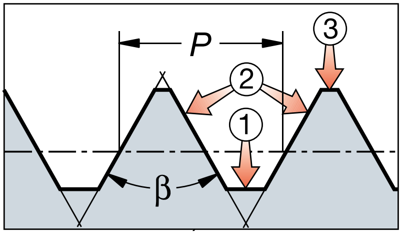
d = major diameter external

D = major diameter internal

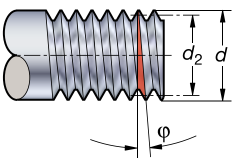
φ = helix angle of the thread

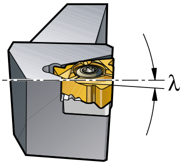


Definitions of terms



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.

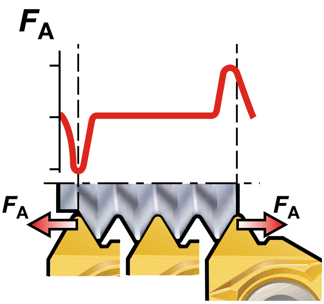




Helix angle

- 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

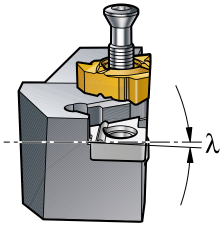


- The highest axial cutting force in the threading operation occurs during the entrance and exit of the cutting tool.
- Aggressive cutting data can lead to movement of insecurely clamped inserts.

Inclining the insert for clearance

Selecting shims for inclination

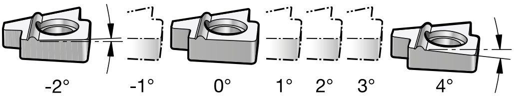
The inclination angle can be set using shims under the insert in the tool holder. The choice of which shim to use can be made by referring to a chart in the catalog.
As standard, all tool holders are delivered with the shim set at 1°.



Tangent of inclination angle

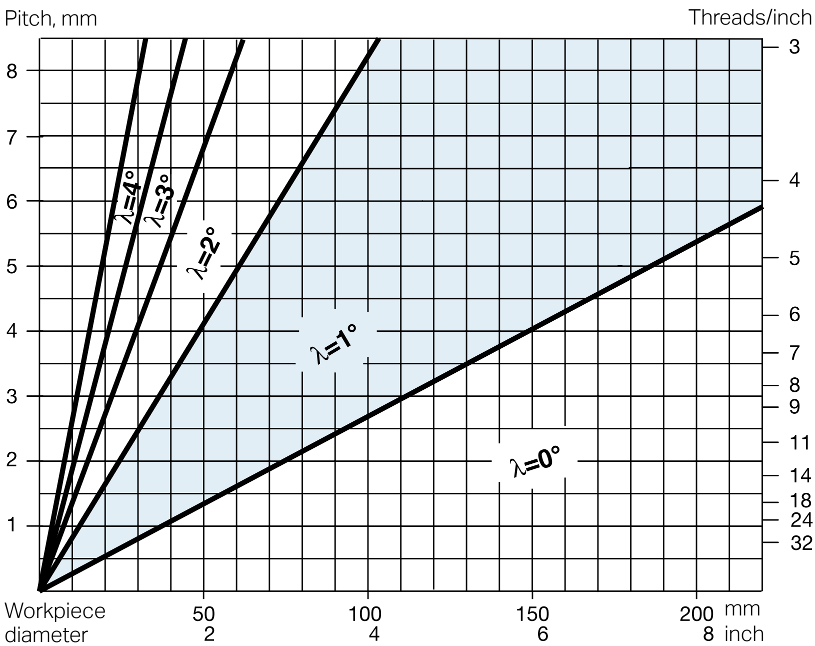
Note! that some pull threading operations require a shim with negative inclination angle.

Standard shim = 1°



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

* ns = number of starts



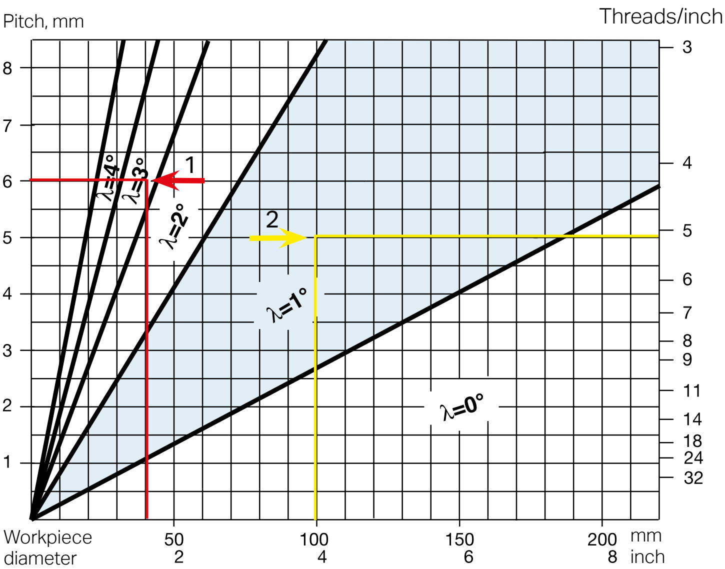
Selecting shims for inclination

The diameter and pitch influence the inclination angles.

Example of how to use the diagram.

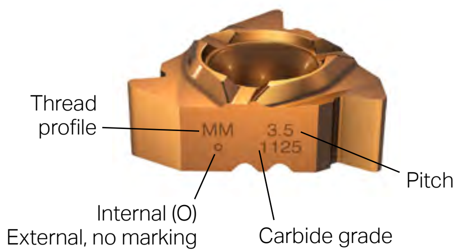
1. The workpiece diameter is 40 mm (1.575") with a thread pitch of 6 mm (.236"). From the diagram we can see that the required shim must have an inclination angle of 3º (standard shim can not be used).

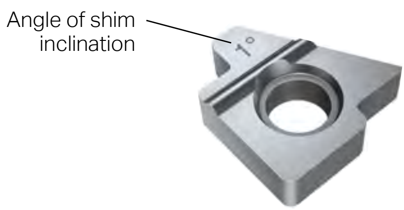
2. The workpiece diameter is 102 mm (4") with a thread of 5 threads per inch. From the diagram we can see that the required shim must have an inclination angle of 1º (standard shim can be used).



Marking of threading inserts and shims

How to read and understand markings.



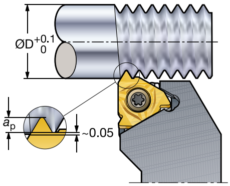


Tool selection procedure

Production planning process

1 **Component**

Dimension and quality of the thread



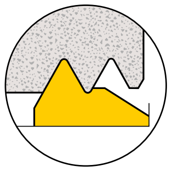
Workpiece material, thread profile and quantity

2 **Machine**

Machine parameters

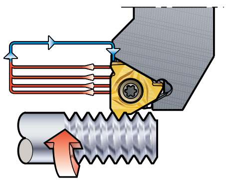
3 **Choice of tool**

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



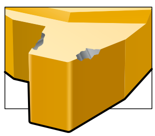
4 **How to apply**

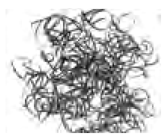
Cutting data, infeed etc.



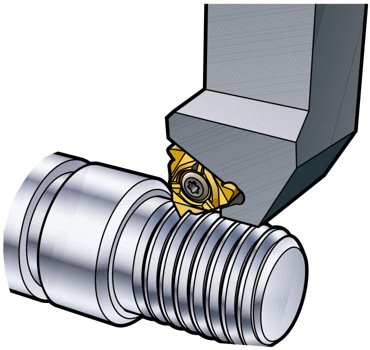
5 **Troubleshooting**

Remedies and solutions





1. Component and the workpiece material



Component

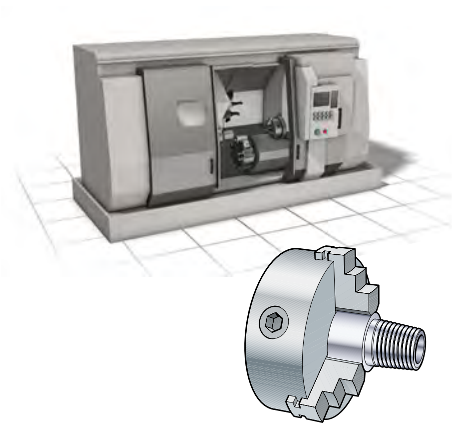
- Analyze the dimensions and quality demands of the thread to be machined
- Type of operation (external or internal)
- Right- or left-hand thread
- Type of profile (metric, UN, etc.)
- Pitch size
- Number of thread starts
- Tolerance (profile, position).

P	M	K
N	S	H

Material

- Machinability
- Chip breaking
- Hardness
- Alloy elements.

2. Machine parameters



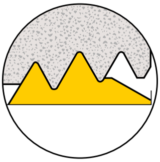
Condition of the machine and setup

- 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



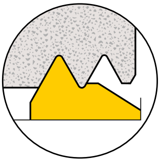
A full profile (topping) insert with several teeth reduces the number of required in-feeds and generates high productivity, e.g. a multi-point insert with two teeth reduces the number of in-feeds to half.

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.

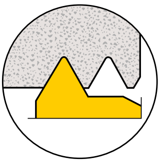
The insert can only cut one pitch.

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.

4. How to apply

Important application considerations

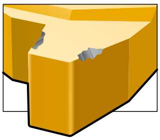
The infeed method can have a significant impact on the thread machining process.

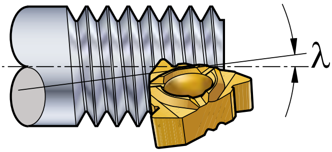
It influences:
- chip control
- insert wear
- thread quality
- tool life.

In practice, the machine tool, insert geometry, workpiece 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).

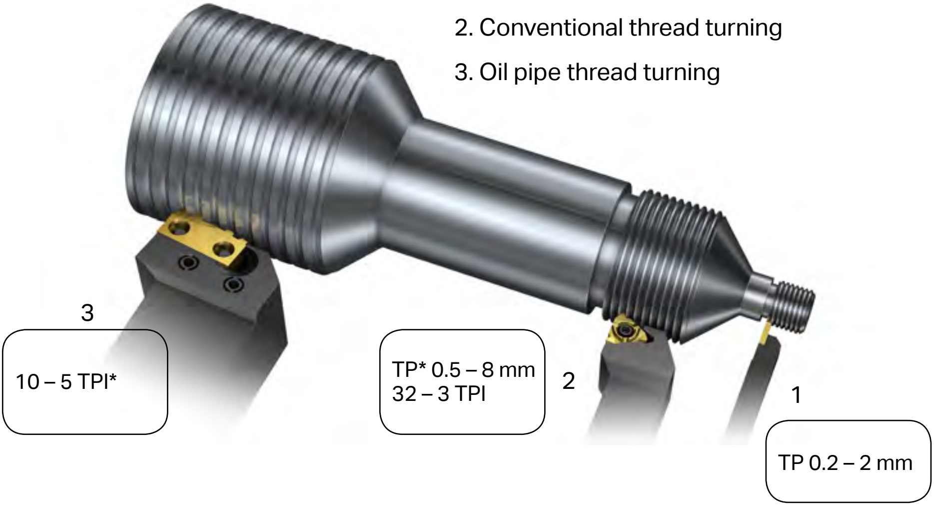
Insert grade
- Select the correct grade based on the material and toughness demands.

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

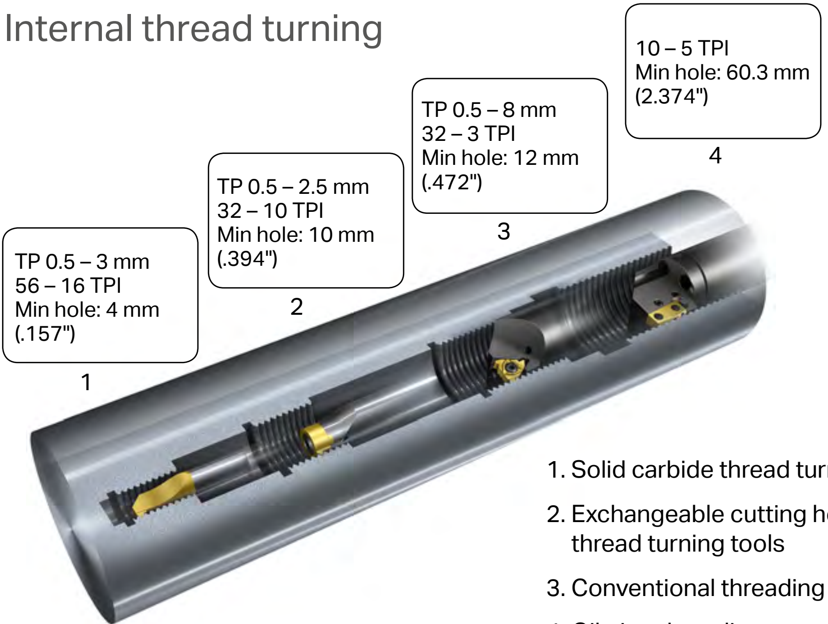
System overview

External thread turning

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



Internal thread turning



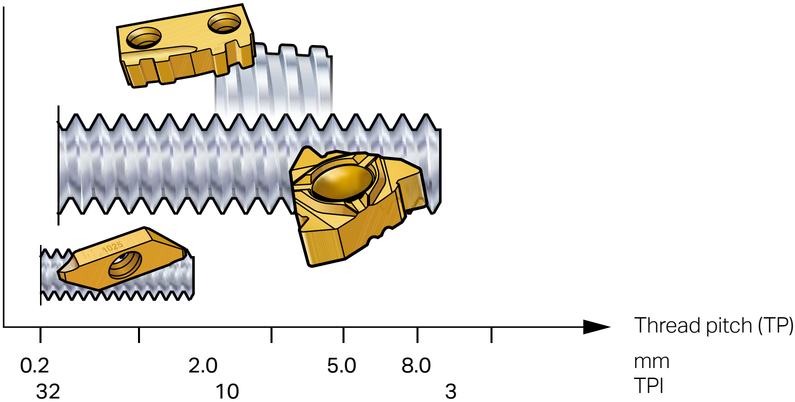
1. Solid carbide thread turning tools
2. Exchangeable cutting head thread turning tools
3. Conventional threading
4. Oil pipe threading

*TPI = Threads per inch
*TP = Thread pitch

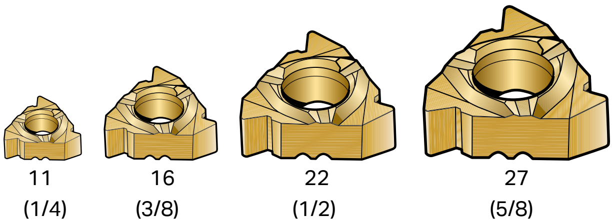
External thread turning assortment

Choose from an extensive program

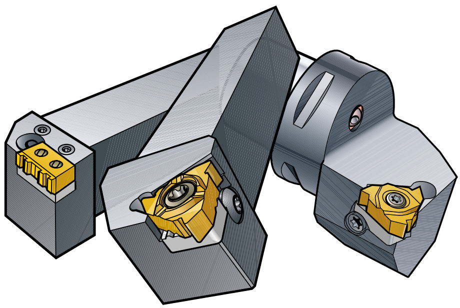
Inserts



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



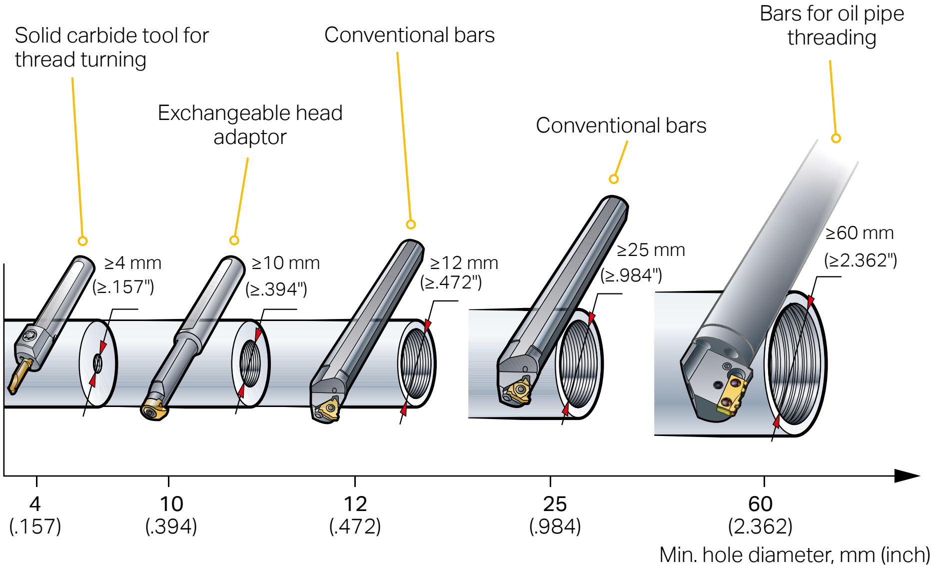
Tool holders



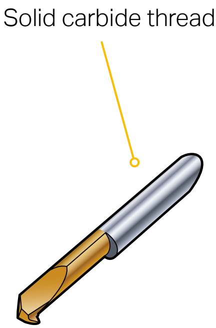
- Coromant Capto® cutting units
- QS-holders
- Shank tools
- Exchangable cutting heads
- Cartridges.

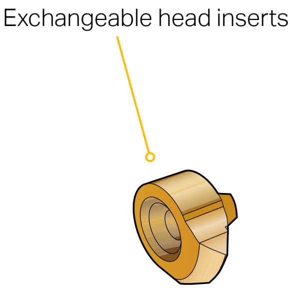
Internal thread turning assortment

Choose from an extensive program and several systems



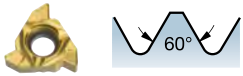
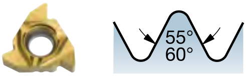
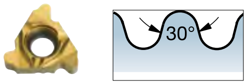
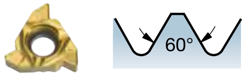
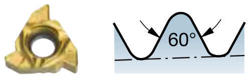
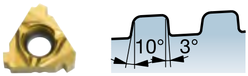
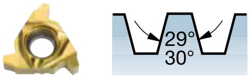
For high precision, internal thread turning of small holes





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

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.

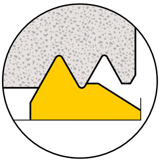
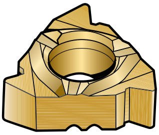
Aerospace
- High precision and minimized risk for stress concentration and breakage.

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.

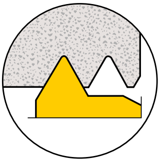
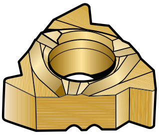
Insert types

Three different types of thread turning inserts

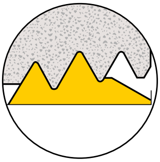
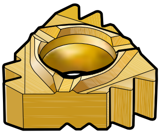
Full profile inserts

- For high productivity in threading.

V-profile inserts - 60º and 55º

- For threading with minimum tool inventory.

Multi-point inserts

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

Three different geometries

A-geometry

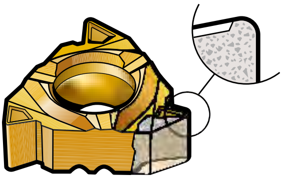
First choice in most operations.

F-geometry

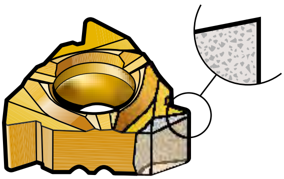
Sharp geometry.

C-geometry

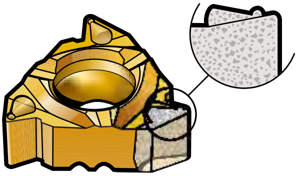
Chip breaking geometry.



Good chip forming in a wide range of materials.



Gives clean cuts in sticky and work hardening materials.



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

Threading solutions

iLock™
ingenious locking interface

- 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

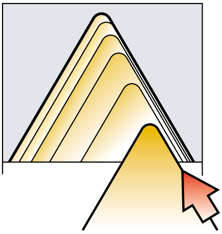
How to apply

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.

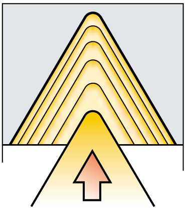
In practice, the machine tool, insert geometry, workpiece material and thread pitch influence the choice of infeed method.

Modified flank infeed



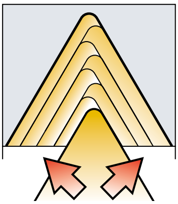
- Most newer CNC machines can be programmed for modified flank.
- Used with C-geometry as the chip breaker will not function with radial infeed.
- Axially directed cutting forces reduce the risk of vibrations.
- Controlled chip direction.
- Used for all insert geometries.
- C-geometry, designed only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

Radial infeed



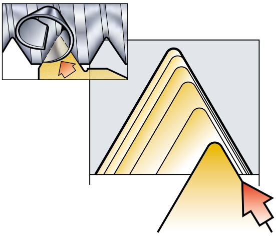
- Used by all manual machines and most canned CNC programs.
- First choice for work hardening materials and suitable for fine pitches.

Incremental infeed



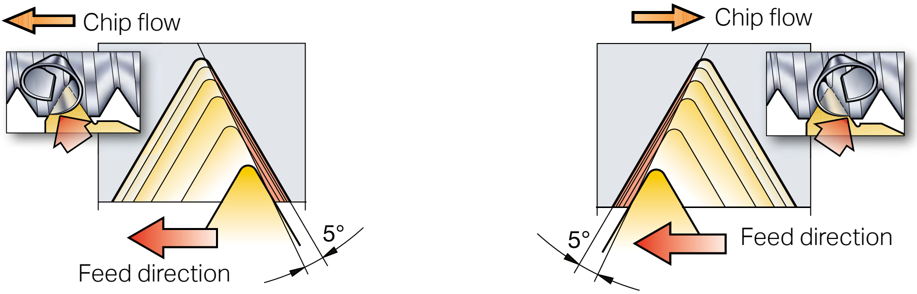
- Normally used with very large profiles and pitches, long work threading cycles where tool life needs to match the length of the thread.
- Requires special programming.

Modified flank infeed

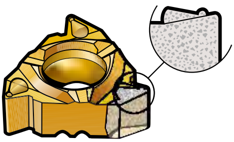


- Most CNC machines have a programmed cycle using this infeed.
- Chip is similar to that in conventional turning - easier to form and guide.
- Axially directed cutting forces reduce the risk of vibrations.
- Chip is thicker, but has contact with only one side of the insert.
- Less heat is transferred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

Infeed direction

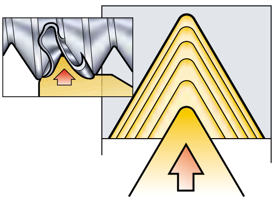


C-geometry insert



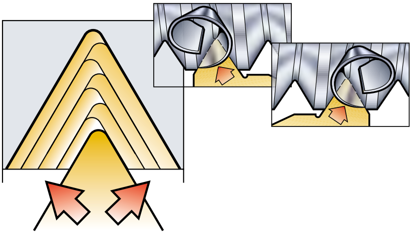
- Better chip control
- Better surfaces
- For C-geometry insert, modified flank infeed is the only suitable infeed.

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

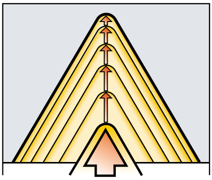


- Recommended for large profiles.
- Even insert wear and longest tool life in very coarse threads.
- Chips are directed both ways, making control difficult.

Programming methods

Ways of improving the machining result

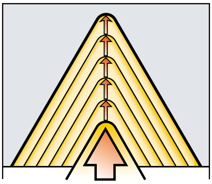
Decreasing depth per pass (Constant chip area)



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



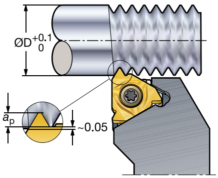
Each pass is of an equal depth, regardless of the number of passes.

- 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.

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

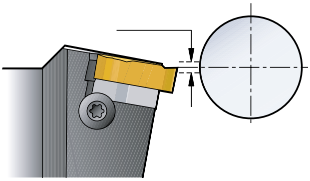
Number of infeeds and total depth of thread.

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



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

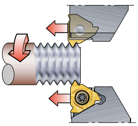
- The thread profile can be incorrect.

Method of thread turning

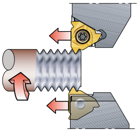
Right and left hand threads and inserts

External

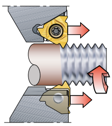
Right hand threads | Left hand threads



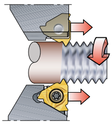
Right hand tool/insert



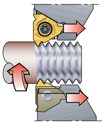
Left hand tool/insert



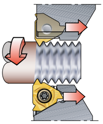
Right hand tool/insert



Left hand tool/insert



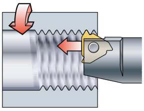
Left hand tool/insert



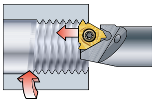
Right hand tool/insert

Internal

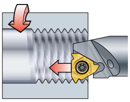
Right hand threads | Left hand threads



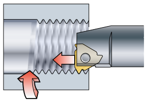
Right hand tool/insert



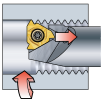
Left hand tool/insert



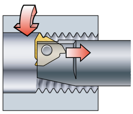
Right hand tool/insert



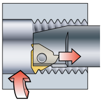
Left hand tool/insert



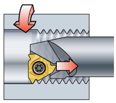
Left hand tool/insert



Right hand tool/insert



Left hand tool/insert



Right hand tool/insert

Negative shims must be used in a pull threading application.

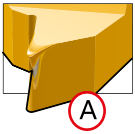
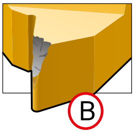
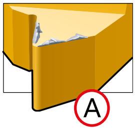
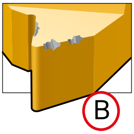
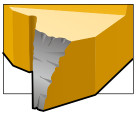
A Turning
B Parting and grooving
C Threading
D Milling
E Drilling
F Boring
G Tool holding
H Machinability Other information

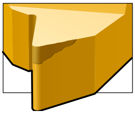
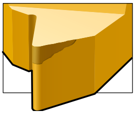
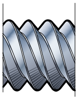
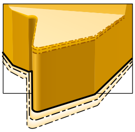
Thread turning application hints

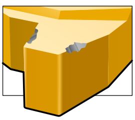
Some vital factors to consider to achieve success

- 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.

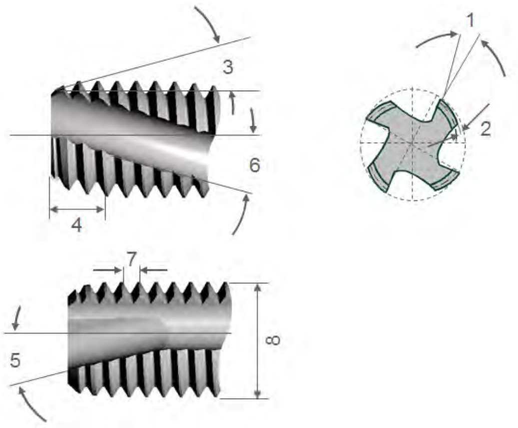
Problem	Cause	Solution
Rapid flank wear 	1. Highly abrasive material. 2. Cutting speed too high. 3. Infeed depths too shallow. 4. Insert is above center line.	1. Wrong grade. Choose a more wear resistant grade. 2. Reduce cutting speed. 3. Reduce number of infeeds. 4. Correct center height.
Abnormal flank 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

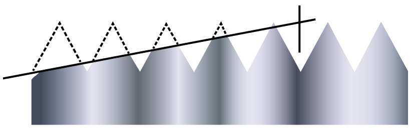
Tapping theory

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

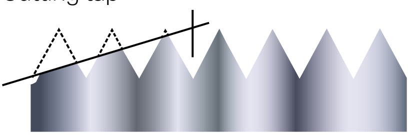
Long chamfer



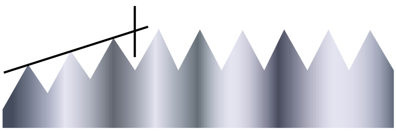
- High torque
- Best surface quality
- Thin chips
- Low pressure at the chamfer
- Longer tool life
- Most common for spiral point tap.

Medium chamfer

Cutting tap

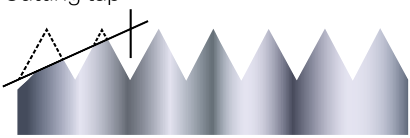


Forming tap

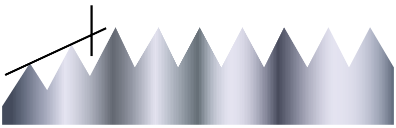


Short chamfer

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

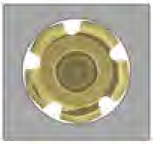
- A chip-free tap solution
- For soft steel, stainless steel and aluminum
- Can be used for all types of holes and depths
- Increases the strength of the thread in some materials, e.g. aluminum.

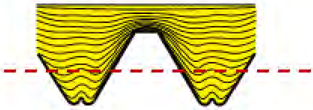
Forming and tapping processes

Forming tap

The thread is formed by deforming the material.

No chips are generated.



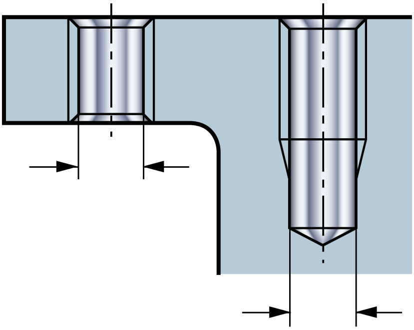


Cutting tap

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)).

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

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 evacuation, 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.

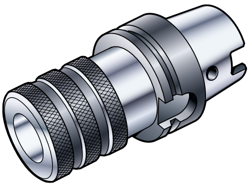
Quick change tapping chuck

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

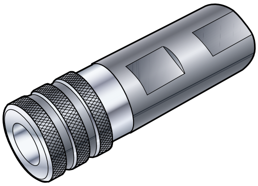
Benefits and recommendations

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

Coromant Capto®



HSK solid holder



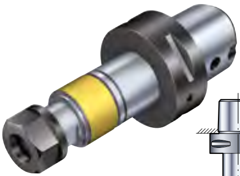
Weldon solid holder

Synchronous feed tap chucks

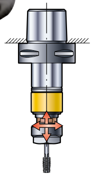
Rigid tap holder with micro float compensation for elimination of oversized threading. First choice for CNC machine tools and synchronized tapping operations.

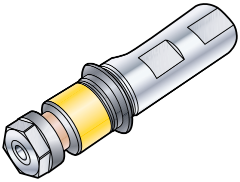
Benefits and recommendations

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

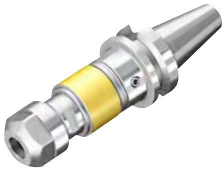


Coromant Capto®

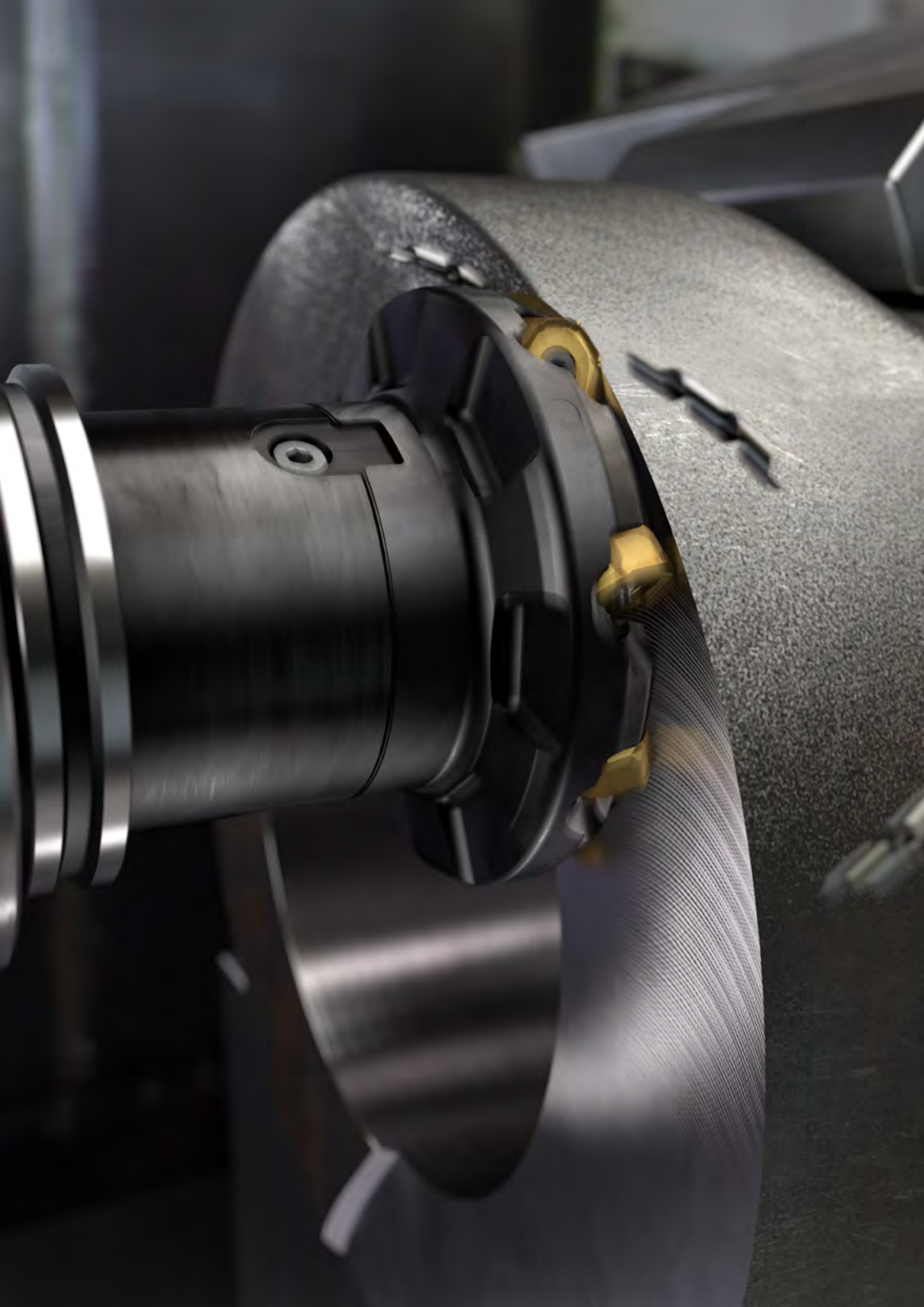




Weldon shank holder



MAS-BT solid holder



Milling

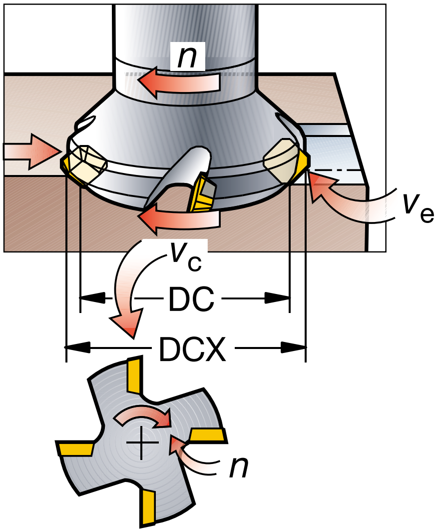
Milling is performed with a rotating, multi-edge cutting tool which performs programmed feed movements against a workpiece in almost any direction.

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



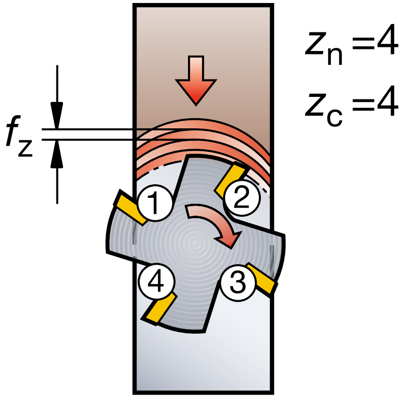
n	= Spindle speed, rpm (revolutions per minute)
v_c	= Cutting speed m/min (ft/min)
v_e	= Effective cutting speed m/min (ft/min)
DC	= Cutter diameter mm (inch)
DCX	= Maximum cutting diameter mm (inch)

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.

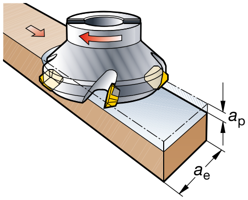
Feed per minute, v_f mm/min (inch/min), also known as the table feed, machine feed or feed speed is the feed of the tool in relation to the workpiece in distance per time-unit related to feed per tooth and number of teeth in the cutter.

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



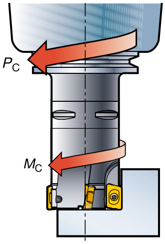
a_e = Radial depth of cut mm (inch) [working engagement]

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

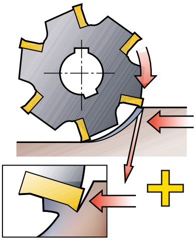
$$M_c = \frac{P_c \times 30 \times 10^3}{\pi \times n} \quad \text{Nm}$$

Inch

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

Climb or conventional milling

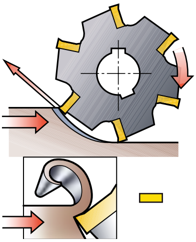
Climb milling – preferred method



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.

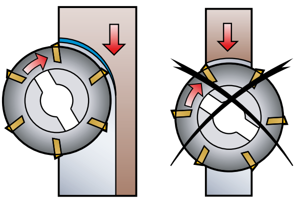
- In conventional milling (also referred to as up milling) the chip thickness starts at zero and increases to the end of the cut.

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.

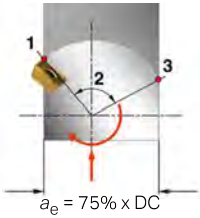


- Cutter diameter should be 20 – 40% larger than the width of cut.
- 2/3 rule (i.e., 150 mm (5.906 inch) cutter)
  - 2/3 in cut, 100 mm (3.937 inch)
  - 1/3 out of cut, 50 mm (1.969 inch).
- By moving the milling cutter off the center, a more constant and favorable direction of cutting forces will be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

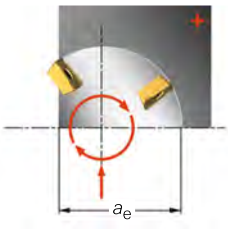
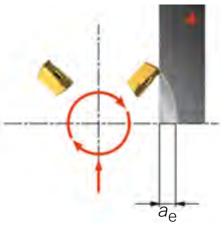
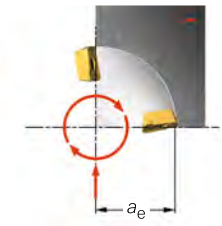
Chip formation through cutter position

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

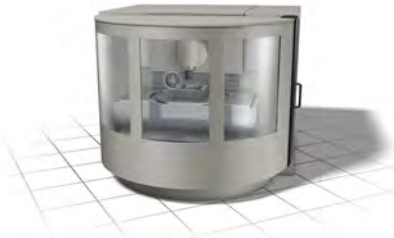
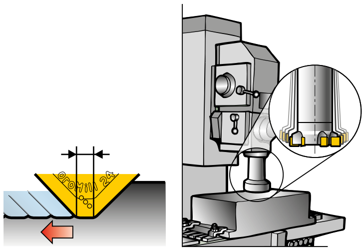
+ 	The centerline of the cutter is well inside the workpiece width, a_e >75% of DC. • Most favorable cutting conditions and optimized use of the cutter diameter. • The initial impact at the entry of cut is taken up further along the cutting edge, away from the sensitive tip. • The insert leaves the cut 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

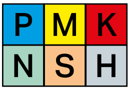
1	Component	Type of operation and method	
		Workpiece material and quantity	
2	Machine	Machine parameters	
3	Choice of tool	Select type of cutter	
4	How to apply	Cutting data, method etc.	
5	Troubleshooting	Remedies and solutions	

1. Component and the workpiece material

Parameters to be considered

Geometric shape

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



Material

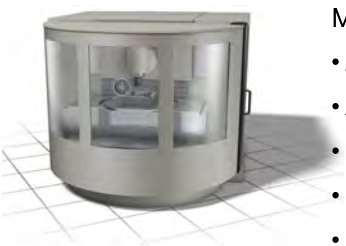
- Machinability
- Cast or pre-machined
- Chip forming
- Hardness
- Alloy elements.

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

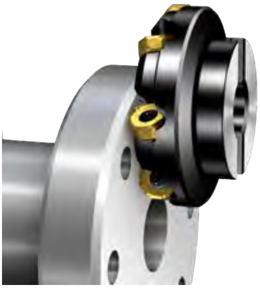
Advantages

- 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

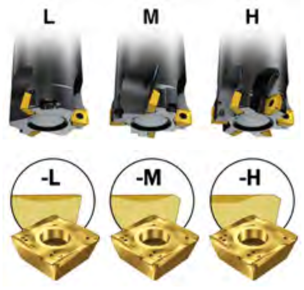
- Great versatility
- Large depth of cut
- Low axial cutting forces (thin workpieces)
- Light-cutting inserts with true four edges.

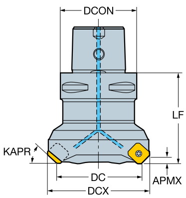
Disadvantages

- 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

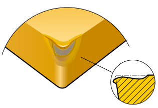
- Choose largest possible spindle size or outer diameter.

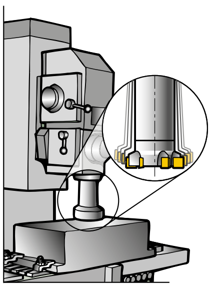
Chip formation through cutter positioning

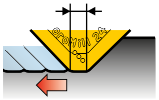
- Always use climb milling
- Move the cutter off the center
- Use a cutter with a diameter 20–50% larger than the cut.

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

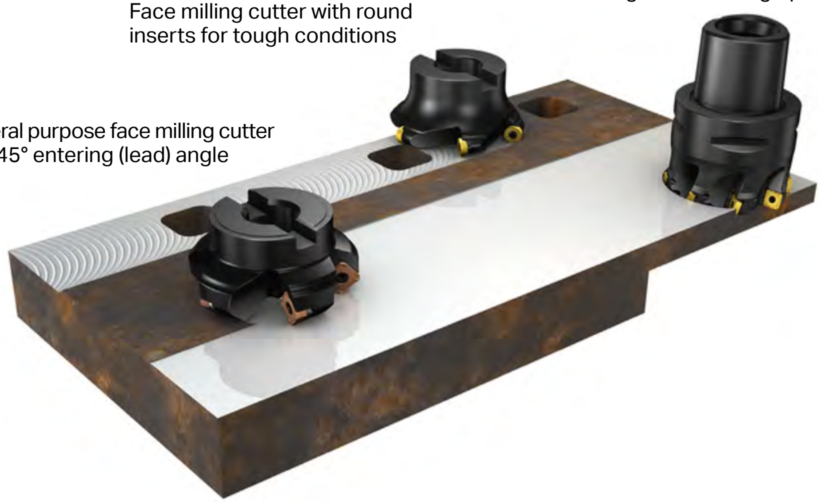
Face milling

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



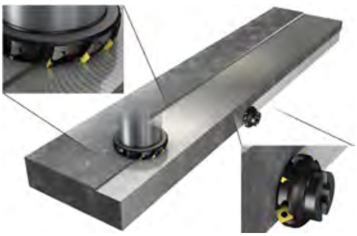
Dedicated cutters

High feed face milling

Face milling cutters for cast iron machining

K

Heavy duty face milling

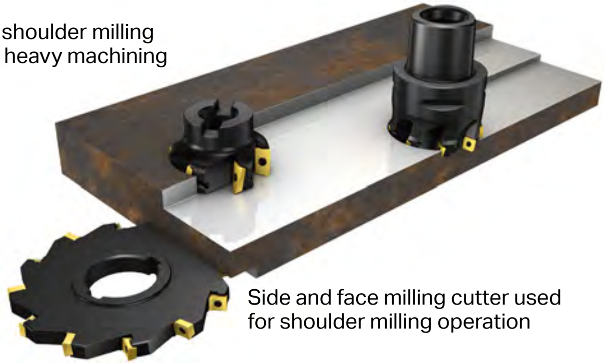


Face milling cutters for aluminum machining

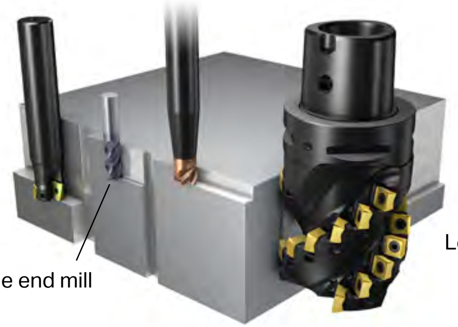
N

Shoulder milling

Cutters for general use



Dedicated end mills and long edge cutters

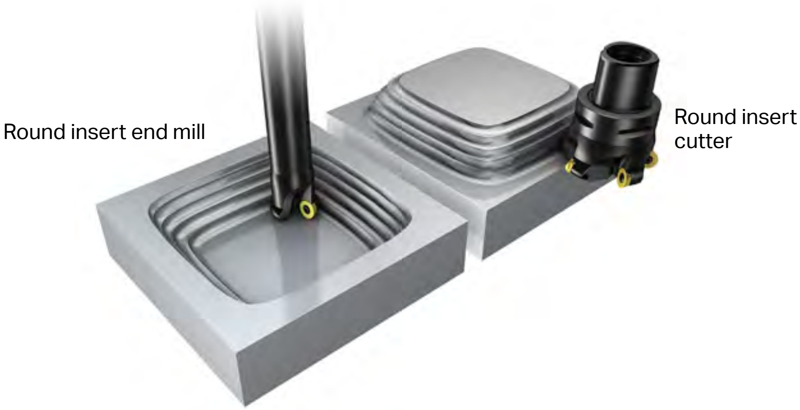


Deep shoulder milling

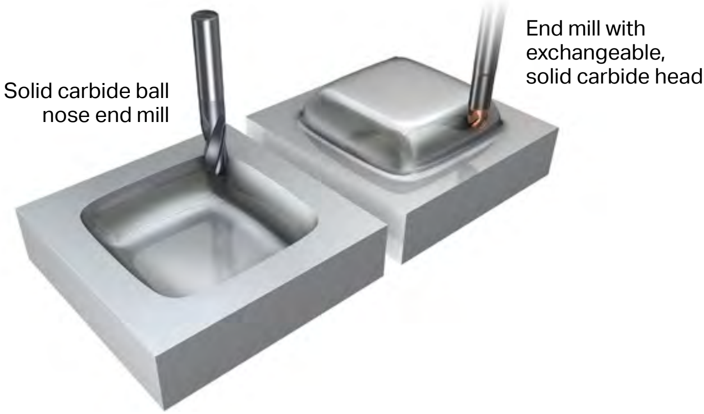
Edging with square shoulder milling cutters

Profiling

Cutters for general use – roughing



Cutters for general use – finishing



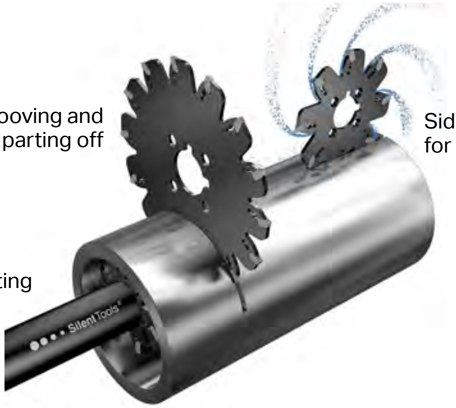
Other methods

Turn milling

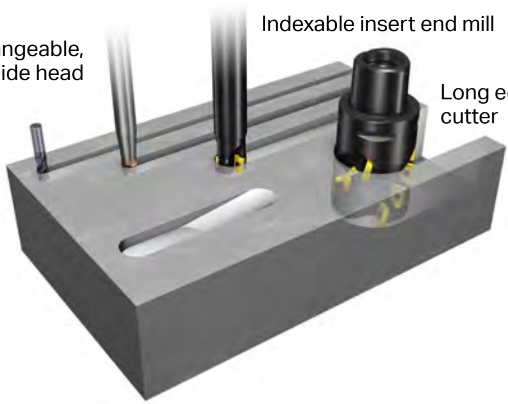
Blade milling

Groove milling

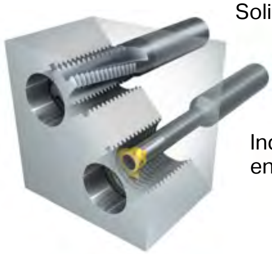
Cutters for general use – radial groove milling



Cutters for general use – axial slot milling



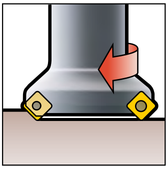
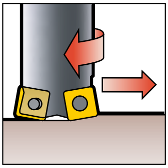
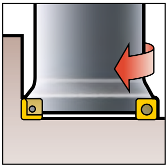
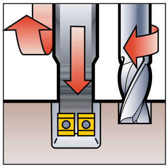
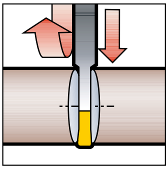
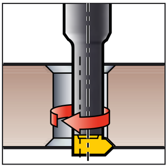
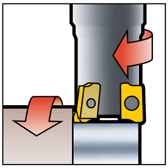
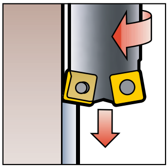
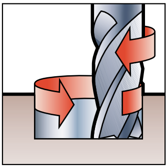
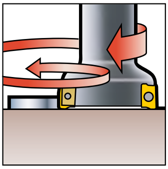
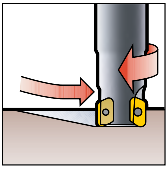
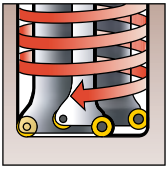
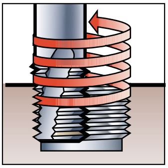
Thread milling and shallow grooving



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.

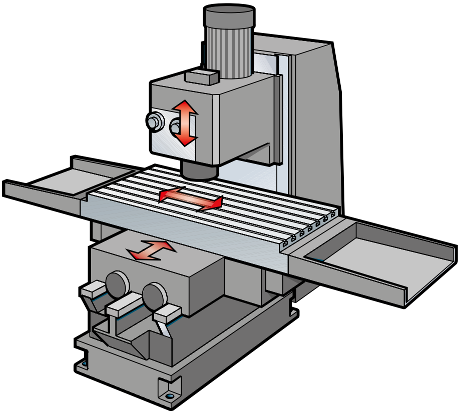
Face milling	High-feed milling	Shoulder milling	Groove milling
			
Parting off	Chamfering	Profile milling	Turn milling
			
Plunge milling	Trochoidal milling	Circular milling	Linear ramping
			
Helical interpolation	Thread milling		
			

Milling methods

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

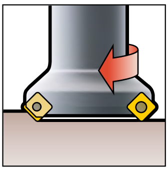
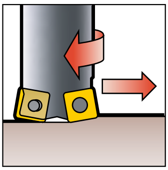
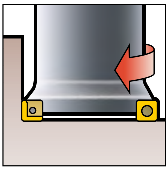
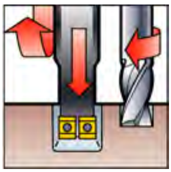
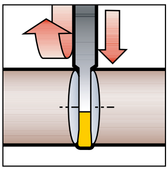
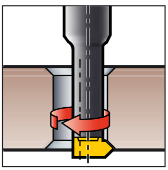
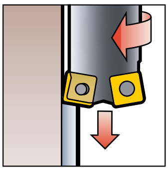
Conventional milling methods

Vertical milling machines



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

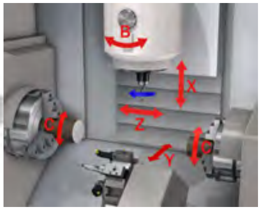
Surfaces and forms, other than those described below, are increasing steadily as the number of five-axis machining centers and multi-task machines grows.

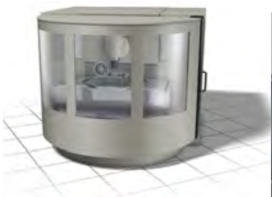
Face milling	High-feed milling	Shoulder milling	Groove milling
			
Parting off	Chamfering	Plunge milling	
			

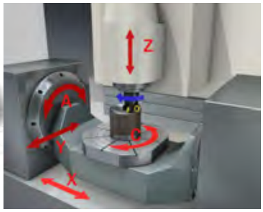
Advanced milling methods

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.

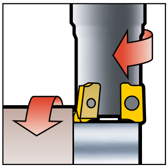
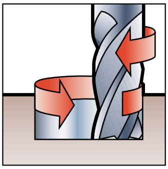
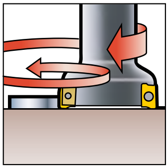
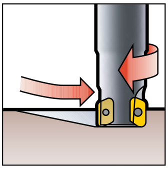
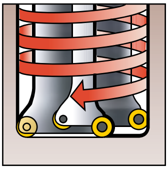
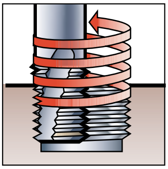






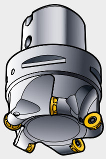
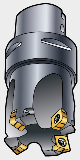
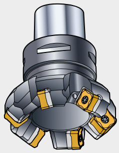
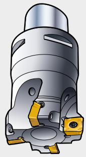
The results of these trends and the development of methods put new demands and opportunities on the tooling, such as:

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

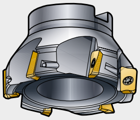
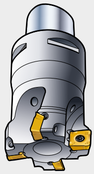
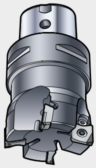
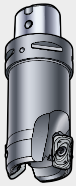
Profile milling	Turn milling	Trochoidal milling	Circular milling
			
Linear ramping	Helical interpolation	Thread milling	
			

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

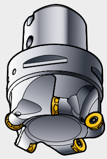
Positioning of cutters for face milling

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

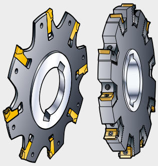
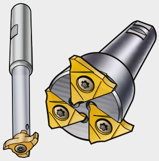
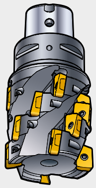
Positioning of cutters for shoulder milling

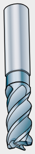
Type of milling cutter / Considerations				
	90°	90°	90°	90°
Machine/spindle size	ISO 40, 50	ISO 30, 40, 50	ISO 40, 50	ISO 30, 40, 50
Stability requirement	High	High	Medium	Low
Roughing	Very good	Good	Acceptable	Good
Finishing	Acceptable	Acceptable	Very good	Good
Cutting depth a_p	Large	Medium	Small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

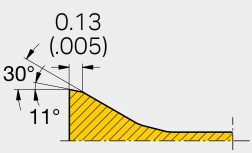
Choice of inserts and how to apply

Modern milling inserts for face milling operations.

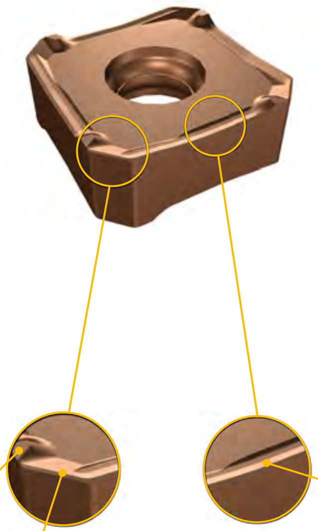
The design of a modern milling insert

Definitions of terms and geometry design

Corner design



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

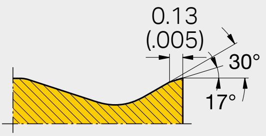


Corner reinforcement

Chip former

Main cutting edge design

Main cutting edge design



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

Making the tool choice in milling

Low	First choice Operation stability Cutter pitch	High
 L	 M	 H
Coarse pitch (-L)	Close pitch (-M)	Extra close pitch (-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

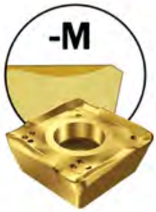
Light milling
- Operations at small depth of cut and low feed rates.
- Operations requiring low cutting forces.

f_n Feed, mm/tooth (inch/tooth)

Selecting the insert geometry

Light (-L)

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



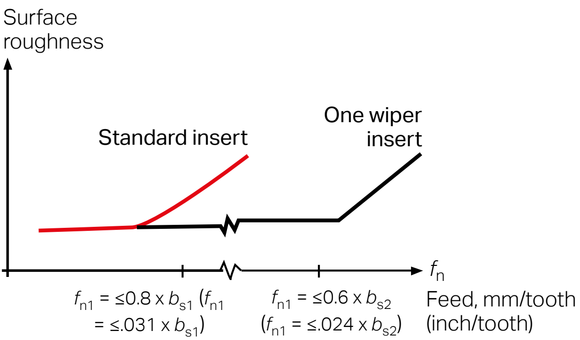
Medium (-M)

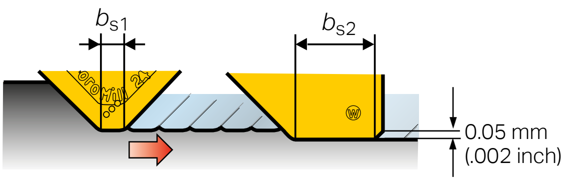
- General purpose geometry
- Medium feed rates
- Medium operations to light roughing.

Heavy (-H)

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

Achieving good surface finish in milling





b_s, mm 2.0 (inch) (.079)

8.2 (.323)

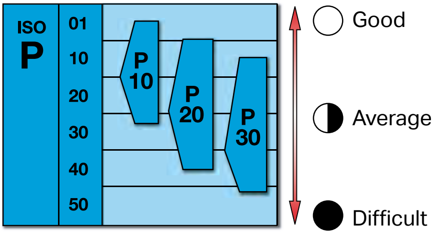
- Use wiper inserts for higher productivity and improved surface finish
- Limit the feed to 60% of the parallel land
- Mount the wiper inserts correctly
- Set the wiper inserts below other inserts.

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

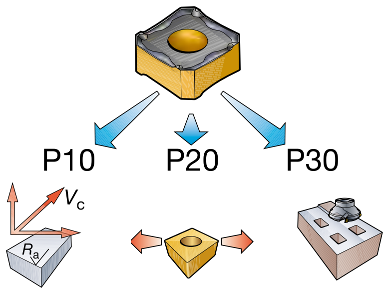
How to select insert grade

Select the geometry and grade according to the application.

Build-up of a grade chart



Machining conditions



Define machining conditions

Good conditions	Average conditions	Difficult conditions
• Cutting depth 25% of max a_p or less	• Cutting depth 50% of max a_p or more	• Cutting depth 50% of max a_p or more
• Overhang under two times cutter diameter	• Overhang two to three times cutter diameter	• Overhang over three times cutter diameter
• Continuous cuts	• Interrupted cuts	• Interrupted cuts
• Wet or dry machining.	• Wet or dry machining.	• Wet or dry machining.

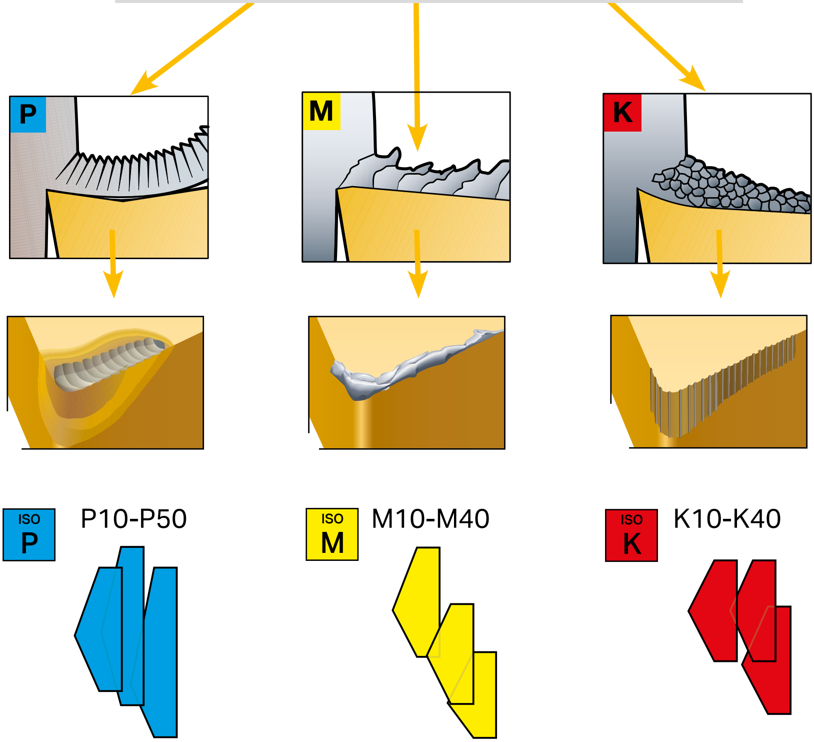
Dedicated grades for ISO P, M and K

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.

Select geometry and grade depending on the type of workpiece material and type of application.



Choice of cutter and how to apply

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

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)
		
Light (-L)	Medium (-M)	Heavy (-H)
Wear resistant ←	Machining conditions/ Grades	→ Toughness
○	◐	●
Good conditions	Average conditions	Difficult conditions

Selecting cutter pitches

Low ←	First choice Operation stability Cutter pitch	→ High

L
Coarse pitch (-L)

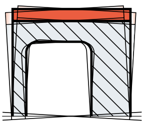
- Reduced number of inserts
- Limited stability
- Long overhang
- Small machines/limited horsepower
- Deep, full slotting operations
- Differential pitch.

M
Close pitch (-M)

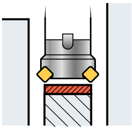
- General purpose
- Suitable for mixed production
- Small to medium machines
- Usually first choice.

H
Extra close pitch (-H)

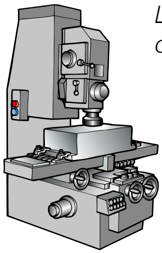
- High number of inserts for maximum productivity
- Stable conditions
- Short chipping materials
- Heat resistant materials.



Limited stability

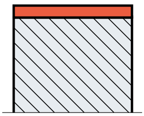


Long overhang

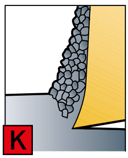


Limited horsepower

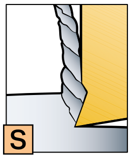
First choice



Stable conditions



Cast iron (CMC 08)

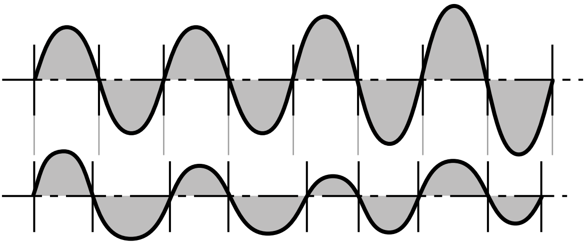


Heat resistant alloys (CMC 20)

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.

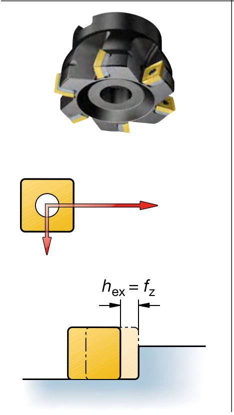
Differential pitch cutters have uneven tooth spacing, which impacts the vibration amplitude of each tooth. Reducing the risk of vibration.

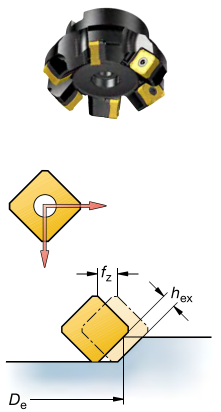


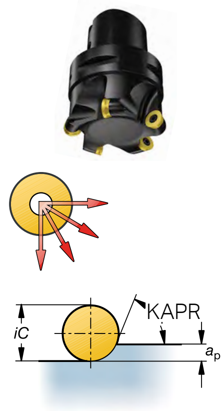
Differential pitch reduces the risk of vibration.

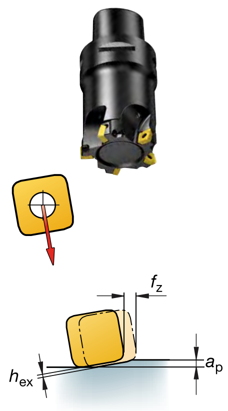
Cutting forces and entering angle

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



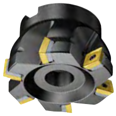
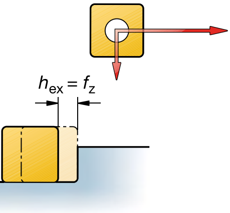






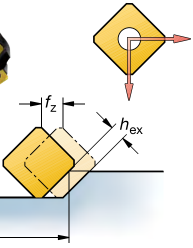
Axial and radial cutting forces

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).

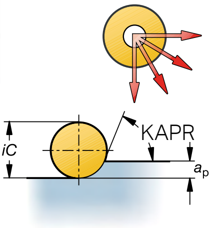
Effect of entering angle (45º)

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

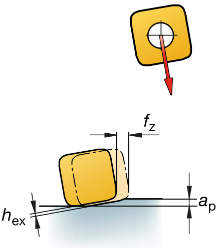
Effect of entering angle (Variable)

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

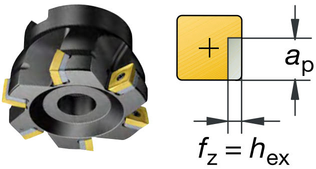
- Strongest cutting edge with multiple indexes
- General purpose cutter
- Increased chip thinning effect for heat resistant alloys
- h_{ex} = depends on a_p.

10° entering angle

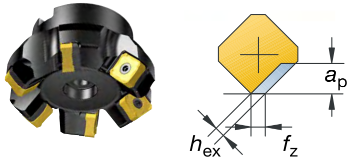



- High-feed milling cutters
- A thin chip is generated, allows very high feeds per tooth
- Axial cutting force is directed towards the spindle and stabilize it.

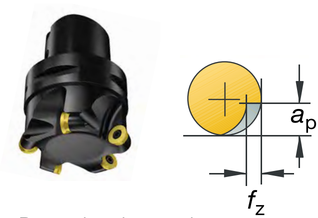
Feed compensation for different entering angles



90° = (f_z or h_{ex}) × 1.0



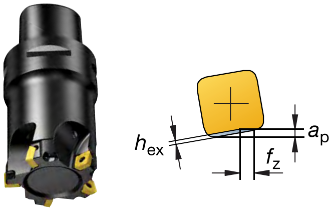
45° = (f_z or h_{ex}) × 1.41



Round = depends on a_p

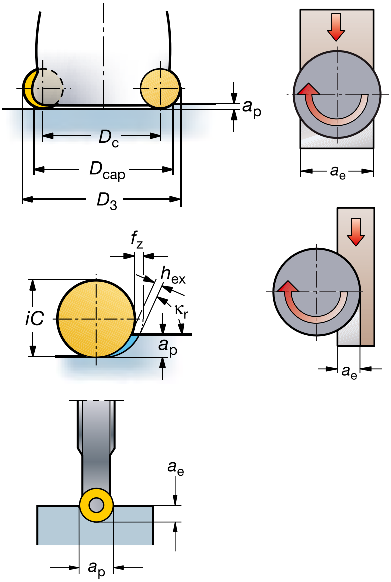
$$\sqrt{\frac{iC}{a_p}}$$

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).

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

Calculating cutting data

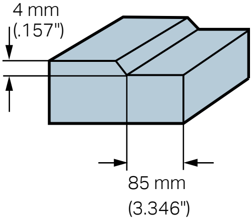
Example in face milling

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 cm³/min (inch³/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

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

Inch

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

Milling with large engagement

ISO	CMC No.	Material	Specific cutting force kc 1 N/mm²	Hardness Brinell HB	mc	CT530 Max chip thickne[ss] 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	m
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.
446,500	380	0.
204,000	150	0.
230,500	200	0.
283,500	200	0.

$$P_c = \frac{85 \times 4 \times 600 \times 1700}{60 \times 10^6} = 5.8 \text{ kW} \qquad P_c = \frac{3.346 \times .157 \times 23.6 \times 246500}{396 \times 10^3} = 7.7 \text{ Hp}$$

The calculation above is approximate and valid for an average chip thickness (h_m) of 1 mm (0.039 inch) .
For a more accurate value of power consumption (P_c) the k_c value should be calculated accordingly.

Metric

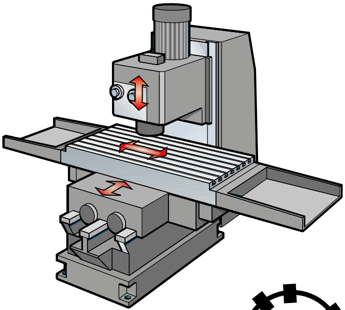
$$k_c = k_{c1} \times h_m^{-mc} \times \left(1 - \frac{\gamma_o}{100}\right) \text{ (N/mm}^2\text{)}$$

Inch

$$k_c = k_{c1} \times \left(\frac{0.039}{h_m}\right)^{-m_c} \times \left(1 - \frac{\gamma_o}{100}\right) \text{ (lbs/inch}^2\text{)}$$

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 1 mm (0.039 inch).

Application hints for milling



Power capacity

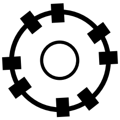
• Check power capability and machine rigidity, making sure that the machine can handle the cutter diameter required.

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

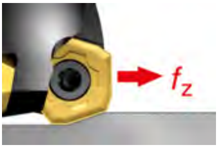
• Use the correct cutter pitch for the operation to ensure that there are not too many inserts engaged in cut, as this may cause vibration.

Cutting engagement

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

Choice of insert geometry

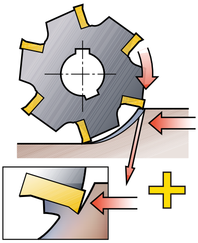
• Use positive geometry indexable inserts whenever possible for smooth cutting action and lowest power consumption.



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.

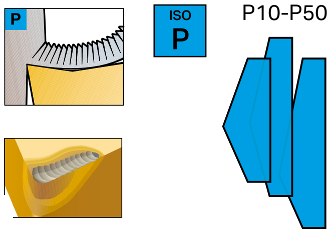


Cutting direction

• Use climb (down) milling whenever possible.

Component consideration

• Work piece material and configuration. Also quality demands on the surface to be machined.

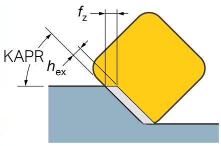


Choice of insert grade

- 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.

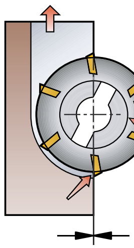


Entering angle

- Select the most suitable entering angle.

Cutter diameter

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

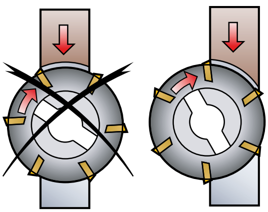


Cutter position

- 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.

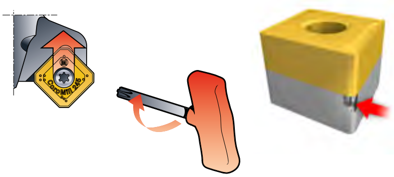


Coolant

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

Maintenance

- Follow tool maintenance recommendations and monitor tool wear.



Drilling

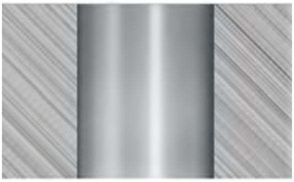
Drilling covers methods of making cylindrical holes in a workpiece with metal cutting tools

The drilling process

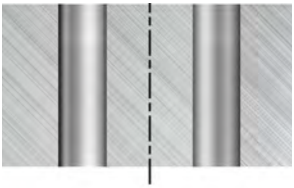
- The drill is always engulfed in the work-piece, leaving no view of the operation.
- Chips must be controlled.
- Chip evacuation is essential; it affects hole quality, tool life and reliability.

Four common drilling methods

Drilling



Trepanning

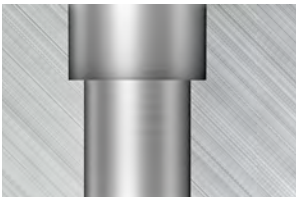


Drilling is classified into four common methods:

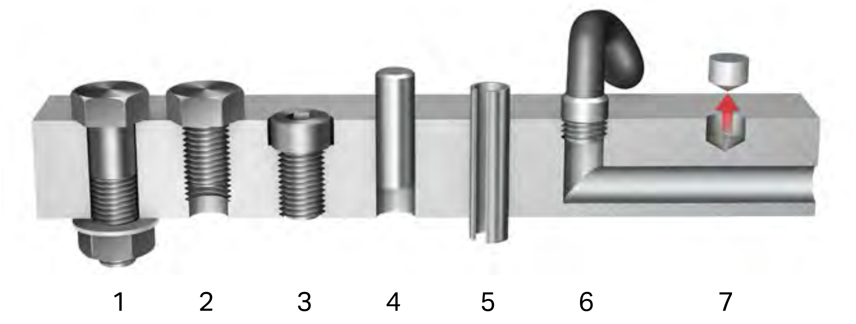
- Drilling
- Trepanning
- Chamfer drilling
- Step drilling.

Chamfer drilling

Step drilling



The most common holes



The most common holes are:

1 Holes with clearance for bolts

2 Holes with a screw thread

3 Countersink holes

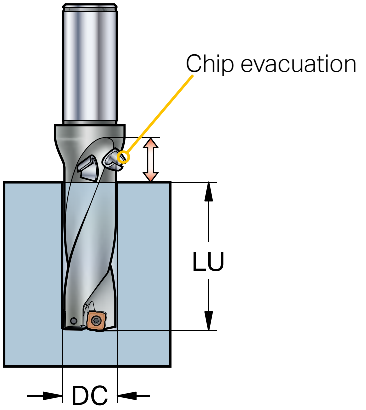
4 Pressed fit holes

5 Slip fit holes

6 Holes that form channels

7 Holes to remove weight for balancing.

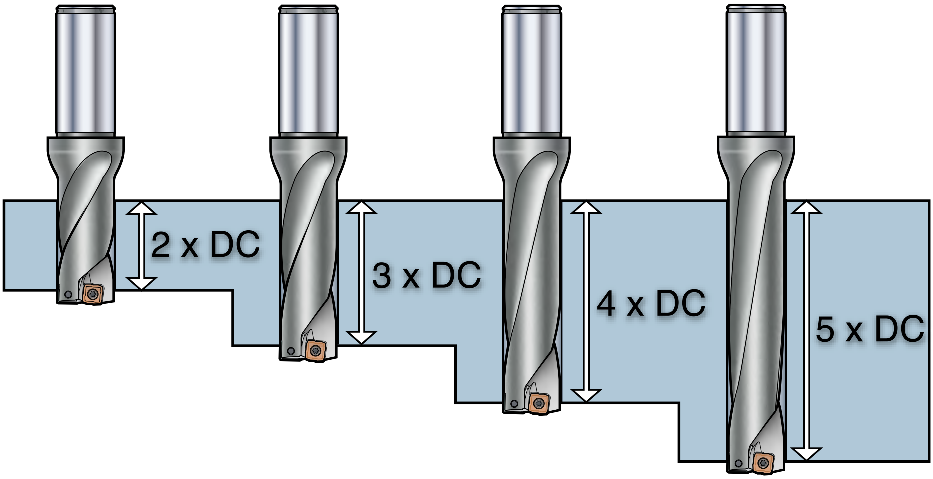
Maximum hole depth



Hole depth (LU) determines the choice of tool.

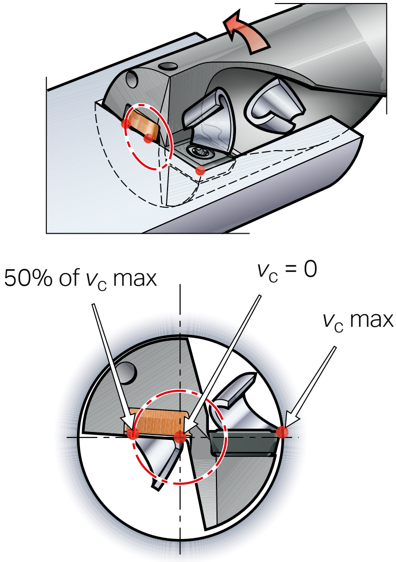
Maximum hole depth is a function of hole diameter DC and hole depth (LU).

Example: max hole depth LU = 3 x DC.



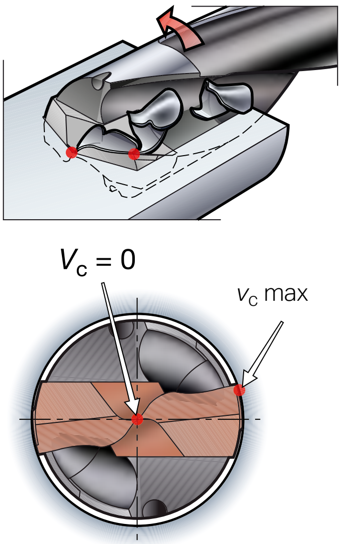
Drilling theory

Cutting speeds for indexable drills



- Cutting speed (v_c) for indexable drills declines from 100% at the periphery to zero at the center.
- The central insert operates from cutting speed zero to approx. 50% of v_c max. The peripheral insert works from 50% of v_c max up to 100% of v_c max.
- One effective cutting edge/rev: = z_c.

Cutting speeds for solid and exchangeable tip drills

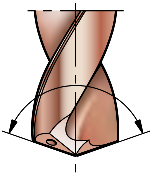


- Two effective cutting edges, from the center to the periphery.
- Two edges/rev: = z_c.

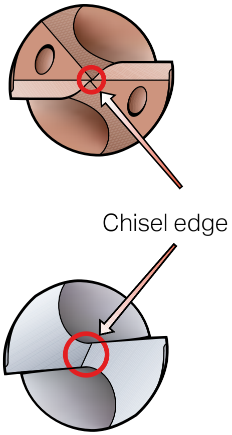
Solid carbide drill (SCD) vs. high speed drills (HSS)

Point angle and chisel edge

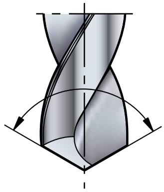
Solid carbide drill



140º point angle



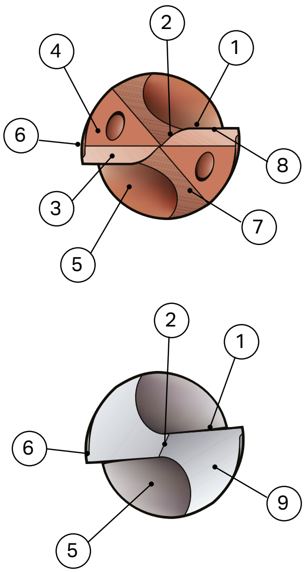
HSS drill



118º point angle

- Chisel edge is practically eliminated with the solid carbide drill.
- The axial cutting force is reduced considerably, because the chisel edge is eliminated on solid carbide drills.
- This results in better centering features and cuts chips close to the center of the drill point. This eliminates the need for a center drill.

1 Main cutting edge
2 Chisel edge
3 Primary clearance
4 Secondary clearance
5 Flute
6 Margin
7 First split
8 Negative chamfer
9 Clearance surface.

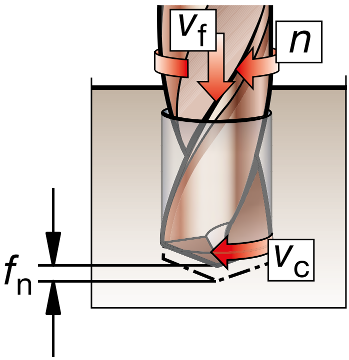


Solid carbide drill - Advantages

- Chisel edge is practically eliminated
- The main cutting edge reaches the center point
- Gives longer life and productivity
- Lower thrust and torque
- Better tolerances.

Definitions of terms

Cutting speed



Productivity in drilling is strongly related to the penetration rate, v_f.

n = spindle speed (rpm)
v_c = cutting speed m/min (ft/min)
f_n = feed per revolution mm/r (inch/r)
v_f = penetration rate mm/min (inch/min)
DC = drill diameter mm (inch)

Metric

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

Inch

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

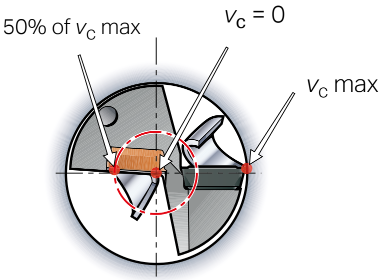
$$v_f = f_n \times n \quad \text{mm/min (inch/min)}$$

Cutting speeds for indexable drills

Cutting speed (v_c) for indexable drills declines from 100 % at the periphery to zero at the center.

The central insert operates from cutting speed zero to approx. 50% of v_c max.
The peripheral insert works from 50% of v_c max up to 100% of v_c max.

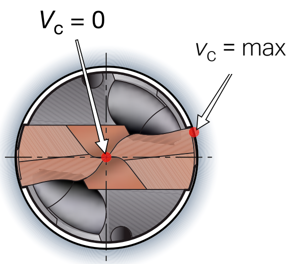
One effective cutting edge/rev: = z_c.



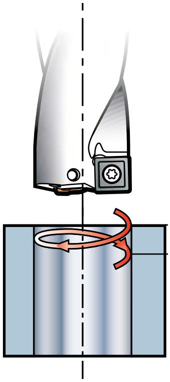
Cutting speeds for solid and exchangeable tip drills

Two edges, from the center to the periphery.

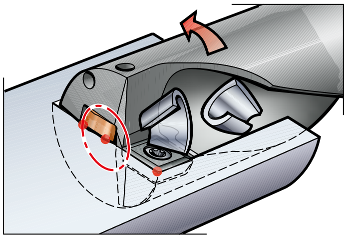
Two edges/rev: = z_c.



Effects of cutting speed – v_c m/min (ft/min)



- Affects the power P_c kW (Hp) and torque M_c Nm (lbf-ft).
- The largest factor determining tool life.
- Higher speed generates higher temperature and increased flank wear, especially on the peripheral corner.
- Higher speed is beneficial for chip formation in long chipping, soft materials, i.e., low carbon steel.
- Affects sound levels.

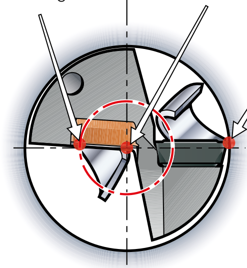


Too high cutting speed causes:

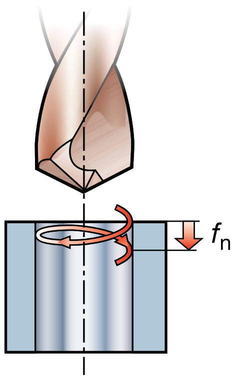
- rapid flank wear
- plastic deformation
- poor hole quality
- bad hole tolerance.

Too low cutting speed causes:

- built-up edge
- bad chip evacuation
- longer time in cut
- higher risk of drill breakage
- reduced hole quality.



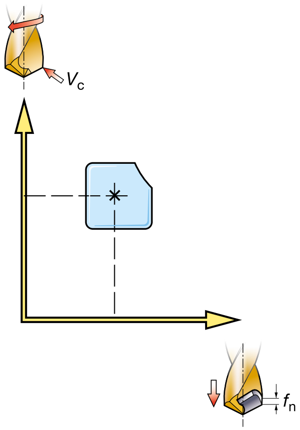
Feed rate



Effects of feed rate – f_n mm/r (inch/r)

- Affects the feed force F_f (N), power P_c kW (Hp) and torque M_c Nm (lbf-ft).
- Controls chip formation.
- Contributes to hole quality.
- Primarily influences surface finish.
- Contributes to mechanical and thermal stress.

$$f_n = f_z \times 2 \quad \text{mm/r (inch/r)}$$



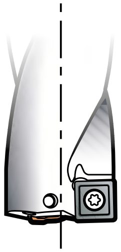
High feed rate:
- harder chip breaking
- reduced time in cut.

Low feed rate:
- longer, thinner chips
- quality improvement
- accelerated tool wear
- longer time in cut.

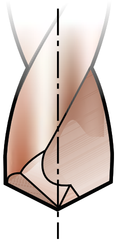
**Note:* Feed rate must correlate with cutting speed.

Approximate calculation of power consumption

CoroDrill® 880



CoroDrill® Delta-C



n = spindle speed (rpm)

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

f_n = feed per revolution mm/rev (inch/rev)

v_f = penetration rate mm/min (inch/min)

DC = drill diameter mm (inch)

f_z = feed per edge mm (inch)

k_{c1} = specific cutting force N/mm2 (lbf ft/inch2)

P_c = power consumption kW (Hp)

F_f = feed force (N)

M_c = torque Nm (lbf ft)

Metric

$$P_c = \frac{f_n \times v_c \times DC \times k_c}{240 \times 10^3} \text{ kW}$$

Inch

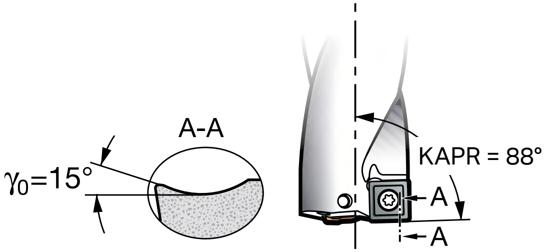
$$P_c = \frac{f_n \times v_c \times DC \times k_c}{132 \times 10^3} \text{ Hp}$$

ISO P MC No.	CMC No.	Material	Specific cutting force k_{c1} 1.0 N/mm^2	Specific cutting force k_{c1} .0394 lbs/in^2	Hardness Brinell HB	mc
		Steel Unalloyed				
P1.1.Z.AN	01.1	C = 0.1-0.25%	1500	216.500	125	0.25
P1.2.Z.AN	01.2	C= 0.25-0.55%	1600	233.000	150	0.25
P1.3.Z.AN	01.3	C = 0.55-0.80%	1700	247.000	170	0.25
P1.3.Z.AN	01.4	High carbon steel, annealed	1800	260.500	210	0.25
P1.3.Z.HT	01.5	Hardened and tempered	2000	291.500	300	0.25
		Low alloyed (alloying elements ≤ 5%)				
P2.1.Z.AN	02.1	Non-hardened	1700	246.500	175	0.25
P2.5.Z.HT	02.2	Hardened and tempered	1900	278.500	300	0.25

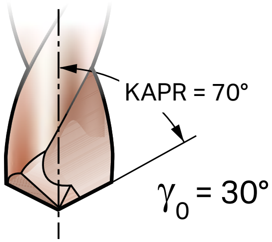
For information about the k_{c1} value, see page H16.

Accurate calculation of power consumption

CoroDrill® 880



CoroDrill® Delta-C



Metric

$$P_c = \frac{f_n \times v_c \times DC \times k_c}{240 \times 10^3} \text{ kW}$$

Inch

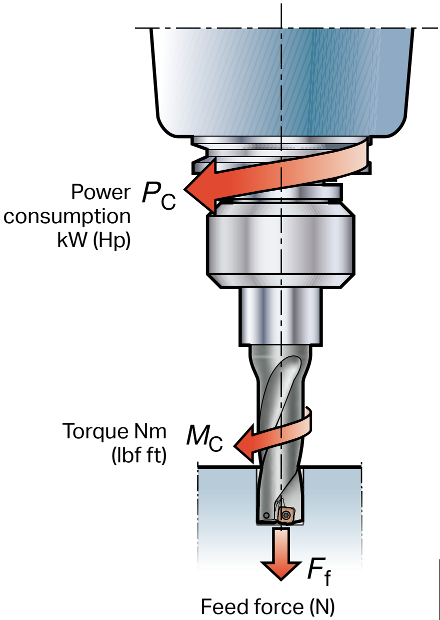
$$P_c = \frac{f_n \times v_c \times DC \times k_c}{132 \times 10^3} \text{ Hp}$$

$$k_c = k_{c1} \times (f_z \times \sin \text{KAPR})^{-m_c} \times \left(1 - \frac{\gamma_0}{100}\right)$$

ISO P MC No.	CMC No.	Material	Specific cutting force k_{c1} 1.0 N/mm^2	Specific cutting force k_{c1} .0394 lbs/in^2	Hardness Brinell HB	mc
		Steel Unalloyed				
P1.1.Z.AN	01.1	C = 0.1-0.25%	1500	216.500	125	0.25
P1.2.Z.AN	01.2	C= 0.25-0.55%	1600	233.000	150	0.25
P1.3.Z.AN	01.3	C = 0.55-0.80%	1700	247.000	170	0.25
P1.3.Z.AN	01.4	High carbon steel, annealed	1800	260.500	210	0.25
P1.3.Z.HT	01.5	Hardened and tempered	2000	291.500	300	0.25
		Low alloyed (alloying elements ≤ 5%)				
P2.1.Z.AN	02.1	Non-hardened	1700	246.500	175	0.25
P2.5.Z.HT	02.2	Hardened and tempered	1900	278.500	300	0.25

For information about the k_{c1} value, see page H16.

Calculation of torque and feed force



n = Spindle speed (rpm)

f_n = Feed per revolution mm/rev (inch/rev)

DC = Drill diameter mm (inch)

k_{c1} = Specific cutting force N/mm2 (lbf ft/inch2)

F_f = Feed force (N)

M_c = Torque Nm (lbf ft)

$$F_f \approx 0.5 \times k_c \times \frac{DC}{2}\, f_n \times \sin KAPR \text{ (N)}$$

Metric

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

Inch

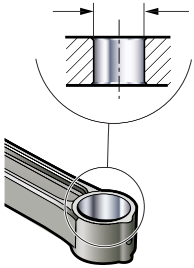
$$M_c = \frac{P_c \times 16501}{\pi \times n} \text{ (lbf ft)}$$

Tool selection procedure

Production planning process

1 **Component**

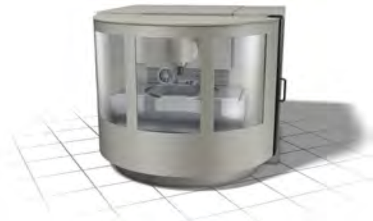
Hole dimension and quality



Workpiece material, shape and quantity

2 **Machine**

Machine parameters

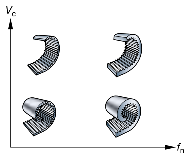


3 **Choice of tool**

Type of tool and machining method

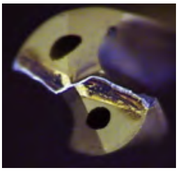
4 **How to apply**

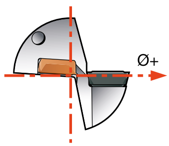
Cutting data, coolant, etc.



5 **Troubleshooting**

Remedies and solutions

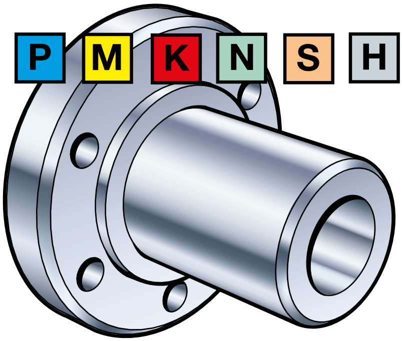




1. Component and the workpiece material

Material:

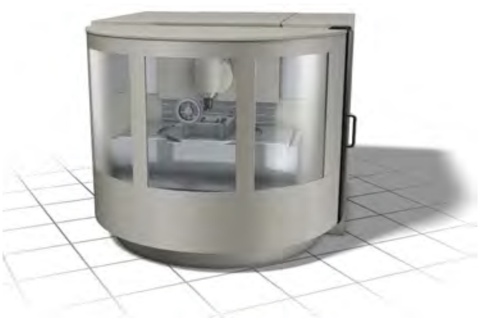
- Machinability
- Chip breaking
- Hardness
- Alloy elements.



Component:

- Is the component rotation symmetric? Use a rotating or stationary drill?
- Clamping, hole size and depth. Also is the component sensitive to feed force and/or vibrations?
- Is a tool extension needed to reach the surface where the hole will be drilled i.e. long tool overhangs?
- Component features, does something complicate the process? Are there inclined, concave or convex surfaces? Crossing holes?

2. Important machine considerations



Condition of the machine:

- Machine stability
- Spindle speed
- Coolant supply
- Coolant flow and pressure
- Clamping of the workpiece
- Horizontal or vertical spindle
- Power and torque
- Tool magazine.

3. Choice of drilling tools

Different ways to make a hole

The basic parameters are:

- Diameter
- Depth
- Quality (tolerance, surface finish, straightness).

The hole type, and the required precision affect tool choice.

Drilling can be affected by irregular or angled entry/exit surfaces and by cross holes.

Drilling

Advantages

- Simple standard tools
- Relatively flexible.

Disadvantages

- Two tools, adapters and basic holders
- Requires an extra tool and operation if it is a step/ chamfer hole
- Depending on choice
  - Productivity
  - Hole quality.

Step/chamfer drilling

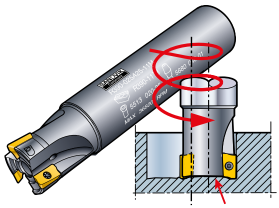
Advantages

- Reduces the number of operations
- Fastest way to make a step/chamfer hole.

Disadvantages

- Requires more power and stability
- Less flexibility.

Helical interpolation



Advantages

- Simple standard tools
- Very flexible
- Low cutting forces.

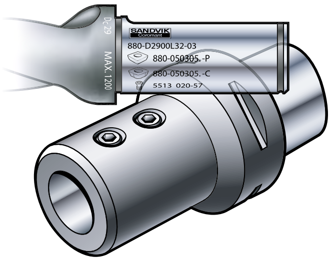
Disadvantages

- Longer cycle times.

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

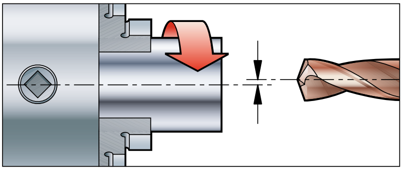
4. How to apply

Important application considerations



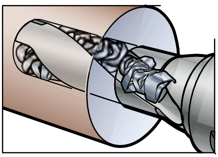
Tool holding

- Always use shortest possible drill and overhang to reduce tool deflection and vibrations, keeping in mind proper chip evacuation.
- For best stability and hole quality, use modular tools, hydro-mechanical or hydraulic holding tools.



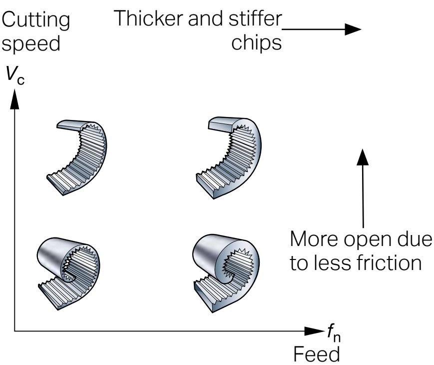
Tool runout

- Minimum tool runout is essential for successful drilling.



Chip evacuation and cutting fluid

- Chip formation and evacuation is the dominant factor in drilling and affects hole quality.



Grade and geometry

- Use recommended grade and geometry.
- Use recommended cutting parameters.
- To ensure a stable process, make sure to achieve good chip formation by adjusting cutting parameters.

5. Troubleshooting

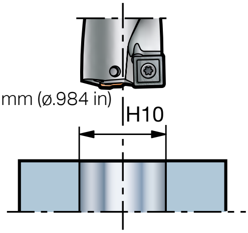
Some areas to consider

Insert wear and tool life

- Check wear pattern and if necessary adjust cutting data accordingly or change grade.

Chip evacuation

- Check chip breaking and cutting fluid supply, if necessary change chip breaker and/or cutting parameters accordingly.



Hole quality and tolerances

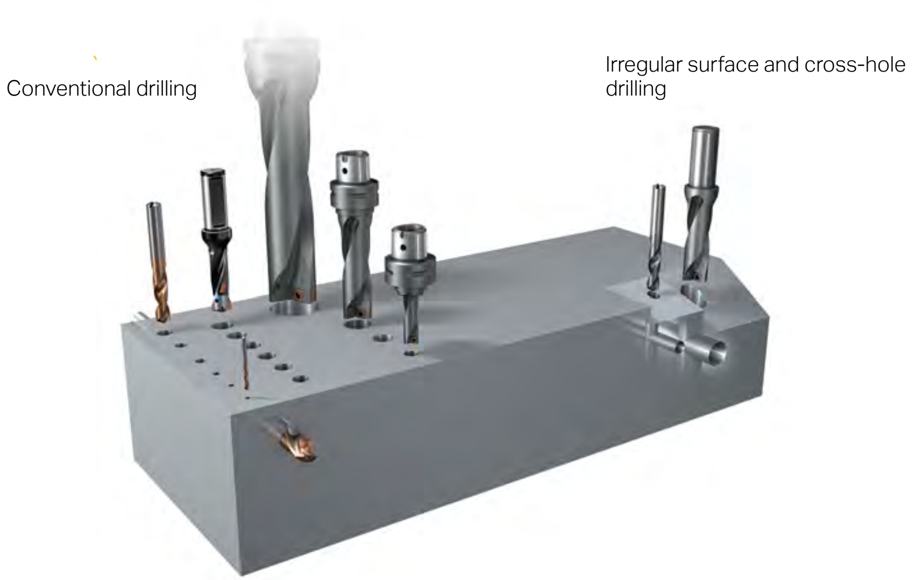
- Check clamping of drill/workpiece, feed rate, machine conditions and chip evacuation.

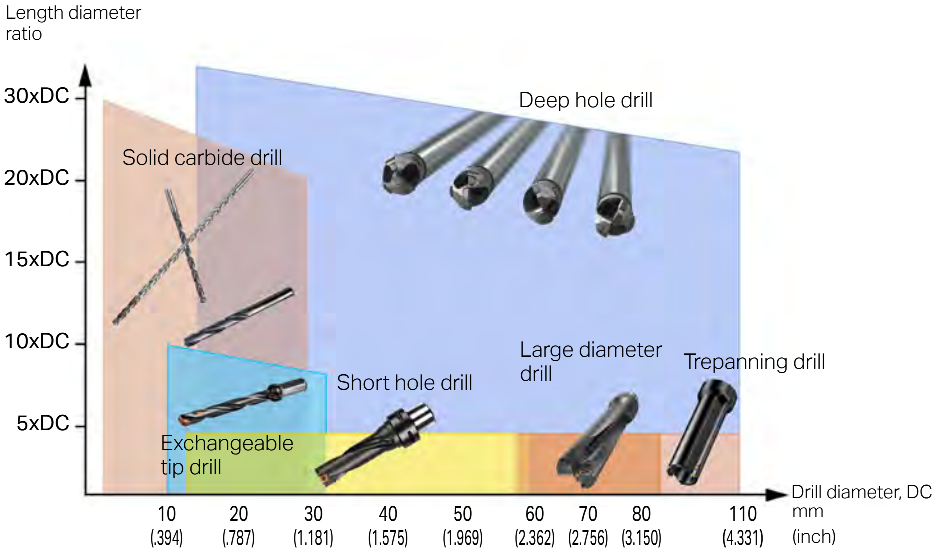
Cutting data

- Correct cutting speed and feed rate is essential for high productivity and tool life.

Drilling tools

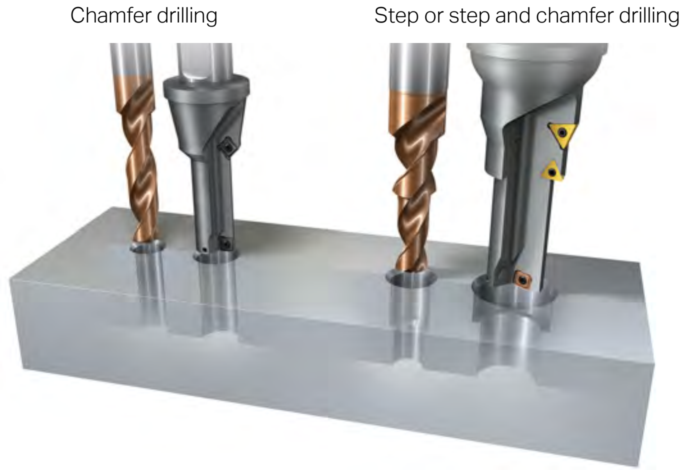
Drilling tools covering diameters from 0.30 mm up to 110 mm (.0118 inch up to 4.331 inch) and even larger as engineered products.



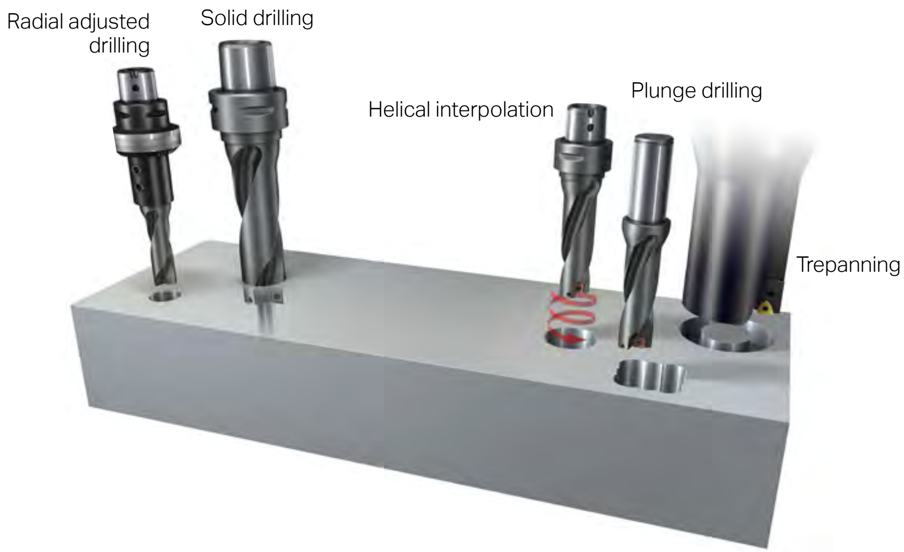


Choice of drilling tools

Step and chamfer drilling



Other methods



Diameter and hole depth

Positioning of short hole drills

Indexable insert drills

Always to be considered as the first choice due to lower cost per hole. They are also very versatile tools.

Application areas

- Medium and large diameter holes
- Medium tolerance demands
- Blind holes requiring a "flat" bottom
- Plunge drilling or boring operations.

Solid carbide drills

First choice for smaller diameters and when closer hole tolerance is required.

- Small diameter
- Close or precision tolerance holes
- Short to relatively deep holes.

Exchangeable tip drills

First choice for medium diameter holes where the exchangeable tip provides for an economical solution.

- Medium diameter
- Close hole tolerances
- Steel body provide toughness
- Short to relatively deep holes.

Indexable insert drills

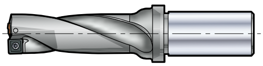
The basic drill

- The most economical way to produce a hole.
- For all workpiece materials.
- Standard, Tailor Made and special drills available.
- A versatile tool that can do more than just drilling.

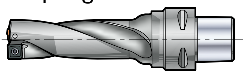
Mounting options

Different mounting options are available, which enables the user to mount the drill to almost all machine configurations. Today, machine tool manufacturers are offering mounting options integrated to the spindle.

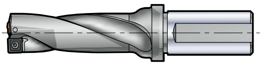
Cylindrical shank



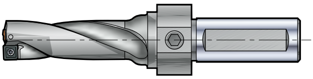
Coromant Capto® coupling



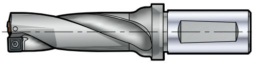
Cylindrical with flat



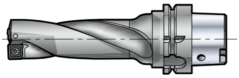
P-shank



Whistle Notch



Other modular systems



Solid carbide drills

The basic choice

P M K
N S H

Material-optimized drills

P M K N S

Application-optimized drills

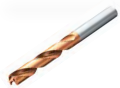
Chamfer drill

P M K
N S H

Precision drill for hard steel

P
H

Short hole drills – ISO material groups

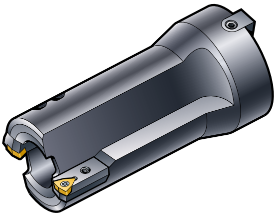
ISO material group		P	M	K	N	S	H
	Solid carbide drills	+++	+++	+++	+++	+++	+++
	Exchangeable tip drills	+++	+++	+++	++	++	+
	Indexable insert drills	+++	+++	+++	+++	+++	+++

Large hole diameters

Large diameter drill

Indexable insert drills are available in diameters up to 84 mm (3.307 inch).

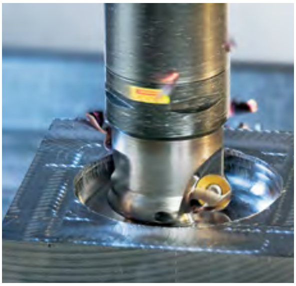
Trepanning drill



Trepanning is used for larger hole diameters and where machine power is limited, because it is not as power consuming as solid drilling. Trepanning drills are available up to diameter 110 mm (4.331 inch).

Note: These drills are for a through hole application only.

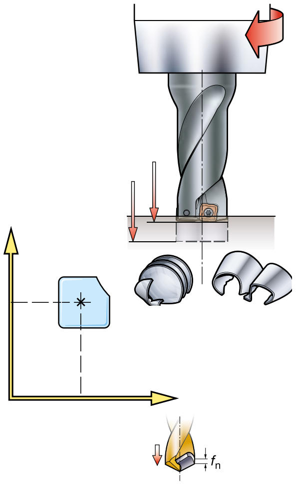
Milling, helical interpolation



A milling cutter with helical or circular interpolation can be used instead of drills or boring tools. The method is less productive but can be an alternative when chip breaking is a problem.

How to apply

Indexable insert drills

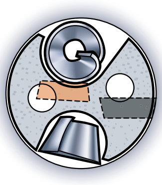


Setup routine

- Use the shortest possible drill.
- Check programming length.
- Start drilling with a mid-range recommended feed rate and cutting speed to a depth of 3.2 mm (.125 inch).
- Check chip formation and measure hole size.
- Inspect the drill to make sure no drill-to-hole rubbing is taking place.
- Increase or decrease feed rate and/or cutting speed according to chip formation, vibration, hole-surface quality, etc.

Chip formation - Indexable

- Improved chip evacuation is initially achieved by improving chip formation.
- Long chips may cause chip jamming in the drill flutes.
- Also the surface finish may be affected and the insert or tool may be at risk.
- Rectification involves selecting the correct insert geometry and adjusting cutting data.
- Apply insert geometries to suit different materials and cutting conditions.



Excellent

Acceptable

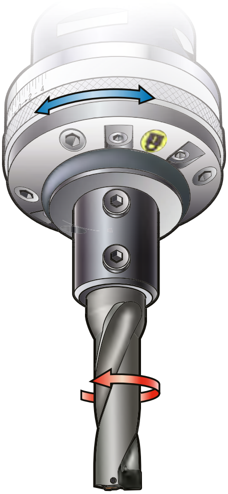
Not acceptable

Rotating indexable drill

Alignment

- If over- or under-sized holes are produced or if the center insert tends to chip, it is often because the drill is off center.
- Turning the drill 180° in its holder may solve this problem.
- But it is important to ensure that the center axis of the drill and the axis of rotation are parallel in order to achieve accurate holes.
- The machine spindle and the holder must be in good condition.

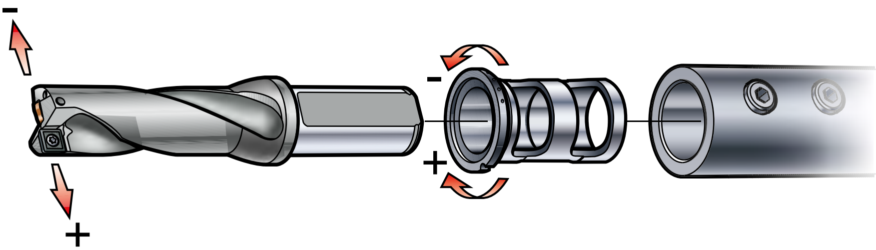
Radial adjustment

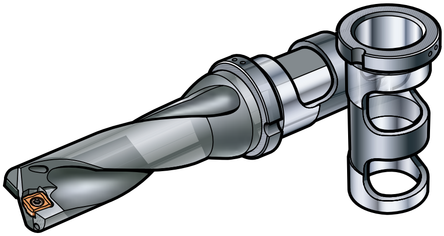


Adjustable holder

- Setting is achieved by turning the scale ring surrounding the holder, marked in increments of 0.05 mm (.002 inch), indicating a diametrical movement of the tool.
- Radial adjustment -0.2 /+0.7 mm (-.008 /+.028 inch). Note that the adjustment range for the drill should not be exceeded. (Maximum adjustment can be seen on the ordering pages in the catalog).
- It may be necessary to reduce the feed/rev (f_n) due to longer tool overhang and less balanced cutting forces created by the offsetting.
- Sleeves are used to adapt various ISO shank sizes for one holder.

Adjustable sleeve for drills with ISO 9766 shanks





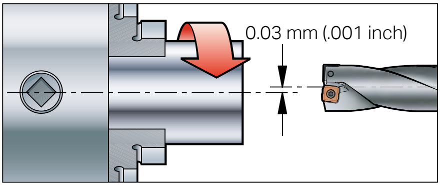
Rotating drill – eccentric sleeve

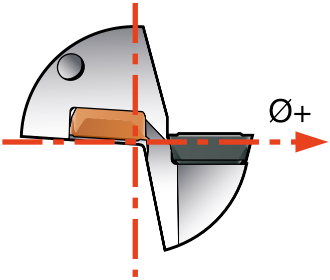
Drill diameter can be adjusted for closer hole tolerance. The adjustment range is approx. ±0.3 mm (±.012), but adjustment in the negative direction should be made only if the drill produces an oversized hole (not in order to achieve undersized holes).

- One dot increases/decreases the diameter by 0.10 mm (.004 inch).
- Increase the diameter by turning the sleeve clockwise.
- Decrease the diameter by turning the sleeve counterclockwise.
- Use both screws to clamp the drill in the fixture and make sure the bolts in the holder are long enough.

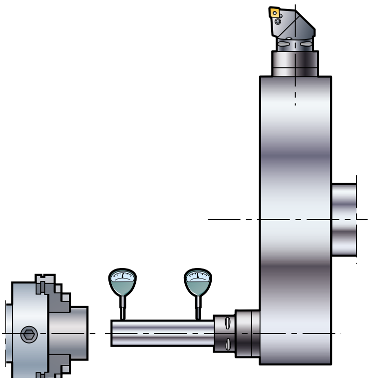
Non-rotating drill

Alignment





- The total runout between the center line of the machine and the workpiece must not exceed 0.03 mm (.001 inch) .
- The drill should be mounted so that the top face of the peripheral insert is parallel to the machine's transverse movement (usually X-axis).



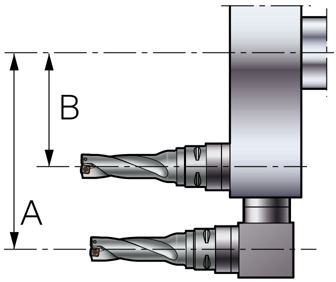
Dial indicator and test bar

- Misalignment also has the effect of radial offsetting, which produces either an over- or under-sized hole.
- Testing can be carried out with a dial indicator together with a test bar.

Drill with four flats

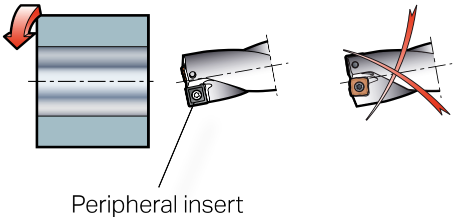
- Another way is by making a drill with four flats equally positioned around the drill shank.
- Make holes with the drill mounted in each of the four flat positions. Hole measurement will indicate the state of machine alignment.

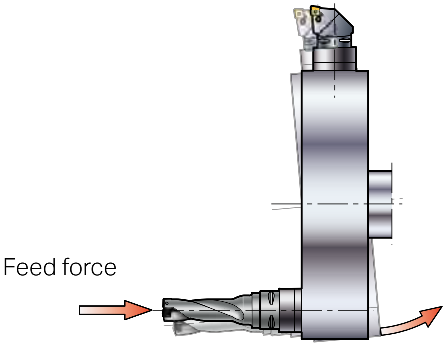
Deflection of turret



Problem solving

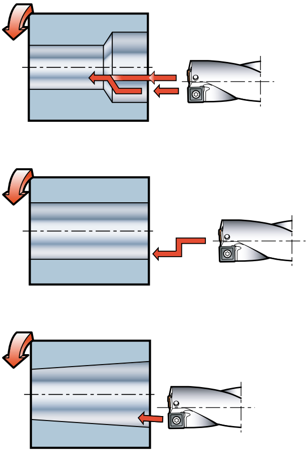
- Deflection of the turret on a CNC lathe can be caused by the feed force.
- First, check if you can minimize torque by mounting the tool differently.
  Position B is preferable to position A.





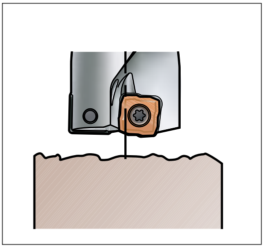
- To avoid wear on the drill body and retraction marks in the hole, mount the drill with the peripheral insert as shown in the picture.
- Finally, a reduction of the feed/revolution (f_n) can be made to minimize the feed force.

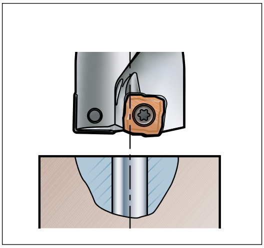
Radial offset



- Holes can be drilled larger than the nominal size of the drill as well as enlarged and finished with a subsequent boring pass.
- Non-rotating indexable insert drills can also be used to generate tapered holes.
- Also chamfering and reliefs can be machined with the drill.
- A hole which is to be threaded can be prepared in one pass along with chamfering.

Irregular surfaces and pre-drilled holes

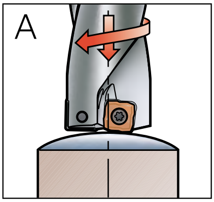




When entering or exiting an irregular surface there is a risk of the inserts chipping.

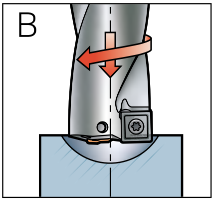
- The feed rate should therefore be reduced.
- A pre-drilled hole should be small rather than large - not more than 25% of the drill diameter - to avoid drill deflection.
- However, reduced feed does allow broad machining of pre-drilled holes.

Entering non-flat surfaces



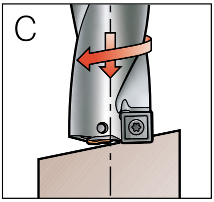
Convex surface

- Normally no feed reduction needed.



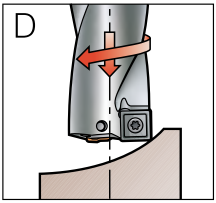
Concave surface

- Reduce feed to 1/3 of original feed rate.



Inclined surface

- With entering angle of 2º–89º, reduce feed to 1/3 of original feed rate.



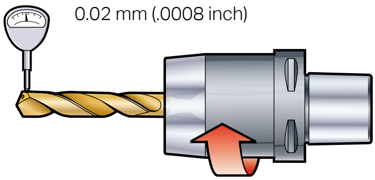
Irregular surfaces

- Reduce feed 1/3 of original feed rate.

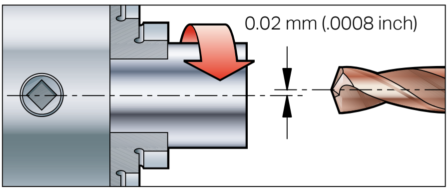
Solid carbide and exchangeable tip drills

Alignment

Rotating drill



Stationary drill

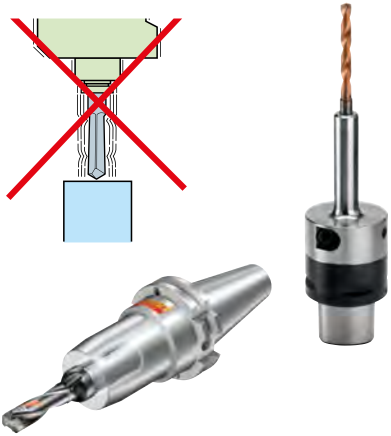


Minimum tool runout is one of the main criteria for successful use of solid carbide drills.

The runout should not exceed 0.02 mm (.0008 inch) in order to achieve:

- close hole tolerance
- good surface finish
- long and consistent tool life.

Tool holding



- A collet and tool shank in bad condition will ruin an otherwise perfect setup.
- Make sure that the TIR (Total Indicator Readout) is within 0.02 mm (.0008 inch).
- An unacceptable runout can be temporarily reduced by turning the drill or the collet 90° or 180° to find lowest TIR.

For best performance use hydro-mechanical, hydraulic or shrink fit chuck.

Solid carbide and exchangeable tip drills

Solid carbide drills

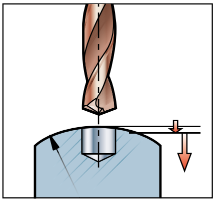
- Not recommended due to risk of chipping on cutting edge.

Exchangeable-tip drills

- Not possible to enlarge existing holes by counter-boring because no chip breaking will take place.

Entering non-flat surfaces

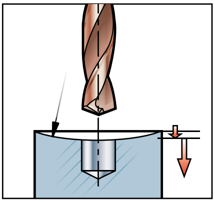
When entering non-flat surfaces there is a risk of drill deflection. To avoid this, the feed can be reduced when entering.



Convex surface

Drill if radius is > 4 times drill diameter and the hole is perpendicular to the radius.

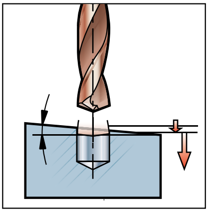
Reduce feed 50% of normal rate during entrance.



Concave surface

Drill if radius is > 15 times drill diameter and the hole is perpendicular to the radius.

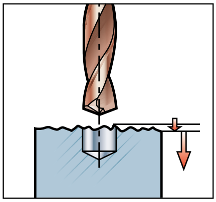
Reduce feed 25% of normal rate during entrance.



Inclined surface

Inclinations up to 10º, reduce the feed to 1/3 of normal feed rate during entrance. More than 10º, not recommended.

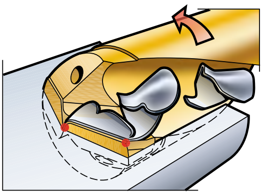
Mill a small flat on surface, then drill the hole.

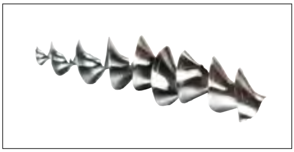


Irregular surfaces

Reduce feed rate to ¼ of normal rate to avoid chipping on the cutting edges.

Chip formation – Solid carbide and exchangeable tip drills

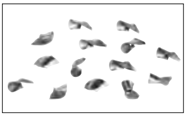




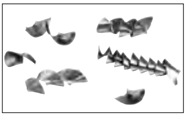
Start chip

Note: The start chip from entry into the workpiece is always long and does not create any problems.

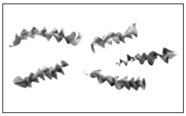
- Improved chip evacuation is initially achieved by improving chip formation.
- Long chips may cause chip jamming in the drill flutes.
- Also the surface finish may be affected and the insert or tool may be at risk.
- Make sure the right cutting data and drill/tip geometry is used to suit different materials and cutting conditions.



Excellent



Acceptable



Chip jamming

Coolant supply

Internal coolant supply

- Always to be preferred especially in long- chipping materials and when drilling deeper holes (4-5 x DC).

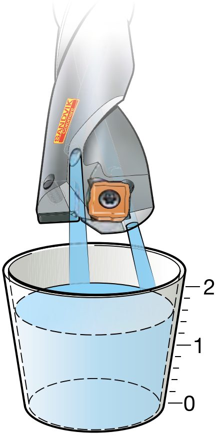
External coolant supply

- Can be used when chip formation is good and when the hole depth is shallow.

Compressed air, minimal lubrication or dry drilling

- Can be successful in favorable conditions, but is generally not recommended.

The cutting fluid



Soluble oil (emulsion)

- 5 to 12% oil (10-25% for stainless steels).
- EP (extreme pressure) additives.

Neat oil

- always with EP additives.
- increases tool life in ISO-M and ISO-S applications
- both solid carbide and indexable insert drills work well with neat oil.

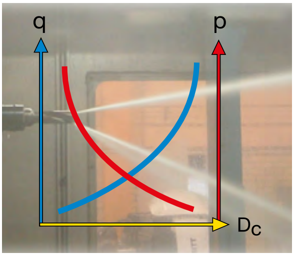
Mist cutting fluid or minimal lubrication

- can be used with good performance in materials with favorable chip forming.

Dry drilling, without any coolant

- can be performed in short-chipping materials.
- hole depths up to 3 times the diameter.
- preferably in horizontal applications.
- tool life will be influenced negatively.

Coolant – Important for successful performance

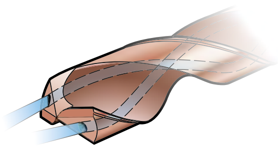


Coolant supply is essential in drilling and influences:
- chip evacuation
- hole quality
- tool life.

- The cubic capacity of the coolant tank should be between 5-10 times larger than the volume of coolant that the pump supplies per minute.
- The volume capacity can be checked using a stopwatch and a suitably-sized bucket.

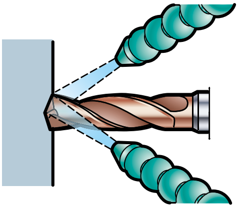
Coolant

Internal or external



Internal coolant supply

- Is always to be preferred to avoid chip jamming.
- Should always be used at hole depths above 3 times the diameter.
- A horizontal drill should have a flow of coolant coming out of the drill without any downward drop for at least 30 cm (12").



External coolant supply

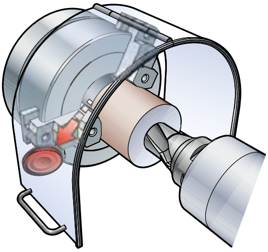
- Can be acceptable in short-chipping materials.
- To improve chip evacuation at least one coolant nozzle (two if drill is stationary) should be directed close to the tool axis.
- Can sometimes help to avoid built-up edge formation due to a higher edge temperature.

Compressed air, minimal lubrication or dry drilling

- Can be used with an Exchangeable tip drill under favorable conditions in short chipping materials.
- Solid carbide drills work well in these types of applications.

Safety precautions

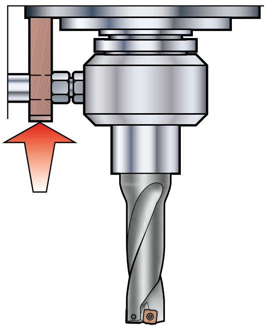
Internal coolant supply



Safety against dangerous discs

- Guarding against through-hole discs is important to avoid damage or injury, especially when using non-rotating drills.

External coolant supply



Rotating stop is an important measure

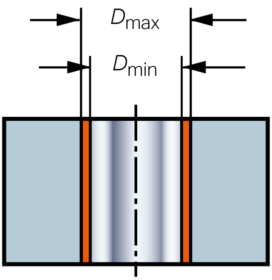
- A rotation stop may be necessary for rotating drills.
- If the coolant contains chip particles, the slit seatings may seize and as a result the housing will rotate.
- If the rotating connector has not been used for a long time, check that the holder rotates in the housing before the machine spindle is started.

Hole quality and tolerance

Steps to ensure good hole quality in drilling

- The machine tool should be in good condition.
- Tool holding influences hole quality and tool life.
- Use the shortest possible drill for maximum stability.
- Chip breaking and chip evacuation must always be satisfactory.
- Coolant supply and coolant pressure is important.

Hole and hole tolerance



Hole dimensions are characterized by three parameters:

- nominal value (the theoretical exact value)
- tolerance width (a number), e.g., IT 7 according to ISO
- position of the tolerance (designated by capital letters according to ISO).

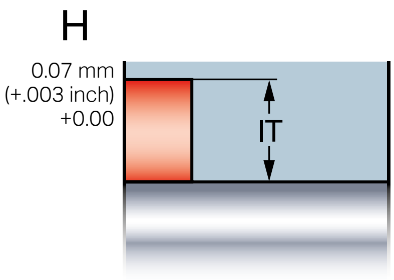
D_{max} minus D_{min} is the tolerance width, also called, e.g., IT 7.

Hole tolerance according to ISO

Tolerance	Diameter range, mm/*inch*							Examples
	3–6 *.118–.236*	6–10 *.236–.394*	10–18 *.394–.709*	18–30 *.709–1.181*	30–50 *1.181–1.969*	50–80 *1.969–3.150*	80–120 *3.150–4.724*	
IT6	0.008 *.0003*	0.009 *.0004*	0.011 *.0004*	0.013 *.0005*	0.016 *.0006*	0.019 *.0007*	0.022 *.0009*	Bearings
IT7	0.012 *.0005*	0.015 *.0006*	0.018 *.0007*	0.021 *.0008*	0.025 *.0010*	0.030 *.0012*	0.035 *.0014*	
IT8	0.018 *.0007*	0.022 *.0009*	0.027 *.0011*	0.033 *.0013*	0.039 *.0015*	0.046 *.0018*	0.054 *.0021*	1) Holes for threading
IT9	0.030 *.0012*	0.036 *.0014*	0.043 *.0017*	0.052 *.0020*	0.062 *.0002*	0.074 *.0029*	0.087 *.0034*	
IT10	0.048 *.0019*	0.058 *.0022*	0.070 *.0028*	0.084 *.0033*	0.100 *.0039*	0.120 *.0047*	0.140 *.0055*	Normal tap holes
IT11	0.075 *.0030*	0.090 *.0035*	0.110 *.0043*	0.130 *.0051*	0.160 *.0062*	0.190 *.0074*	0.220 *.0089*	
IT12	0.120 *.0047*	0.150 *.0059*	0.180 *.0071*	0.210 *.0083*	0.250 *.0098*	0.300 *.0118*	0.350 *.0138*	
IT13	0.180 *.0071*	0.220 *.0087*	0.270 *.0106*	0.330 *.0130*	0.390 *.0154*	0.460 *.0181*	0.540 *.0213*	

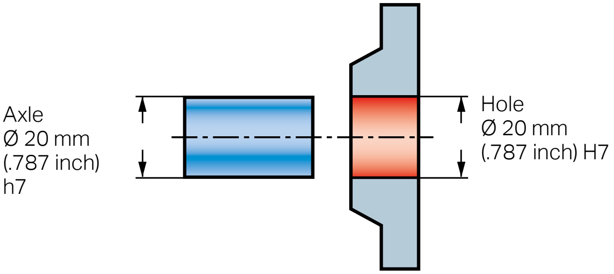
1) Holes for threading with fluteless taps (rolled threads)

- The lower the IT-number, the closer the tolerance.
- The tolerance for one IT-class grows with larger diameters.



Example:	Ø 15.00 mm (.591 inch) H10
Nominal value:	15.00 mm (.591 inch)
Tolerance width:	0.07 mm (.003 inch) (IT 10 acc. to ISO)
Position:	0 to plus (H acc. to ISO)

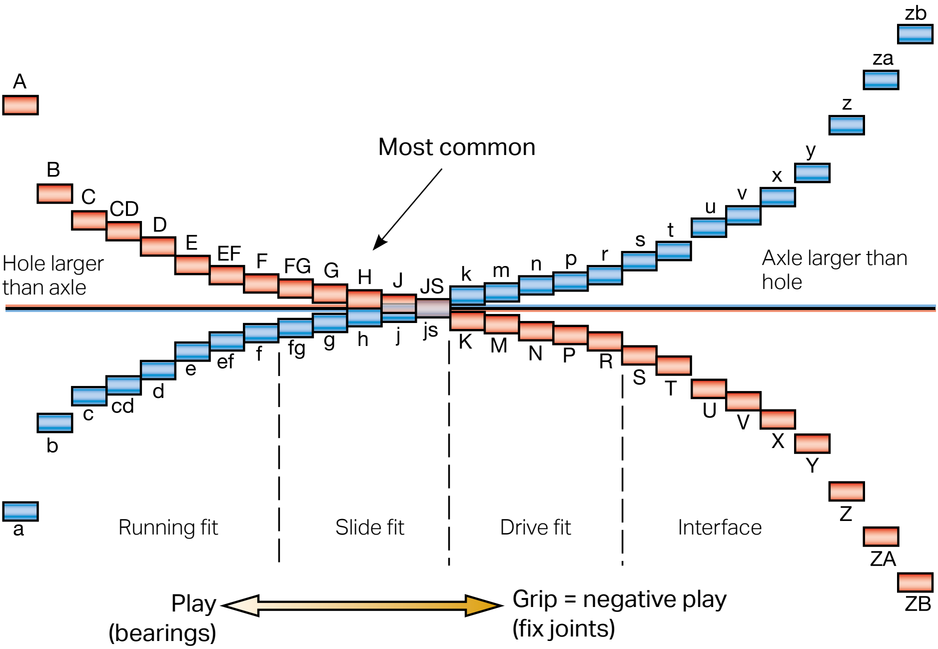
Hole tolerances according to ISO



The hole tolerance is often connected to the tolerance of an axle, that should fit the hole.

Hole and axle tolerance according to ISO

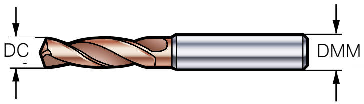
Axle tolerance position is denominated by lower case letters corresponding to the hole tolerance in upper case letters. The figure below gives a complete picture.



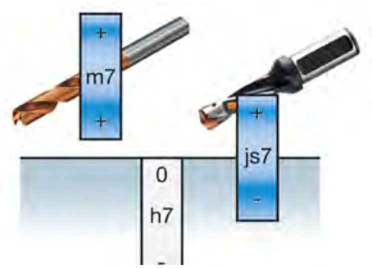
Hole and tool tolerance

Obtainable hole tolerance with different tools

Drill diameter DC tolerance



DC tolerance for a solid carbide drill and a exchangeable tip drill

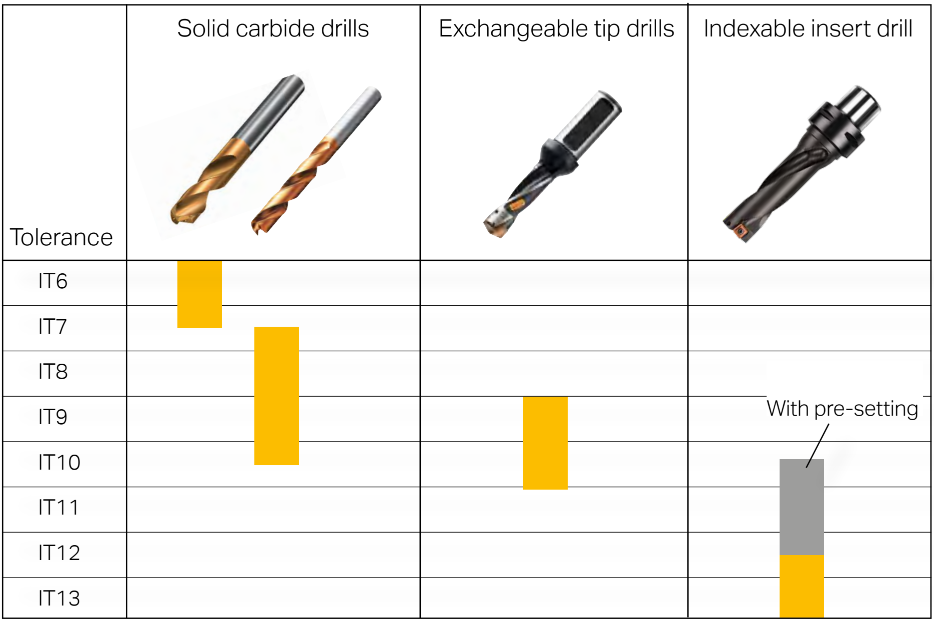


Drill tolerance

- The drill is ground to a certain diameter tolerance, designated by lower case letters according to ISO.

The hole tolerance

- For modern solid carbide or exchange-able tip drills, the hole tolerance is very close to the drill tolerance.



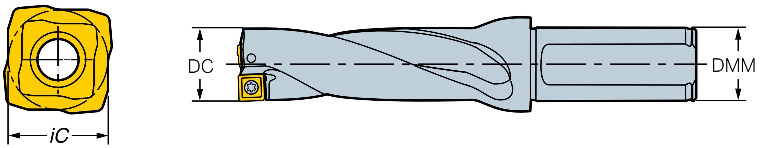
Indexable insert drills

Drill tolerance

- The diameter tolerance of an indexable insert drill is a combination of the tip seat tolerance in the drill body and the insert tolerance.

Hole tolerance

- Indexable insert drills give an optimal cutting force balance and a plus tolerance (oversized) hole, because most holes are with H-tolerance.

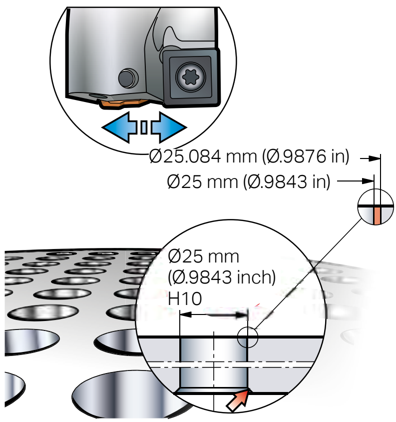


Drill depth 2-3 x DC

Drill diameter, mm (inch)	12 – 43.99 (.472 – 1.732)	44 – 52.99 (1.732 – 2.086)	53 – 63.5 (2.087 – 2.5)
Hole tolerance, mm (inch)	0/+0.25 (0/+.0098)	0/+0.28 (0/+.011)	0/+0.3 (0/+.0118)
Tolerance DC, mm (inch)	0/+0.2 (0/+.0079)	0/+0.25 (0/+.0098)	0/+0.28 (0/+.011)

Drill depth 4-5 x DC

Drill diameter, mm (inch)	12 – 43.99 (.472 – 1.732)	44 – 52.99 (1.732 – 2.086)	53 – 63.5 (2.087 – 2.5)
Hole tolerance, mm (inch)	0/+0.4 (0/+.0157)	0/+0.43 (0/+.0169)	0/+0.45 (0/+.0177)
Tolerance DC, mm (inch)	+0.04/+0.24 (+.0016/+.0094)	+0.04/+0.29 (+.0016/+.0114)	+0.04/+0.32 (+.0016/+.0126)



How to improve the hole tolerance

One way of eliminating the manufacturing tolerance of the drill body and inserts is to preset the drill.

This can be done in a lathe or with an adjustable holder/sleeve, see page E28.

A tolerance width (IT) inside 0.10 mm (.004 inch) can then be obtained.

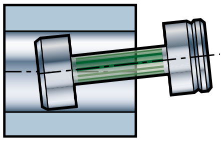
Hole size can be influenced by changing insert geometry on one of the inserts.

Troubleshooting

Indexable insert drill

Problem	Solution	

Oversized holes



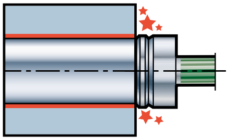
Rotating drill

1. Increase coolant flow, clean filter, clear coolant holes in drill.
2. Try a tougher geometry on peripheral side (keep center insert).

Non-rotating drill

1. Check alignment on lathe.
2. Rotate drill 180°.
3. Try a tougher geometry on peripheral side (keep center insert).

Undersized holes



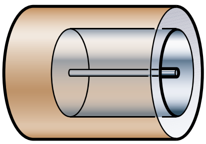
Rotating drill

1. Increase coolant flow, clean filter, clear coolant holes in drill.
2. Try a tougher geometry on center side and a light cutting geometry on peripheral side.

Non-rotating drill

1. Stationary: Check alignment on lathe.
2. Stationary: Rotate drill 180°.
3. Try a tougher geometry on center side (keep peripheral).

Pin in hole



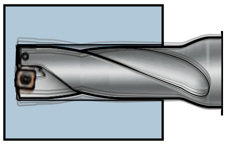
Rotating drill

1. Increase coolant flow, clean filter, clear coolant holes in drill.
2. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data.
3. Shorten drill overhang.
4. Use a lower feed rate during the first 3 mm of the hole depth.

Non-rotating drill

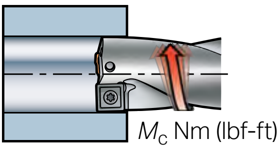
1. Check alignment on lathe.
2. Increase coolant flow, clean filter, clear coolant holes in drill.
3. Shorten drill overhang.
4. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data.

Vibrations

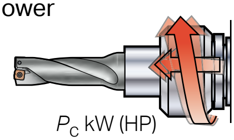
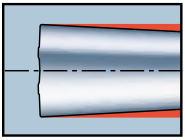
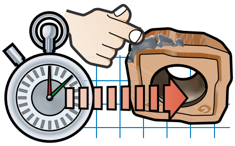
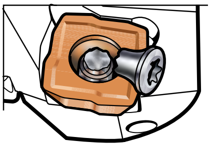
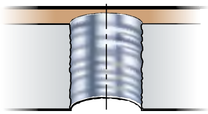
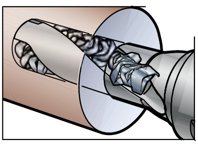


1. Shorten drill overhang, Improve the workpiece stability.
2. Reduce cutting speed.
3. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data.

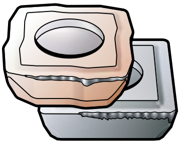
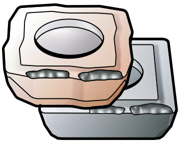
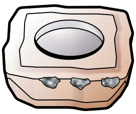
Insufficient machine torque

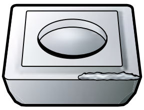


1. Reduce feed.
2. Choose a light cutting geometry to lower the cutting force.

Problem	Solution
Insufficient machine power 	1. Reduce cutting speed. 2. Reduce cutting feed. 3. Choose a light cutting geometry to lower the cutting force.
Hole not symmetrical 	Hole widens at bottom (due to chip jam on center insert) 1. Increase coolant flow, clean filter, clear coolant holes in drill. 2. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data. 3. Shorten drill overhang.
Poor tool life 	1. Adjust to higher or lower cutting speed depending on type of wear. 2. Choose a light-cutting geometry to lower the cutting force. 3. Increase feed
Broken insert screws 	1. Use torque wrench to fasten the screw together, apply Anti-seize. 2. Check and change insert screw on a regular basis.
Bad surface finish 	1. Important to have good chip control. 2. Reduce feed (if it is important to keep v_f, increase speed as well). 3. Increase coolant flow, clean filter, clear coolant holes in drill. 4. Shorten drill overhang, improve the workpiece stability.
Chip jamming in the drill flutes 	Caused by long chips 1. Check geometry and cutting data recommendations. 2. Increase coolant flow, clean filter, clear coolant holes in drill. 3. Reduce feed within recommended cutting data. 4. Increase cutting speed within recommended cutting data.

Tool wear – Indexable insert drill

Problem	Cause	Solution
Flank wear 	a) Cutting speed too high. b) Insufficiently wear resistant grade.	a) Reduce cutting speed. b) Choose a more wear resistant grade.
Crater wear 	Peripheral insert • Diffusion wear caused by temperature too high on rake face. Central insert: • Abrasive wear caused by built-up edge and smearing.	Peripheral insert • Select a more wear resistant grade. • Reduce speed. Central insert: • Reduce feed. General: • Choose a more positive geometry i.e. -LM.
Plastic deformation (peripheral insert) 	a) Cutting temperature (cutting speed) too high, combined with high pressure (feed, hardness of workpiece). b) As a final result of excessive flank wear and/or crater wear.	a–b) Select a more wear resistant grade with better resistance to plastic deformation. a–b) Reduce cutting speed. a) Reduce feed.
Chipping 	a) Insufficent toughness of grade. b) Insert geometry too weak. c) Built-up edge (BUE). d) Irregular surface. e) Bad stability. f) Sand inclusions (cast iron).	a) Select a tougher grade. b) Select a stronger geometry. c) Increase cutting speed or select a more positive geometry. d) Reduce feed at entrance. e) Improve stability. f) Choose a stronger geometry. Reduce feed.

Problem	Cause	Solution
Built-up edge (BUE) 	a) Low cutting speed (temperature too low at the cutting edge). b) Cutting geometry too negative. c) Very sticky material, such as certain stainless steels and pure aluminum. d) Percent of oil mixture in cutting fluid too low.	a) Increase cutting speed or change to a coated grade. b) Select a more positive geometry i.e. -LM. c-d) Increase oil mixture and volume/pressure in cutting fluid.

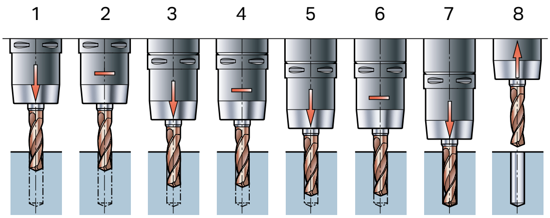
Chip evacuation - general recommendations

Checkpoints and remedies

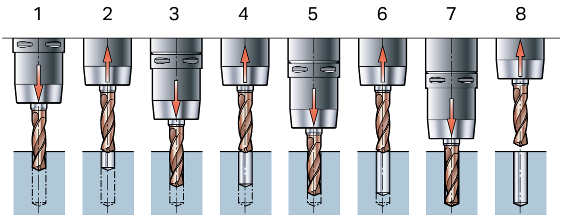
1. Make sure the right cutting data and drill geometry are used.
2. Inspect chip form (compare with picture on page E 26).
3. Check if the cutting fluid flow and pressure can be increased.
4. Inspect the cutting edges. Chipping on the edge can cause long chips because the chip is divided. Also a large Built-up-edge can cause poor chip forming.
5. Check if the machinability has changed due to a new batch of workpiece material. Cutting data may need to be adjusted.
6. Adjust feed and speed. See diagram on page E 18.

Peck drilling – solid carbide / exchangeable tip drills

Peck drilling can be used if no other solution can be found.
There are two different ways to perform a peck drilling cycle:



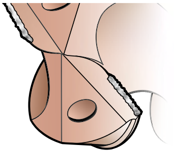
- Method 1 for best productivity
Do not retract the drill more than approx. 0.3 mm (.012 inch) from the hole bottom. Alternatively, make a periodical stop, while the drill is still rotating, before continuing to drill.



- Method 2 for best chip evacuation
After each drilling cycle, retract the drill out from the hole to ensure that no chips are stuck onto the drill.

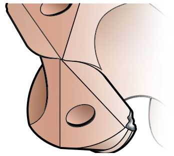
Tool wear – solid carbide / exchangeable tip drills

Built-up edge



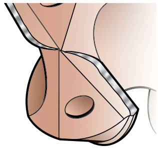
Cause	Solution
1. Cutting speed too low and edge temperature too high	1. Increase cutting speed or use external cutting fluid
2. Negative land too large	2. Sharper cutting edge
3. No coating	3. Coating on the edge
4. Percentage of oil in the cutting fluid too low	4. Increase the percentage of oil in the cutting fluid

Chipping on the edge corner



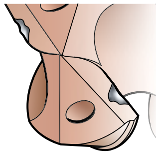
Cause	Solution
1. Unstable fixturing	1. Check fixture
2. TIR too large	2. Check radial runout
3. Intermittent cutting	3. Decrease the feed
4. Insufficient cutting fluid (thermal cracking)	4. Check cutting fluid supply
5. Unstable tool holding	5. Check the tool holder

Flank wear on the cutting edges



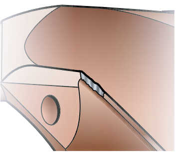
Cause	Solution
1. Cutting speed too high	1. Decrease the cutting speed
2. Feed too low	2. Increase the feed
3. Grade too soft	3. Change to harder grade
4. Lack of cutting fluid	4. Check for proper cutting fluid supply

Chipping on the cutting edge

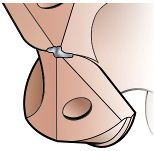
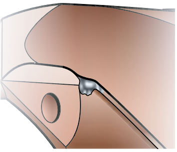
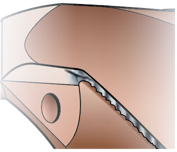


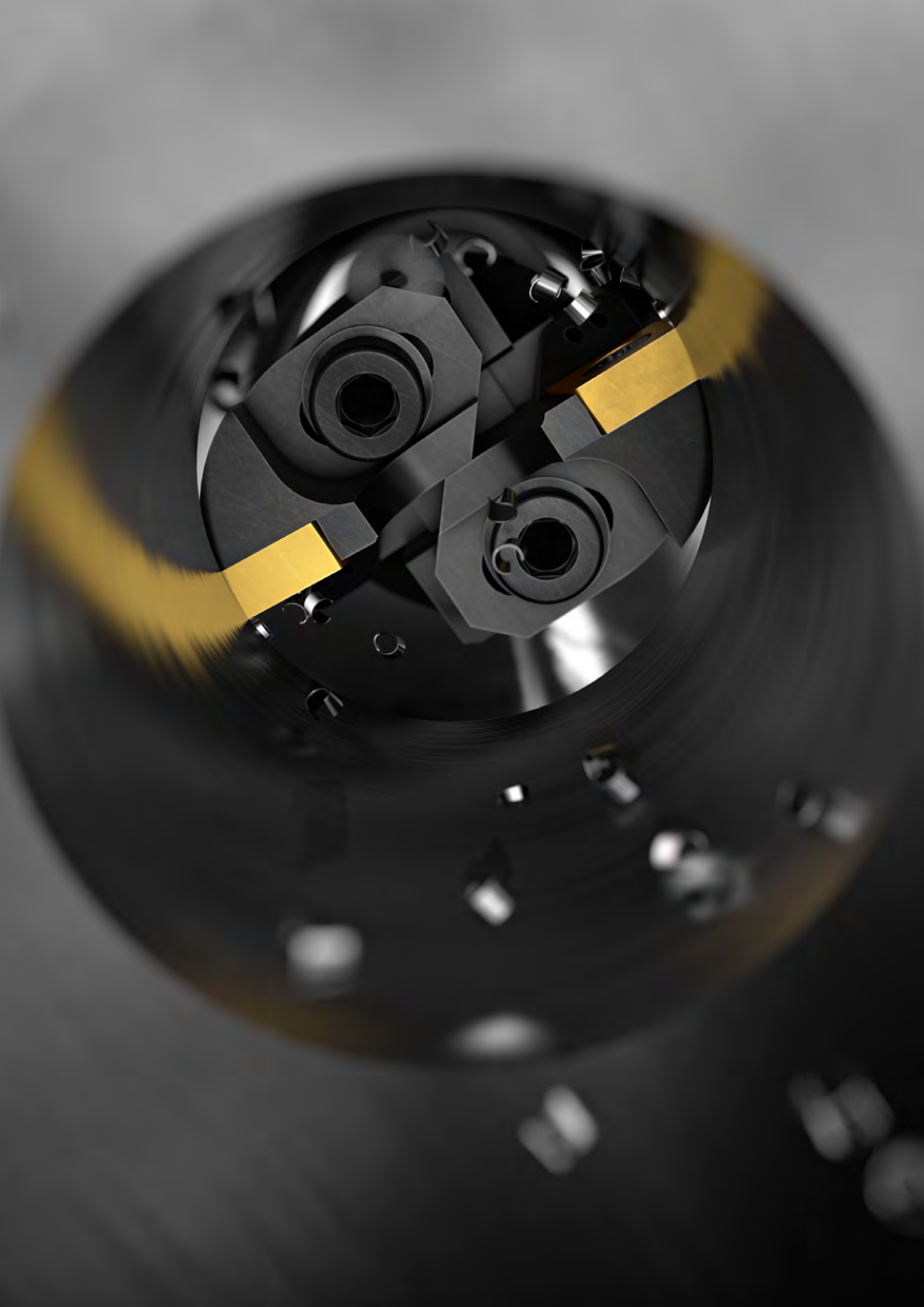
Cause	Solution
1. Unstable conditions	1. Check the setup
2. Maximum allowed wear exceeded	2. Replace drill sooner
3. Grade too hard	3. Change to softer grade

Wear on the circular lands



Cause	Solution
1. TIR too large	1. Check the radial runout
2. Cutting fluid too weak	2. Use neat oil or stronger emulsion
3. Cutting speed too high	3. Decrease cutting speed
4. Abrasive material	4. Change to harder grade

	Cause	Solution
Wear on the chisel edge		
	1. Cutting speed too low 2. Feed too high 3. Chisel edge too small	1. Increase cutting speed 2. Decrease feed 3. Check dimensions
Wear due to plastic deformation		
	1. Cutting speed and/or feed too high 2. Not enough cutting fluid supply 3. Unsuitable drill/grade	1. Decrease the cutting speed and/or feed 2. Increase cutting fluid pressure 3. Use a harder grade
Thermal cracks (notches)		
	1. Inconsistent cutting fluid	1. Check cutting fluid supply 2. Fill cutting fluid tank



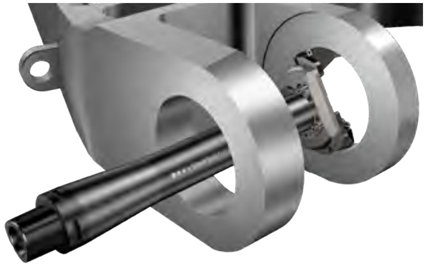
Boring

Boring operations involving rotating tools are applied to machine holes that have been made through methods such as pre-machining, casting, forging, extrusion, flame-cutting, etc.

Boring theory

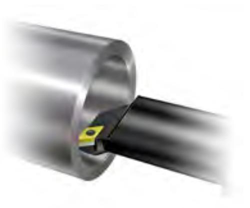
The boring process

- Typically, boring operations are per-formed in machining centers and horizontal boring machines.
- The rotating tool is fed axially through the hole.
- Most holes are through-holes, often in prismatic components such as housings and casings.



Three different basic boring methods

Boring with a stationary tool



- To be used only for symmetrical components in a turning lathe.
- Profiling can be carried out with standard boring bars.
- Very flexible tool solutions with interchangeable cutting heads.

Boring with a rotating tool

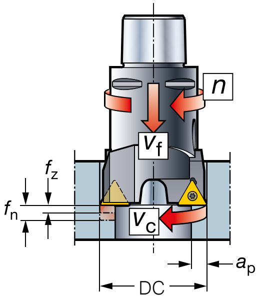
- For unsymmetrical components machined in a machining center.
- Flexible tool solutions with adjustable diameters.
- Highly productive in roughing operations.
- High quality hole tolerance and surface finish.

Milling, helical interpolation

- Very flexible solution where one milling cutter can be used for different diameters.
- Saves space in the tool magazine.
- Good solution when chip breaking is a problem.
- High quality demands of the machine (for finishing).

Definitions of terms

Definitions of cutting data terms



n	= spindle speed (rpm)
a_p	= radial depth of cut mm (inch)
v_c	= cutting speed m/min (ft/min)
f_n	= feed per revolution mm/r (inch/r)
DC	= boring diameter mm (inch)
v_f	= penetration rate mm/min (inch/min)
f_z	= feed per tooth mm/rev (inch/rev)
z_c	= effective number of teeth that machine the final surface

Cutting speed

The boring tool rotates at a certain number of revolutions (n) per minute generating a certain diameter (DC). This gives a specific cutting speed (v_c) measured in m/min (ft/min) at the cutting edge.

Metric

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

Inch

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

Feed

The axial tool movement is called feed rate (f_n) and is measured in mm/rev (inch/revolution). The feed rate is obtained by multiplying the feed per tooth, mm/rev (inch/rev), by the number of effective teeth (z_c). The feed rate is the key value in determining the quality of the surface being machined and for ensuring that the chip formation is within the scope of the insert geometry.

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

$$f_n = z_c \times f_z \quad \text{mm/r (inch/r)}$$

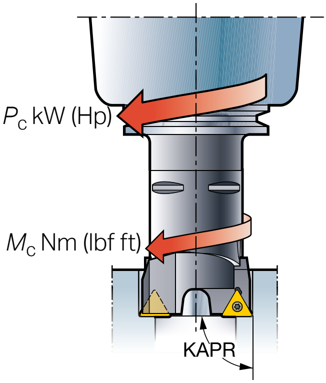
Penetration rate

The penetration rate (v_f) is the speed of the axial movement and is strongly related to productivity.

Cutting depth

The cutting depth (a_p) is the difference between the uncut and the cut hole radius.

Calculating torque and power consumption



n	= spindle speed (rpm)
v_c	= cutting speed m/min (ft/min)
f_n	= feed per revolution mm/r (inch/r)
DC	= boring diameter mm (inch)
k_c	= specific cutting force N/mm2 (lbs/inch2)
P_c	= power consumption kW (Hp)
M_c	= torque Nm (lbf ft)
KAPR	= tool cutting edge angle

Torque

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

Metric

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

Inch

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

Net power

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 mechanical and electrical efficiency of the machine must be taken into consideration when selecting cutting data.

Net power, kW

$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{60 \times 10^3}\left(1 - \frac{a_p}{DC}\right)$$

Net power, HP

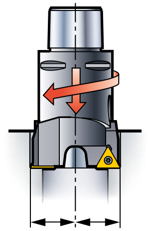
$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{132 \times 10^3}\left(1 - \frac{a_p}{DC}\right)$$

Specific cutting force

Cutting force/area for a given chip thickness in tangential direction.
The k_c value indicates the machinability of a certain material and is expressed in N/mm2 (lbs/inch2).

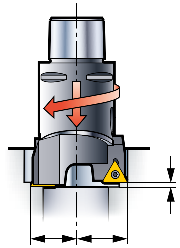
Hole making methods

Productive boring



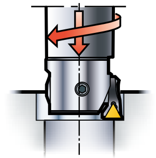
Productive boring involves 2-3 cutting edges and is used for roughing operations of hole tolerances of IT9 or larger, where metal removal rate is the 1st priority. In multi edge boring all slides are set to the same diameter and height. The feed rate is given by multiplying the feed for each insert by the number of inserts ($f_n = f_z \times z$). This is the basic set up for most boring applications.

Step boring



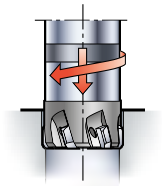
In Step boring the slides are set to different axial heights and diameters. Step boring is used where large radial depth of cuts are required or when boring in soft material (long chipping material). The width of the chip is divided into two small easy to handle chips by this method. The feed rate and surface finish result is the same as if only using one insert ($f_n = f_z$).

Single-edge boring



Single-edge rough boring is used where chip control is demanding (long chipping material) and/or when machine tool power is limited. Only one slide is used. The slide surfaces are protected by covers when not in use. When finish boring an adjustable single-edge tool is used for closer hole tolerances, ($f_n = f_z$).

Reaming



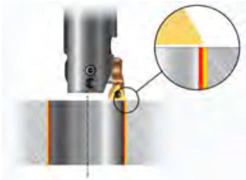
Reaming is a light finishing operation performed with a multi-edge reamer at high feeds.

Tool selection procedure

Production planning process

1 **Component**

Hole dimension and quality



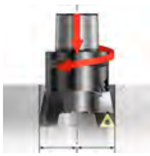
Workpiece material, shape and quantity

2 **Machine**

Machine parameters

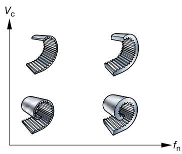
3 **Choice of tool**

Type of tool



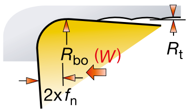
4 **How to apply**

Cutting data, coolant, 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

- Identify the type of operation and note characteristics regarding the hole to be machined, limitations, material and machine.
- Clamping, clamping forces and cutting forces. Is the component sensitive to vibrations?
- Select the tool that covers the boring diameter range and depth for the operation, surface finish and tolerance.

Material

- Machinability
- Chip breaking
- Hardness
- Alloy elements.

2. Machine parameters

Condition of the machine

- Spindle interface
- Machine stability
- The spindle speed
- Coolant supply
- Coolant pressure
- Clamping of the workpiece
- Horizontal or vertical spindle
- Power and torque
- Tool magazine.

3. Choice of tools

Bending stiffness and torque transmission are the foremost important factors when choosing a tool holder for boring operations. Choose the tool according to your specific needs:

- Tools for various materials, applications and conditions.
- Accurate adjustment mechanisms and high precision coolant for finishing.
- Optimize productivity with multiple cutting edge tools.
- Small and large diameter tools.
- For vibration free machining at long overhangs – use dampened tools.
- Reduce tool assembly weight for ease of handling and less momentum.

Engineered solutions

- Often a combination of multiple operations in one tool.
- The operations can be completed during one feed motion.

4. How to apply

Important application considerations

Tool holding

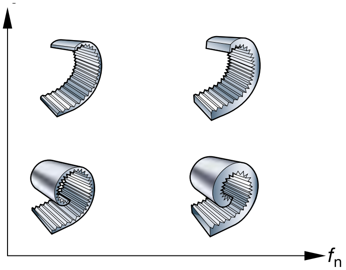
- Always use the strongest coupling and aim for the shortest tool overhang.
- For best stability and hole quality use Coromant Capto®, dampened tools and tapered shanks.

Tool considerations

- Consider entering (lead) angle, insert geometry and grade.

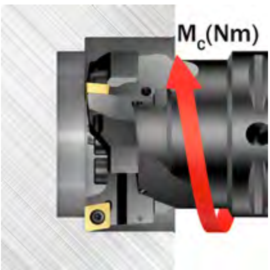
Chip evacuation and cutting fluid

- Chip formation and evacuation are important factors in boring and affect hole quality and hole tolerance.



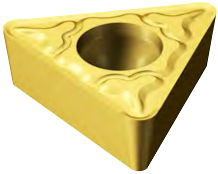
Cutting data

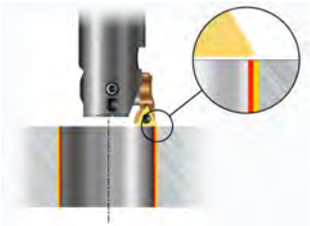
- Correct cutting speed and feed rate is essential for high productivity, tool life and hole quality.
- Keep in mind the torque and power of the machine.

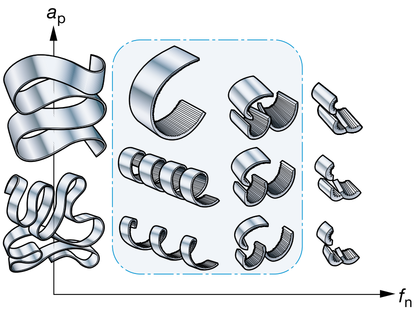


5. Troubleshooting

Important application considerations







Insert wear and tool life

- Correct geometry, grade and cutting data is essential in boring operations.

Chip evacuation

- Check the chip breaking and cutting fluid supply.

Hole quality and tolerances

- Check clamping of boring tool/work-piece, feed rate, machine conditions and chip evacuation.

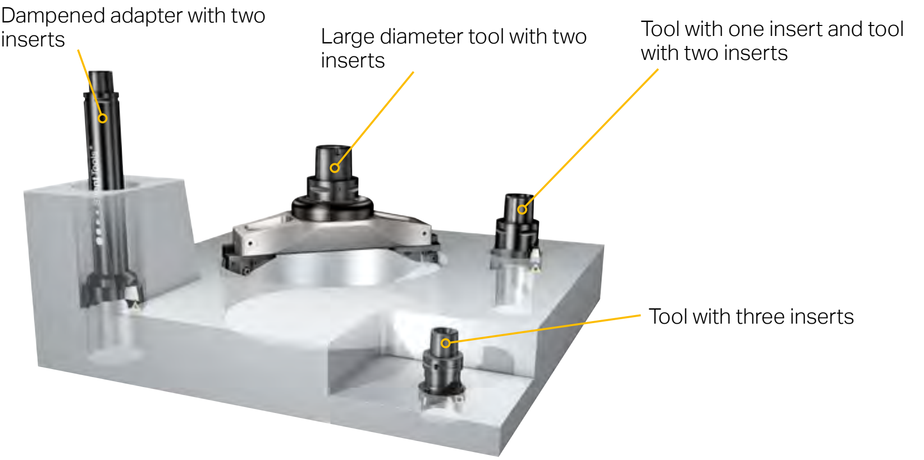
Cutting data

- Correct cutting speed, feed rate and cutting depth is essential for high productivity, tool life and to avoid vibrations.

System overview

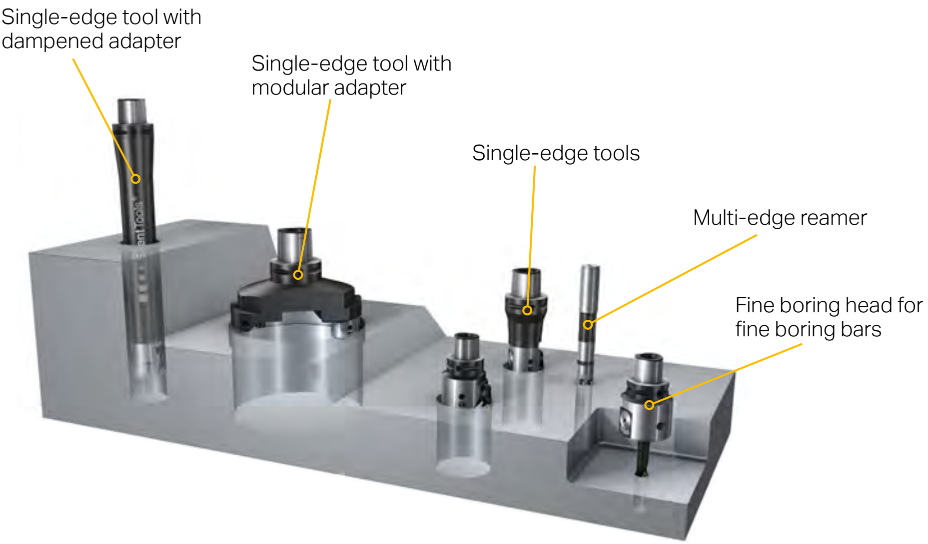
Rough boring tools

Rough boring operations are performed to open up an existing hole to prepare for finishing.



Fine boring tools

Fine boring operations are performed to finalize hole within tolerance and surface finish limits.



Rough boring

Ø inch: 0 2 4 6 8 10 12 14 16 18 20 22 50

Ø mm: 0 50 100 150 200 250 300 350 400 340 500 550 1260

Rough boring tools with two inserts Ø23-170 mm (0.908-6.893")

Rough boring tools with three inserts Ø36-306 mm (1.4-12")

Rough dampened boring tools with two inserts Ø25-150 mm (1-6")

Large diameter rough boring tools with two inserts Ø150-1260 mm (6-50")

Large diameter rough boring tools with two inserts (lightweight). Ø148-300 mm (5.82-11.81")

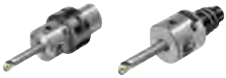
Large diameter rough boring tools with two inserts (dampened). Ø148-300 mm (5.82-11.81")

Fine boring – small diameter

Ø inch: 0 0.4 0.6 0.8 1.0 1.2 1.4 6 12 24 36 61

Ø mm: 0 10 15 20 25 30 35 150 300 600 800 1275

Fine boring heads with solid carbide bar Ø1-8.2 mm (0.04-0.320")



Fine boring heads with indexable boring bar Ø6-20 mm (0.24-0.79")

Fine Boring Head with indexable bar or grooving bar Ø8-32 mm (0.31-1.26")

Multi-edge reamer Ø3.97-31.75 mm (.156 - 1.25")

Fine boring – medium diameters

Ø inch: 0 | 0.4 | 0.6 | 0.8 | 1.0 | 1.2 | 1.4 | 6 | 12 | 24 | 36 | 61

Ø mm: 0 | 10 | 15 | 20 | 25 | 30 | 35 | 150 | 300 | 600 | 800 | 1275

Fine boring with exchangeable heads Ø19-36 mm (0.75-1.42")

Fine boring with cylindrical shank Ø19-36 mm (0.75-1.42")

Fine boring with Coromant Capto (modular) Ø19-167 mm (0.75-6.58")

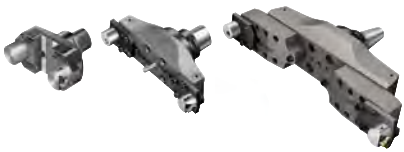
Fine boring with Coromant Capto (dampened) Ø23-167 mm (0.91-6.58")

Fine boring with Coromant Capto (lightweight) Ø69-167 mm (2.716-6.575")

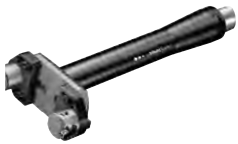
Fine boring – large diameters

Ø inch: 0 | 0.4 | 0.6 | 0.8 | 1.0 | 1.2 | 1.4 | 6 | 12 | 24 | 36 | 61

Ø mm: 0 | 10 | 15 | 20 | 25 | 30 | 35 | 150 | 300 | 600 | 800 | 1275



Fine boring Ø150-1275mm (5.9-50")

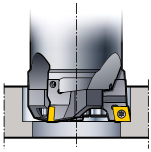


Fine boring (dampened) Ø150-315mm (5.9-12.4")

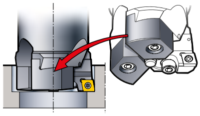
Fine boring with Coromant Capto or arbor mount (lightweight) Ø150-315mm (5.9-12.4")

Choice of tools

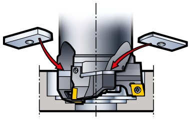
Roughing



Productive boring



Single-edge boring



Step boring

Productive boring

- High metal removal rate.
- Multi-edge boring, inserts on the same level.

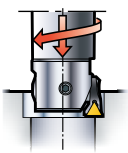
Step-boring

- For rough boring with large stock removal.
- Improved chip control.

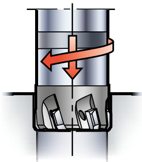
Single-edge boring

- Improved chip control.
- Less machine-power demanding.

Finishing



Single-edge boring



Reaming

Single-edge boring

- High precision fine boring.
- Tolerance capability IT6.
- Adjustability of 0,002 mm (0.00008").

Reaming

- Very good surface finish at high penetration rates.
- Suitable for mass production.

Engineered solutions

- Often a combination of multiple operations in one tool.
- The operations can be completed during one feed motion.

Rough boring tools

Rough boring tool with three inserts

First choice recommendation for medium and high power machines is a rough boring tool with three cutting edges for optimized productivity. Which can also be configured for single-edge and step-boring.

Rough boring tool with two inserts

A rough boring tool with two cutting edges is first choice for low to medium power machines, unstable operations or large diameters.

Light weight rough boring tool

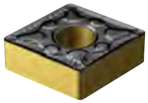
Reduces tool assembly weight, for decreased momentum, easier tool exchange and tool handling. For boring large diameters with increased stability without increased tool weight.

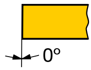
Dampened rough boring tool for long overhangs

Choose dampened rough boring tools for overhangs longer than 4 times the coupling diameter.

Slides for rough boring tools

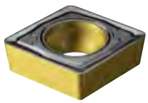
Slides with negative inserts

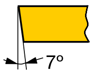




- For stable conditions, choose negative shape inserts for better insert economy.
- Use negative inserts in tough applications that require strong inserts and improved process security.

Slides with positive inserts





- In rough boring, it is an advantage to use positive basic-shape inserts as they give lower cutting forces compared to negative inserts.
- A small nose angle and small nose radius also contribute to keeping the cutting forces down.

Entering (lead) angle and insert shape

The entering (lead) angle of boring tools affects the direction and magnitude of axial and radial forces. A larger entering (smaller lead) angle produces a larger axial force, while a smaller entering (larger lead) angle results in a larger radial cutting force.

Positive inserts

Negative inserts

For interrupted cuts, sand inclusions, stack boring etc. Through holes only.

First choice for general operations, step boring and for shoulder operations.

For high feeds or improved surface finish with Wiper inserts in stable conditions.

Fine boring tools

Single-edge fine boring tool

A single-edge fine boring tool is the first choice for fine boring operations.

Light weight fine boring tool

Reduces tool assembly weight, for decreased momentum, easier tool exchange and tool handling. For boring large diameters with increased stability without increased tool weight.

Fine boring head with fine boring bars

For small diameters a fine boring head with fine boring bars is required.

Silent Tools for long overhangs

Silent Tools (dampened) are the first choice for overhangs longer than 4 times the coupling diameter.

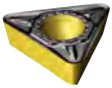
Multi-edge reamer

Multi-edge reamers are suitable for high feeds in mass production.

Cartridges for fine boring tools

General recommendations

Positive inserts 7° clearance angle



Positive inserts 11° clearance angle

Entering (lead) angle

affects the direction and magnitude of the axial and radial cutting forces. The largest entering (smallest lead) angle results in increased axial forces, which is beneficial in boring application. Opposed to a smaller entering (larger lead) angle, which results in larger radial forces, causing vibration in the application.

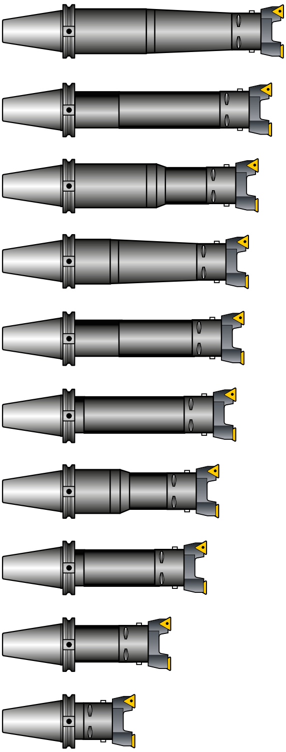
Insert shape

should be selected dependent on the cutting edge engagement. The larger point angle, ensures insert strength and reliability, but also needs more machine power and has a higher tendency to vibrate due to a large cutting edge engagement. Minimizing the insert point angle can improve tool stability and possible radial movements, giving less variation and cutting force. Positive basic shape inserts with 7° clearance angles are first choice.

Insert nose radius

is a key factor in boring operations. The selection of nose radius is dependent on depth of cut and feed rate which influences the surface finish, chip breaking and insert strength. A large nose radius will deflect the boring tool more than a smaller nose radius and be more prone to vibrations. Using a light cutting insert geometry, thin coating and small nose radius with lighter depths of cut contributes to keeping cutting forces low.

Tool overhang



- Choose the shortest possible adapter length.
- Choose the largest possible diameter/size of adapter.
- For long overhangs (larger than 4 x coupling diameter) use dampened adapters.
- If possible, use a tapered adapter to increase the static stiffness and to reduce the deflection.
- For long overhangs, ensure rigid clamping with flange contact to spindle if possible.

How to apply

Hole tolerance

Tolerances will be influenced by:

- the clamping of the tool holder
- the fixture of the component
- the wear of the inserts etc.

Always ensure a final adjustment is made after measurement of the hole diameter while the tool is still in the machine spindle. This will compensate for any misalignment that can happen between the machine-tool spindle and tool setting, radial deflections and insert wear.

Boring and reaming tools

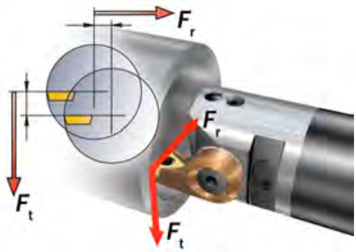
	Rough boring tool with multiple edges 	Single-edge fine boring tool 	Multi-edge reamer for high feed finishing 
IT6		■	■
IT7		■	■
IT8			
IT9	■		

Fine boring tools

Adjustable fine boring mechanism

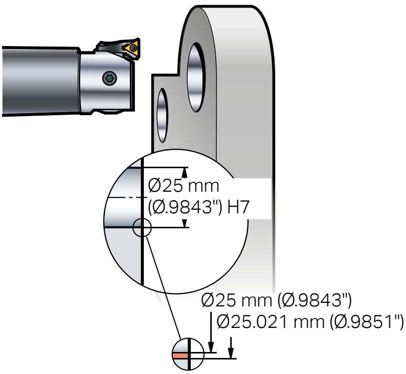
Single-edge fine boring tools have adjustment possibilities to accurately pre-set the cutting edge within microns.

Tool deflection



- Boring tools for finishing, with one cutting edge, will experience some degree of radial deflection during machining due to the cutting forces.
- The depth of cut and length of overhang influence the radial deflection of the boring tool.
- The deflection might cause undersized holes or vibrations.
- A measuring cut is normally needed, followed by a final adjustment of the tool.

Hole tolerance



Boring tools – general

Cutting fluid supply

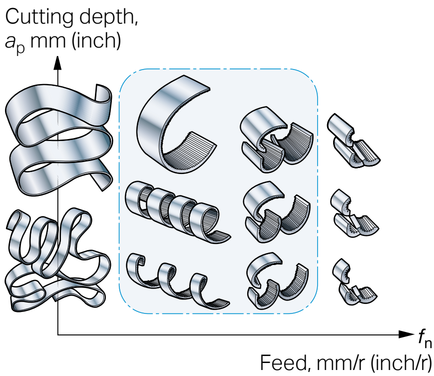
Chip evacuation, cooling and lubrication between the tool and the workpiece material are primary functions of cutting fluid.

- Apply cutting fluid for optimized chip evacuation, cooling and lubrication.
- Affects hole quality and tool life.
- Internal cutting fluid is recommended in order to direct the fluid to the cutting zone.

Chip control and chip evacuation

Chip formation and chip evacuation are critical issues in boring operations, especially in blind holes.

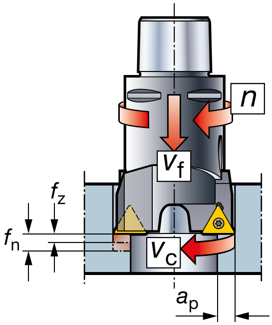
Ideally, chips should be in the form of defined commas or spirals.



Factors that have an influence on chip breaking are:

- the insert micro and macro geometry
- nose radius
- entering (lead) angle
- cutting depth
- feed
- cutting speed
- material.

Cutting data recommendations

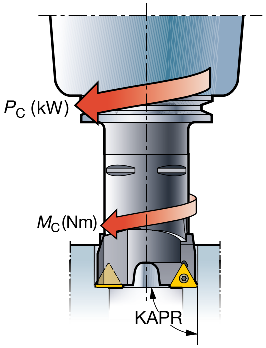


Setting the right cutting speed (v_c) and feed (f_n) is dependent on application. Increased cutting speed and/or feed, increases the risk of poor process security and reliability, leading to poor chip evacuation, chip jamming and insert breakage. Especially in deep hole applications. Low cutting speed can generate increase chances for built-up edge (BUE), leading to bad surface finishes, higher cutting forces and decrease in tool life. General cutting data for insert geometry and grade can be followed, with the following exceptions:

- Rough boring
  Max start value v_c = 200 m/min (656 ft/min).
- Fine boring with fine boring adapters:
  Max start value v_c = 240 m/min (787 ft/min).
- Fine boring with fine boring bars:
  Max start value v_c = 90 – 120 m/min (295 – 394 ft/min).
- Fine boring:
  Max APMX = 0.5 mm (.020 inch).

Cutting speed is mainly limited by:

- vibration tendencies
- chip evacuation
- long overhangs.



Feed and cutting depth

Excessive cutting edge engagement, large depth of cut (a_p) and/or feed (f_n), can cause vibration and larger power consumption. To small of cutting depth and the insert will tend to ride on the pre-machined surface, only scratching and rubbing it, also leading to poor result in tool wear and surface finish.

Power and torque consumption

When boring make sure the machine can prove sufficient power and torque.

Tool maintenance and use of torque wrench

- Always use a torque wrench and apply the recommended torque on screws for insert and tool assembly.
- Check inserts and insert seats regularly to be free from dirt & are not damaged.- Clean all assembly items before assembly
- Replace worn or exhausted spare parts.
- Lubricate all assembly items as well as the fine boring adjustment mechanism with oil at least once a year.
- Use a suitable assembly mounting fixture and tool pre-setter.
- When assembling dampened tools, never clamp straight over the adaptor body. Adaptors are easily deformed due to the thin wall thickness.
- Check machine spindle run-out, wear and clamping force.

How to apply reaming tools

- The reamer should not be expected to correct any positional or straightness errors in the pre-machined hole.
- The straightness of the pre-machined hole should be less than 0.05 mm (.0020 inch).
- A small runout is very important for reaming operations.
- Maximum recommended runout is 5 microns.
- Make sure the reamer is concentric with the pre-machined hole.
- Choose the shortest possible tool holder and shank.
- Emulsion as cutting fluid generates better tool life than oil.
- Use recommended cutting data.

Troubleshooting

Factors that affect vibration tendencies

Vibration tendencies grow towards the right.

Entering angle

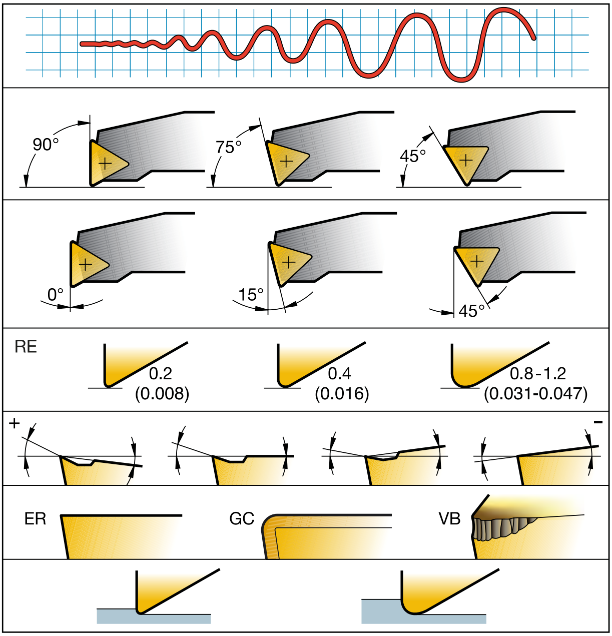
Lead angle

Corner radius

Micro and macro geometry

Edge design

Depth of cut (DOC)



- Decrease cutting speed.
- Apply step boring.
- Choose a 2-edge rough boring tool.
- Choose a light-cutting geometry and grade.
- Use a smaller nose radius.
- Check workpiece clamping.
- Check machine spindle, wear, clamping, etc.
- Increase depth of cut (finishing).
- Decrease depth of cut (roughing).
- Use dampened tools if long overhang.
- Check that all units in the tool assembly are assembled correctly with the correct torque.
- Reduce feed or increase feed.
- Use the largest tool diameter possible.
- Use the shortest tool overhang possible.

Insert wear

Insert wear patterns and remedies in boring are generally very similar to turning.

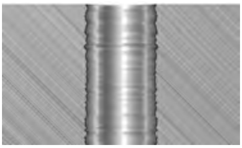
Chip breaking

	Cause	Solution
	Too short, hard.	• Increase cutting speed. • Decrease feed. • Change geometry to a more open chip breaker.
	Too long.	• Increase feed. • Decrease cutting speed. • Change geometry to a more closed chip breaker.

Tool vibration

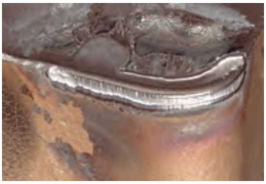
	Too high feed. Too high speed. Too large cutting depth.	• Decrease feed. • Decrease speed. • Apply step boring.
	Too high cutting forces.	• Decrease depth of cut. • Use positive inserts. • Use smaller nose radius.

Feed marks

	Too high feed.	• Choose knife edge wiper insert. • Use larger nose radius. • Decrease feed.

	Cause	Solution

Insert wear



Cause: Wrong cutting data.

Solution:
- Change cutting edge and investigate reason for wear pattern – cutting data, insert geometry and insert grade.

Chips scratching surface

Cause: Bad chip breaking.

Solution:
- Change cutting data.
- Change insert geometry.

Surface finish

Cause: Bad surface finish.

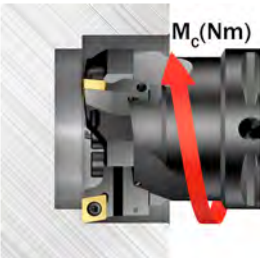
Solution:
- Increase speed.
- Use coolant.
- Use a cermet grade.

Machine power limitation

Cause: Limited machine power.

Solution:
- Decrease cutting data.
- Apply step boring.
- Decrease number of inserts in cut.
- Reduce depth of cut.

Power and torque consumption



Cause: When rough boring, make sure the machine can provide sufficient power and torque.

Solution:

Important parameters are:
- Feed.
- Number of inserts.
- Diameter.
- Depth of cut.


BIG-PLUS SYSTEM
8 Nm
BIG-PLUS SYSTEM

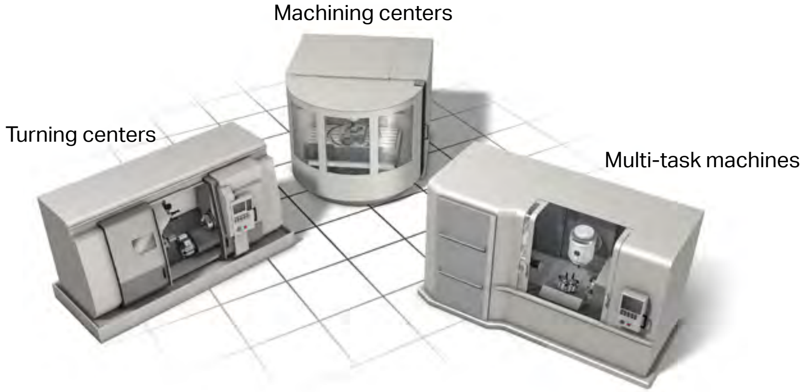
Tool holding

The clamping of a cutting tool can influence the productivity and performance of the cutting tool dramatically. Therefore it is important to choose the right holding tools. This chapter will simplify the decision process and give guidelines how to apply and maintain the holding products.

Tool holding systems

- The tool holding interface with the machine plays a very important part in the cutting process.
- Stability, time for tool changing, accuracy, flexibility, modularity, handling and storing is of vital importance for successful machining.
- Compared to conventional shank tools, a quick change system can increase the effective cutting time by 25% in turning centers.

Tool holding systems today



- Tooling has evolved through the necessity to produce new types of machine manufacturing standards.
- These tools have generally followed the spindle interface design of MTMs, without any standardization controls.
- There are over 35 types of spindle interface on machines today, with as many tooling options to support, hence exchangeability and assortment availability decreases dramatically.

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

History of machine tapers

- The first version of this steep taper type was introduced during the 1920's and standardized (DIN) in 1974.
- The taper was the basis of most machine tool spindles, due to the long taper, giving secure contact and stability.
- It is still popular today, in various sizes and different standards, using 7/24 taper. They are however not suitable for both rotating and static applications.

Rotating machine interfaces

- There has been an ever increasing variety of different rotating machine interfaces on the market today.
- Unfortunately, these systems are not designed for both clamping in a spindle and modular use.
- None of these systems are suitable for rotating and static applications.

Coromant Capto®

Three systems in one

- Coromant Capto® was introduced in 1990.
- Coromant Capto® was adopted as an ISO Standard during 2008.
- Coromant Capto® is a true universal tooling system for use in:
  - Turning centers
  - Machining centers
  - Multi-task machines

The history of the Coromant Capto® system

- **Machining center / Rotating tools**

Solid holders

Varilock

Coromant Capto®/ Basic holders

1980 1990

Today

- **Turning center / Turning tools**

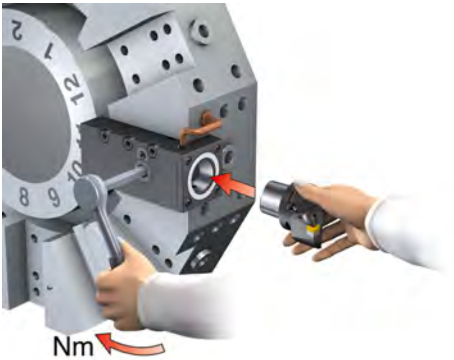
Shank holders

Block Tool System

Coromant Capto®/ Clamping units

The history of the Coromant Capto® system

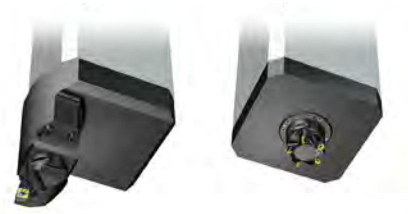
Quick change



- Turning Centers
- Vertical Lathes

Increased machine utilization

Integrated spindle



- Multi-Task Machines
- Vertical Lathes
- Machining Centers with Turning

Increased stability and versatility

Modular systems

- Machining Centers
- Multi-Task Machines
- Vertical Lathes

Increased flexibility

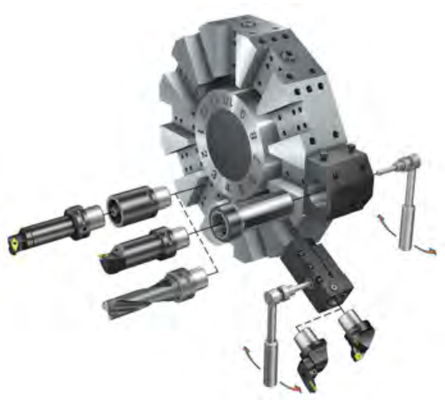
A dramatic development of the machines

Trends

Machines and machining methods

- Multi-task machines requiring one holder system for both spindle and turrets.
- Several turrets on multi-task machines and turning centers.
- More multi-function tools for multi-task machines.
- Driven tools in turning centers.
- Powerful interfaces in the machine control system for higher degrees of automation.
- 3-D models of tools and holders to virtually check the machine process.
- Integration of various manufacturing technologies into fewer machine types.
- High pressure coolant.

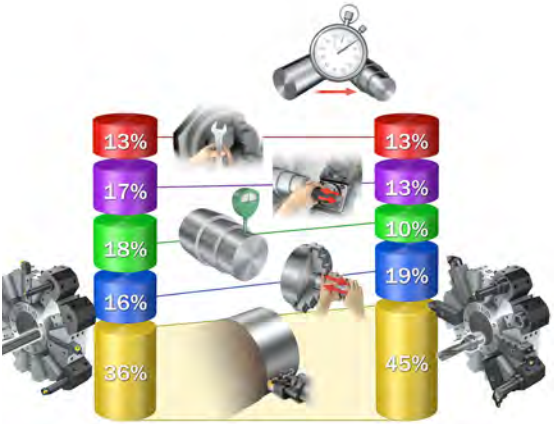
When to use quick change tooling



- Machine requires frequent setup changes.
- Measuring cuts are necessary to get correct size.
- Machining is performed with high cutting data and relatively short tool life.
- One operator services more than one machine.

Reduce down time in your machines

Only 36% of the machine time is used for metal cutting



Conventional shank tools

Coromant Capto® quick- change system

- Service and maintenance
- Insert change and tool change
- Measuring of the tool and workpiece
- Change of workpiece
- Effective cutting time

Quick change tooling offers a productivity increase of 25%

Coromant Capto® system

In which machine types and sizes do we need a modular system?

Horizontal machining center

Machining Center with:

- Coromant Capto® size C6 and bigger
- 7/24 tapers in size 40 and bigger
- HSK63 and bigger.

- Multi-task machine with need of long overhangs
- Vertical Turning Center
- Turning Center together with SL*.

*SL is a universal modular system of adaptors with exchangeable cutting heads.

Minimize tool holder inventory

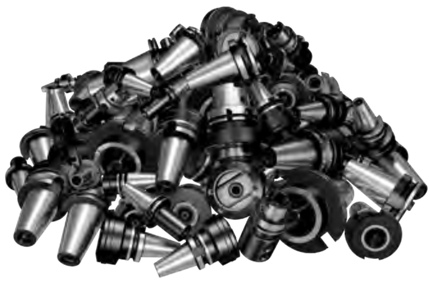
By combining basic holders, adapters and (when needed) extensions or reductions, many different assemblies for different machines can be built.

Modular

ISO 40 ISO 50 HSK 100 HSK 63

Number of items with modular tools:
4 + 2 + 8 = 14 items

Solid



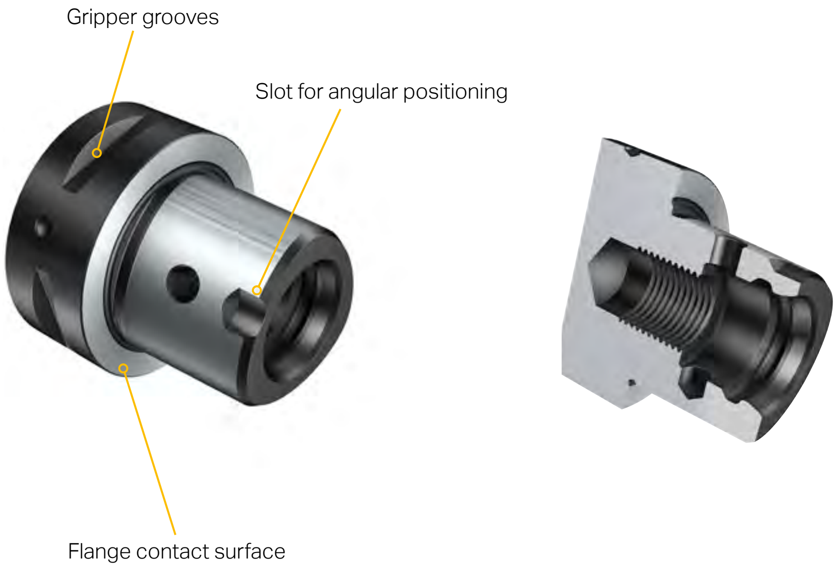
Total 64 items

Modular tools give access to very large number of tooling solutions, with very few items.

The Coromant Capto® coupling

The unique Coromant Capto® coupling has som very specific features:

- The good flange contact face in relation to the ground taper polygon gives maximum stability due to two-face contact and interference fit.
- There are four gripper grooves for the automatic tool change.
- There is one slot for angular positioning of the cutting tool.



The only universal coupling that can be used in all applications without compromise.

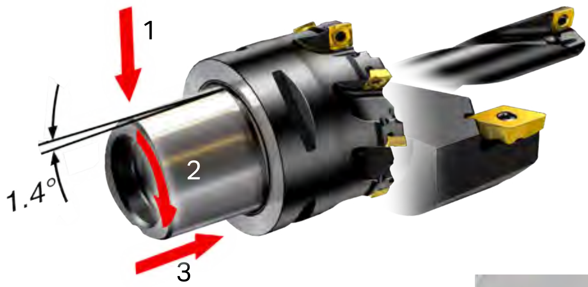
Coupling features and benefits

The main feature of the coupling is the positive 3-way locking

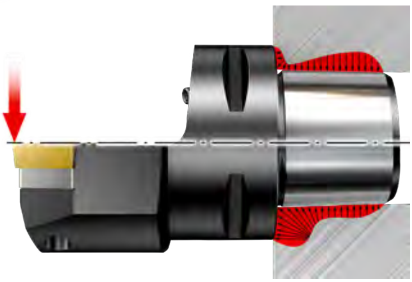
1. The radial centering is taken care of by the conical part of the polygon.
2. The low taper angle makes it possible to transmit the full force into the flange contact. The strength of the polygon coupling makes it possible to clamp with higher force than other systems. This is very important for the bending stiffness.
3. A polygon shape is self centering and takes care of the orientation without the need for a driving slot, therefore there is no play in the coupling. The polygon shape is also unique due to its capability to transmit high torque due to three contact areas.

Due to the above features - radial and axial contact and self centering ability - the coupling has extremely good repeatability, within 2 microns (.00008 inch).

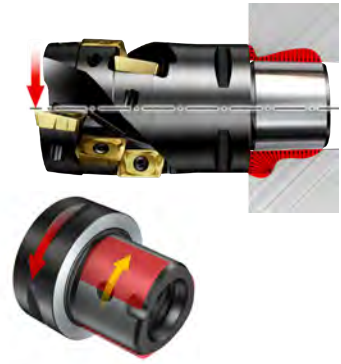
The gripper grooves are designed to give maximum bending stiffness and a higher clamping force, due to the fact that the Capto polygon has a greater surface area.



3-way locking



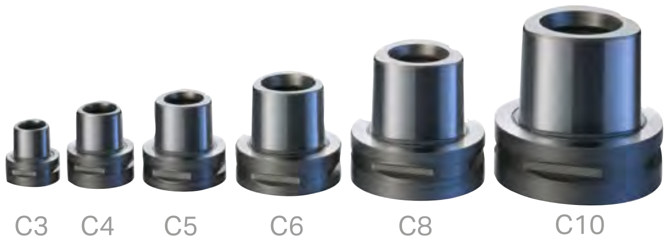
Transmission of torque



The polygon shape transmits torque without any loose parts such as pins or keys.

- No pins, keys, etc.
- No play in the coupling
- Symmetrical loads
- Two face contact/high clamping force.

Six different coupling sizes



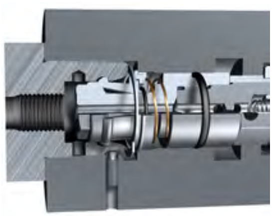
C3 = D 32 mm (1.260 inch)
C4 = D 40 mm (1.575 inch)
C5 = D 50 mm (1.969 inch)

C6 = D 63 mm (2.480 inch)
C8 = D 80 mm (3.150 inch)
C10 = D 100 mm (3.937 inch)

Different methods of clamping

One coupling offers two methods of clamping.

Segment clamping

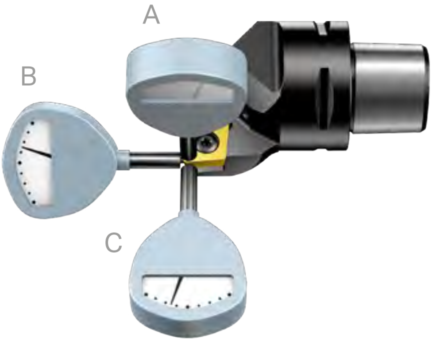


Clamping method for quick-change and automatic tool changing.

Center bolt clamping

For modular clamping solutions, e.g., when using extensions and basic holders.

Excellent repetitive accuracy and guaranteed center height



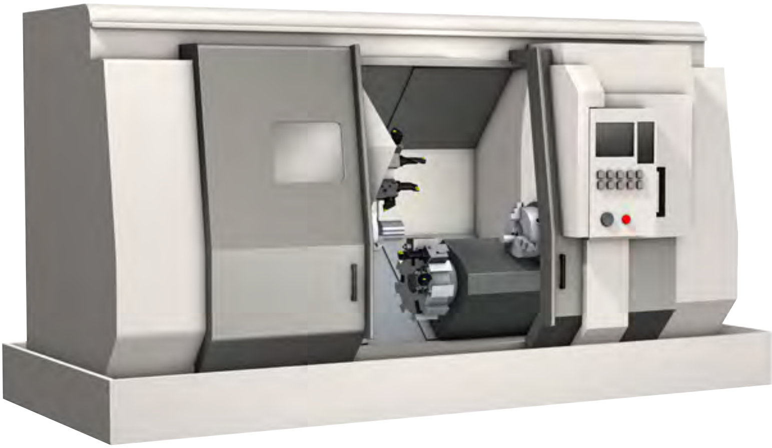
- The repeatable accuracy is ±2 microns [μm] (±.00008 inch) of the center height, length and the radial measurement (A),(B),(C).
- Few or no measuring cuts needed if pre-measuring is used (first component right).

Less vibration with stable coupling

In internal machining the Coromant Capto® coupling is an outstanding solution to clamp the boring bar, with a firm secure grip around the entire polygon.

The boring bar is very often clamped with 2-3 screws. This causes problems with vibration, bad surface finish, inserts worn out quickly and production disturbances, with downtime spent on adjusting cutting data and measuring the component.

Quick change tooling for turning centers



What is a turning center?

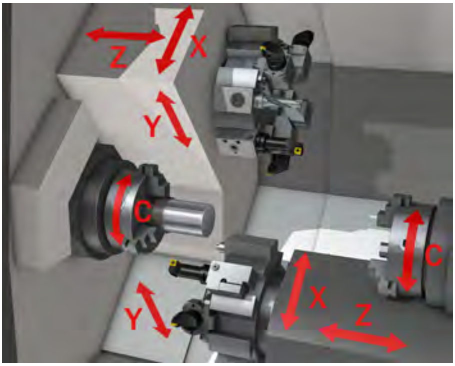
- The principle of lathes and turning centers is to cut a rotating component with a stationary cutting tool.
- The cutting tool moves parallel and perpendicular to the workpiece axis to provide the desired finished shape.
- When a cutting tool is applied to the workpiece, it can be shaped to produce a component which has rotational symmetry.

The turning center has a choice of configurations

- Horizontal and vertical design
- Sub-spindle for two-sided machining
- Driven tools
- Y-axis for eccentric boring and milling.

Configuration of a turning center

Spindle rotation and definitions of axis



- Several multi-axis machine tool programs can provide turning results from roughing and grooving to threading and finishing.

Quick change tooling for turning centers

A quick-change system offers:

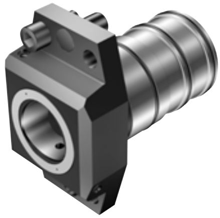
- faster and efficient tool changing
- inserts which can be changed outside the machine
- pre-setting possibilities.

The most economical system for:

- small batch production, quicker setup times
- operations with frequent insert changes.

Less than 180° for clamp and unclamp

Typical clamping units for turning centers

VDI angled Camshaft activated	Square shank Camshaft activated	Automatic unit Hydraulically operated
		
VDI straight Camshaft activated	**Round shank** Segment clamping	**Special applications** Camshaft activated
		

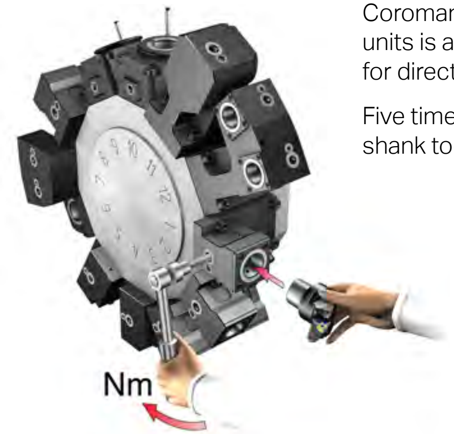
Different methods how to install quick change

Directly integrated into the turret

Coromant Capto® directly integrated in turrets is the best solution to get maximum performance out of the Coromant Capto® coupling.

Different methods how to install quick change

Converted by using standard clamping units



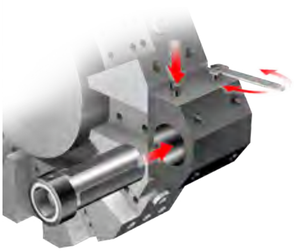
Coromant Capto® as a machine interface via clamping units is a good alternative when it's not possible to go for direct integration, (existing machines etc).

Five times faster tool change than with conventional shank tools.

Turning lathes can easily be converted to Coromant Capto® quick change tools using standard clamping units. No modifications to the turret, and no special adaptors required.

Internal tools

External tools



Machine adapted clamping units

Coromant Disc Interface (CDI)

- Flexible and symmetrical interface, 180° mountable.
- Same interface for static and driven tool holders. Static and driven tool holders can be used in all positions.
- Higher cutting performance.
- Longer cutting tool life.
- Better workpiece quality.
- More available tool length for radial drilling operations.
- Increased production.
- Rationalized tooling.
- Reduction in tooling costs.

Static clamping unit, straight

Driven drill/milling unit, straight

Static clamping unit, right angle

Driven drill/milling unit, right angle

Coromant Bolt-on Interface (CBI)

- Flexible and symmetric interface, 180° mountable.
- Same interface for static and driven tool holders.
- Static and driven tool holders can be used in all positions.
- Higher cutting performance.
- Longer cutting tool life.
- Better workpiece quality.
- More available tool length for radial drilling operations.
- Increased production.
- Rationalized tooling.
- Reduction in tooling costs.

Driven tool holder

Clamping unit for external turning

Clamping unit for internal turning

Double clamping unit for external turning for tool change with Y-axis

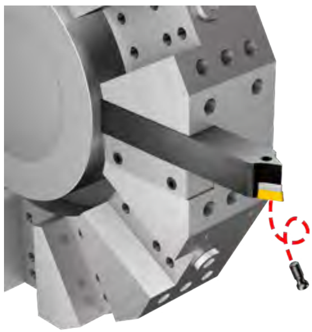
A quick change system

Insert change by using sister tools

- Less downtime
- Few or no measuring cuts. Improved profitability
- No risk of losing insert screws in the chip conveyer
- Ergonomic
- Easy to clean the tip seat outside the machine.

0.5 min

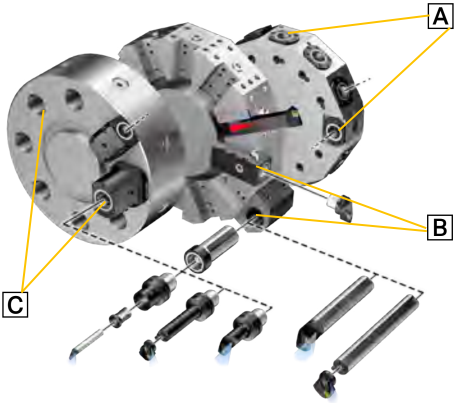
1.5 min



Changing to a sister tool with a quick change system is faster than changing the insert inside the machine.

Different ways how to install quick change

Tooling alternatives in conventional turrets



A Hydraulically operated clamping units

- Manual push-button tool changing
- Fully automatic tool changing possi-bilities.

B Shank type clamping units

- Square and round shank tools as well as cutting units for external and internal operations.

C Clamping units for VDI turrets

- Angled and straight clamping units for external and internal operations.

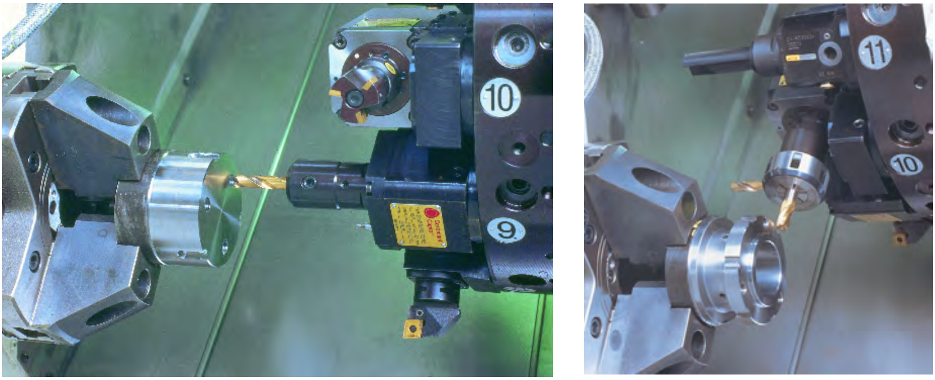
Example of installations.

Coromant Capto® driven tool holders

Driven tool holders provide the key to dramatic improvements in machining economy by allowing milling, turning and drilling operations to be carried out in a single setup.



- Driven tool holders can be supplied for specific machine requirements.
- Spindle dimensions
  - Machine type and model
  - Maximum turret swing diameter
  - Maximum tool length.



Example of installations.

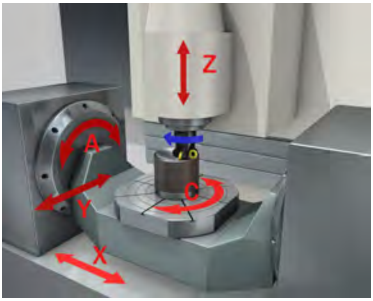
Modular tooling for machining centers

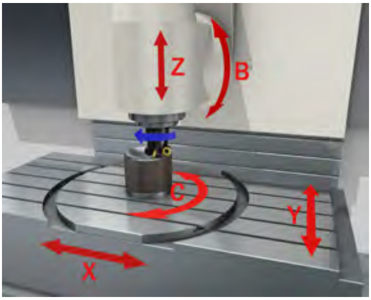
What is a machining center?

- A machining center is a multi-function machine that typically combines boring, drilling and milling tasks.
- Machining centers could be in horizontal design as well as vertical design.
- 5-axis machining centers add two more axes in addition to the three normal axes (X/Y/Z).

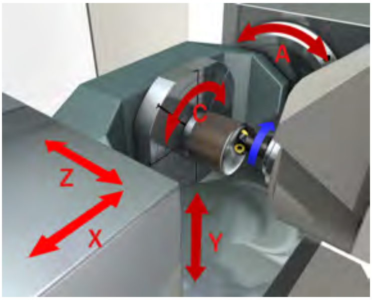
Spindle rotation and definitions of axis

Configuration of a vertical machining center





Configuration of a horizontal machining center



Machining centers can be horizontal and vertical designs

- The basic type has 3 axes. The spindle is mounted along the Z-axes.
- 4- and 5-axes machining centers adds more axes (A/B/C) in addition to the three normal axes (X/Y/Z).
- With several 5-axis machining centers, ones with a rotating or indexing attachments, the fifth-axis moves around the X-axis. (A-axis) and ones with a B-axis head, the fifth-axis moves around the Y-axis. (B-axis).
- Often the B-axis controls the tilt of the cutting tool itself and the A- and C-axes allow the workpiece to be rotated.

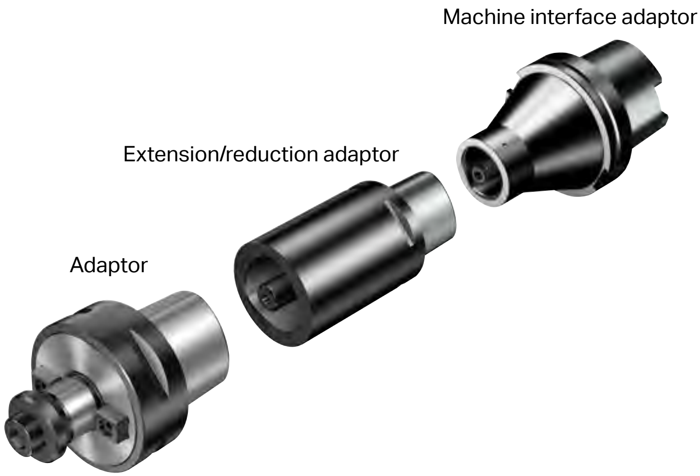
Modular tooling for machining centers

In a machining center a modular system can provide many advantages such as:

- Flexible tooling – the same tools can be used in several machines and machine interfaces.
- Flexible tooling – build your own assemblies and reduce the need for special significantly.
- Reduced inventory.

Build your own assemblies

Use Coromant Capto® adaptors for all spindle interfaces



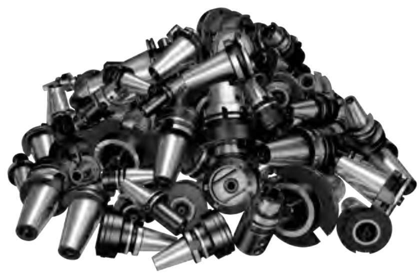
Minimize tool holder inventory in machining centers

Modular tools give access to a very large number of tooling solutions, with very few items!

Modular

Number of items with modular tools:
4 + 2 + 30 + 10 = 46 items.

Solid



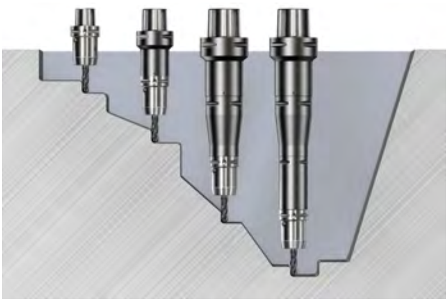
Number of items solid tools:
4 x 3 x (30 + 10) = 480 items.

Right combination for best possible rigidity

Extension adaptors and reduction adaptors

Extended tools for machining centers are frequently required to be able to reach the surface to be machined.

With Coromant Capto® modular system it is possible to build an assembly, so the right length can be achieved.



- It is important that the minimum length is used, particularly when long overhangs are required.
- With modular tools it is always possible to use optimal cutting data for best productivity!
- Modular tools are built together in minutes!
- Get closer tolerances.

All main machine interfaces covered

CAT-V 40
CAT-V 50
CAT-V 60
ISO 40
ISO 50
ISO 60
MAS-BT 30
MAS-BT 40
MAS-BT 50
MAS-BT 60

CAT-V BIG PLUS® 40
CAT-V BIG PLUS® 50

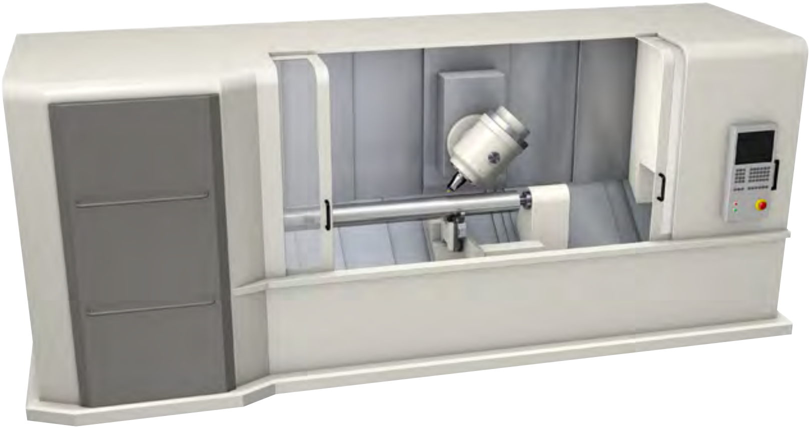
ISO BIG PLUS® 40
ISO BIG PLUS® 50

MAS-BT BIG PLUS® 30
MAS-BT BIG PLUS® 40
MAS-BT BIG PLUS® 50

HSK A/C 40
HSK A/C 50
HSK A/C 63
HSK A/C 80
HSK A/C 100
HSK A/C 125
HSK A/C 160
HSK A/C/T 40
HSK A/C/T 63
HSK A/C/T 100
HSK F 80 (with pins)

Coromant Capto® C3
Coromant Capto® C4
Coromant Capto® C5
Coromant Capto® C6
Coromant Capto® C8
Coromant Capto® C10

Modular tooling for multi-task machines

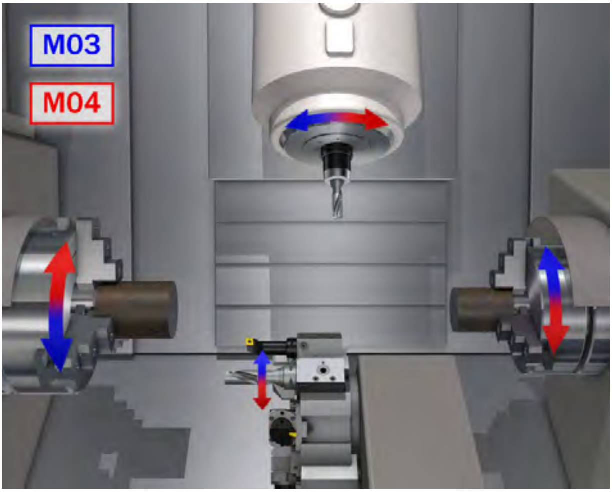


What is a multi-task machine?

- Multi-task machines come in a variety of configurations:
  - horizontal or vertical design.
  - two spindles (main and sub) and a B-axis spindle enable milling and turning operations on both front and back face of the workpiece.
  - each spindle acts as a workpiece holder allowing multi-axis machining on either front or back face of the workpiece.
- In a multi-task machine, the workpiece can be completed in a single machine setup, e.g., turning, milling, contouring and milling of angled surfaces, and grinding.
- Multi-task machines are a combination of a turning center and a machining center.

Definitions of the spindle directions

The program language for defining the spindle direction

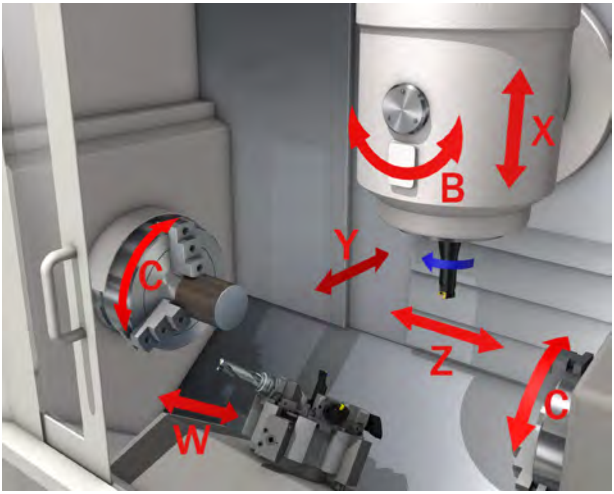


M03 = Clockwise spindle direction

M04 = Counterclockwise spindle direction

Configuration of a multi-task machine

Spindle rotation and definitions of axis

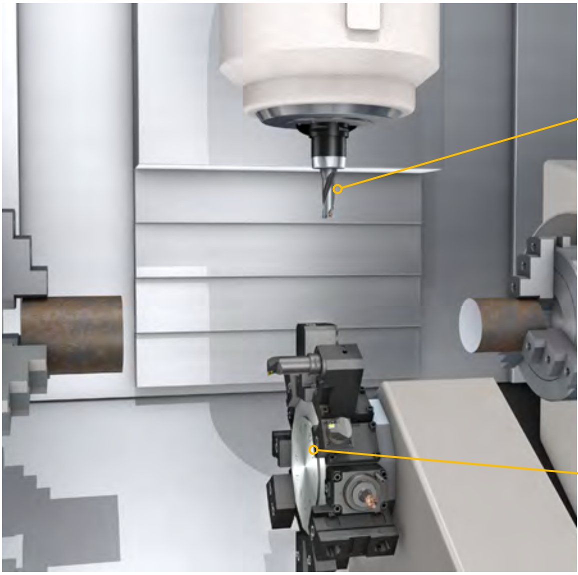


How to use modular tooling in a multi-task machine

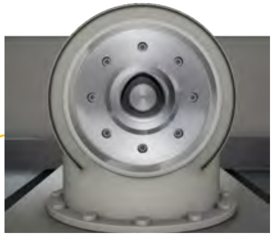
The milling spindle in a multi-task machine tool should be able to carry both rotating and non-rotating tools. Coromant Capto® is the only tooling system that can fulfill this demand without compromise.

Multi-task machine tools are often used in "done-in-one" applications in which operations run from roughing to finishing in one machine tool setup.

Therefore multi-task machine tools needed a tooling system with unsurpassed rigidity and repetitive accuracy both radially and axially, like Coromant Capto®.



Multi-task machine tool with Coromant Capto® integrated tool spindle and lower turning turret with Coromant Capto® clamping units.



The Coromant Capto® tooling system is directly integrated in the spindle.

Turret with Coromant Capto® tooling system

New multifunctional tools for multi-task machines

For taking advantage of versatile multi-task machine tools and to optimize their efficiency, there is sometimes a demand for running them with dedicated tooling. These tools are only available with Coromant Capto® and have been invented for multi-task machine tools, offering:

- accessibility, stability and higher productivity
- reduced tool changing time
- saved tool pocket in tool magazine
- cost reduction - one tool replaces many tools.

Multifunctional tools
– one milling and four turning tools in one

Twin tools
– two turning tools in one

Mini-turrets
– four turning tools in one

Build your own mini-turret

Four cutting heads applied to one tool holder

Radial

Axial

Pick and choose from a large number of exchangeable cutting heads for turning, threading, parting and grooving operations for building an optimized tool for the component.

- Reduce tool changing time
- Save tool pockets in tool magazine
- For both external and internal use.

Use of adaptors in a multi-task machine

Tool adaptors for shank tools

Turning tool adaptors for
- shanks
- bars
- blades
- mini-turrets

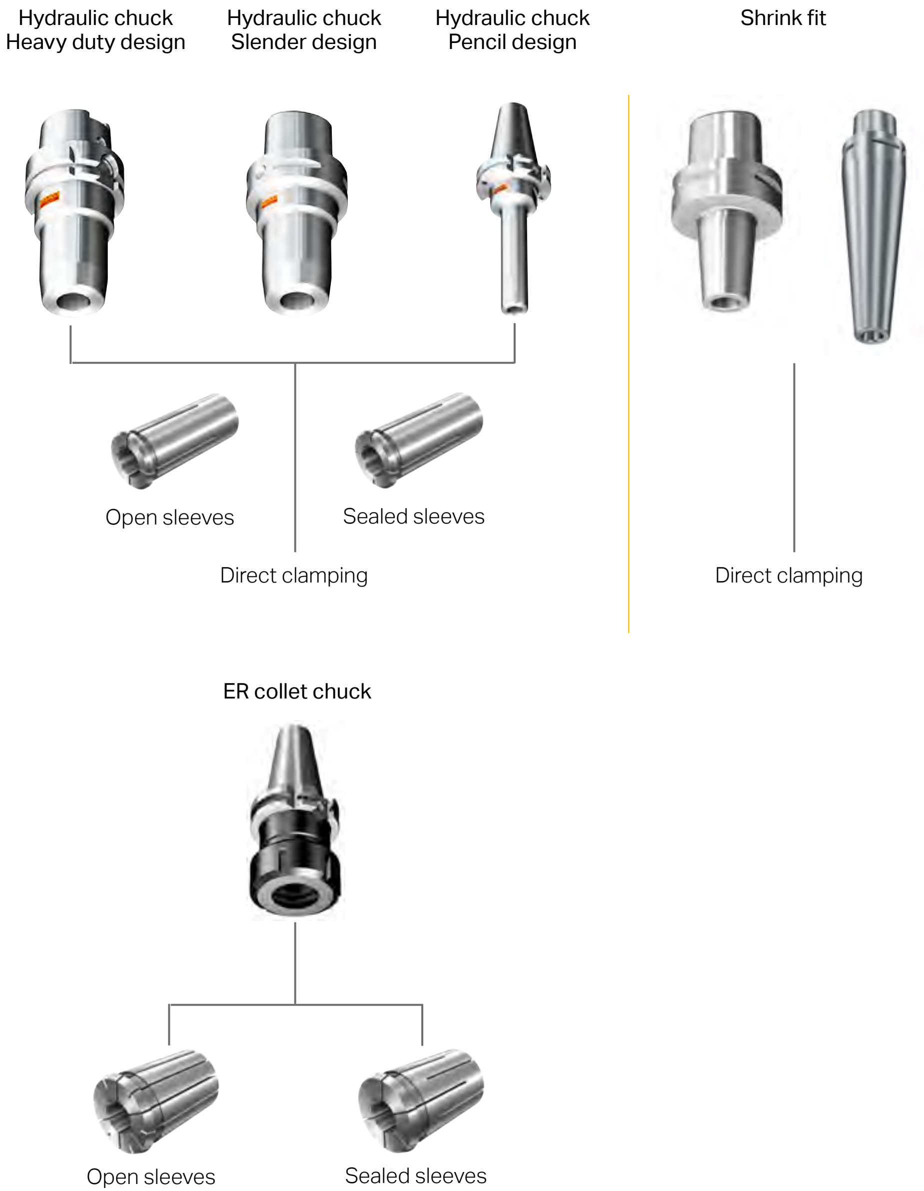
...to make it possible to use shank tools also in a multi-task machine with an integrated modular tool system in the spindle.

Tool adaptor with blade for parting off

Tool adaptor for boring bar

Chucks

Benefits of using hydraulic chucks

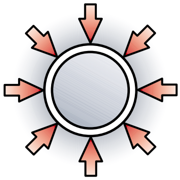


Choice of chucks

	Hydraulic chuck	Shrink fit chuck	Mechanical chuck	ER collet chuck	Side-lock adaptors Weldon, ISO 9766
					
Pull out security, torque transmission	Very good	Very good	Good	Acceptable	Very good
Easy handling	Very good	Acceptable	Good	Good	Very good
High precision, run-out	Very good	Very good	Good	Acceptable	Acceptable
Flexibility	Very good	Acceptable	Very good	Very good	Acceptable
Accessibility	Very good	Very good	Acceptable	Good	Acceptable

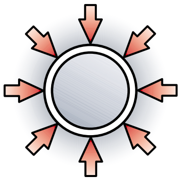
Very good Good Acceptable

Hydraulic chucks



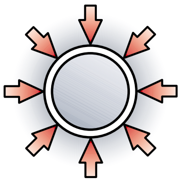
- Best pull out security on the market - clamping force repeats time after time.
- Precision run out < 4 µm (.00016") at 2.5 x DC - high precision repetition.
- Easy handling - torque wrench used for secure clamping.

Shrink fit chuck



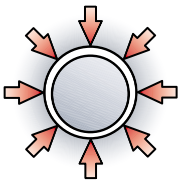
- High pull out security and high precision.
- Small nose diameter possible – good accessibility.
- Symmetrical design.

Mechanical chucks



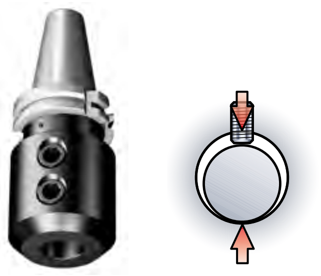
- Cylindrical sleeves can be used – good flexibility.
- Accessibility not so good because of its design (often Heavy Duty).

ER collet chuck



- Very flexible in clamping diameters thanks to collets.
- Not depending on shank tolerance h6.
- Low torque transmission and run-out.

Side-lock adaptors Weldon, ISO 9766

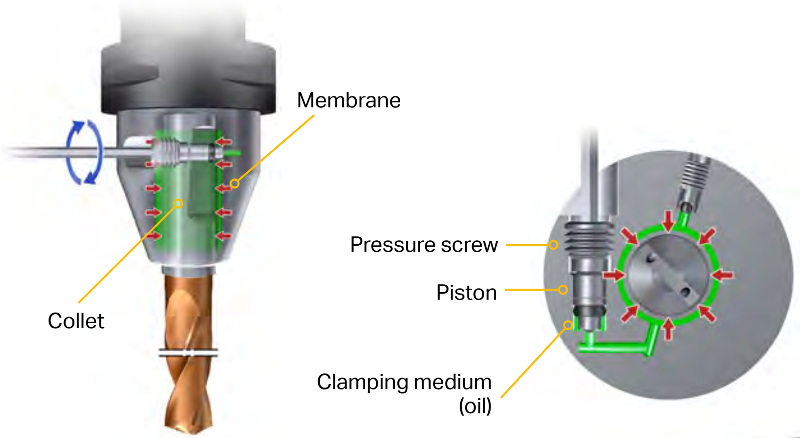


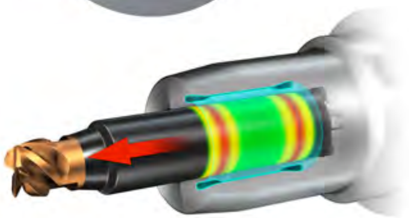
- High torque transmission.
- Low precision – low tool life and low surface finish.

Hydraulic chucks

The secret behind the high precision and pull-out security

- A new generation of hydraulic chucks provides highest precision and torque transmission capability.
- The secret behind the high precision and pull-out security of CoroChuck 930 is the optimized design of the membrane. It allows for secure clamping with two supports on each side (fulcrums).





Try to minimize the gauge length

- It is important to maintain as short a gauge length as possible to increase stability and reduce deflection.
- Length reduction as little as 20% can have a significant reduction in deflection (-50%).

Influence of run-out on tool life

- Runout should be < 0.006 mm (< .001 inch).
- For every 0.01 mm (.0004 inch) runout - up to 50% decrease in tool life.
- More critical as tool diameter gets smaller.

Tool holding requirements

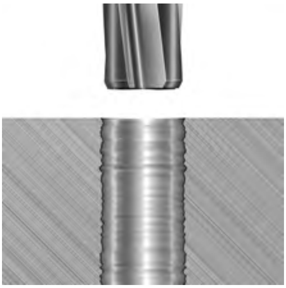
Application - Roughing and semi-finishing

- Main criteria = clamping force
- High torque capability
- For best performance use cylindrical shanks
- Versatility of collets.

Application - Finishing

- Main criteria = runout
- Influence on tool life and component - finish and accuracy.

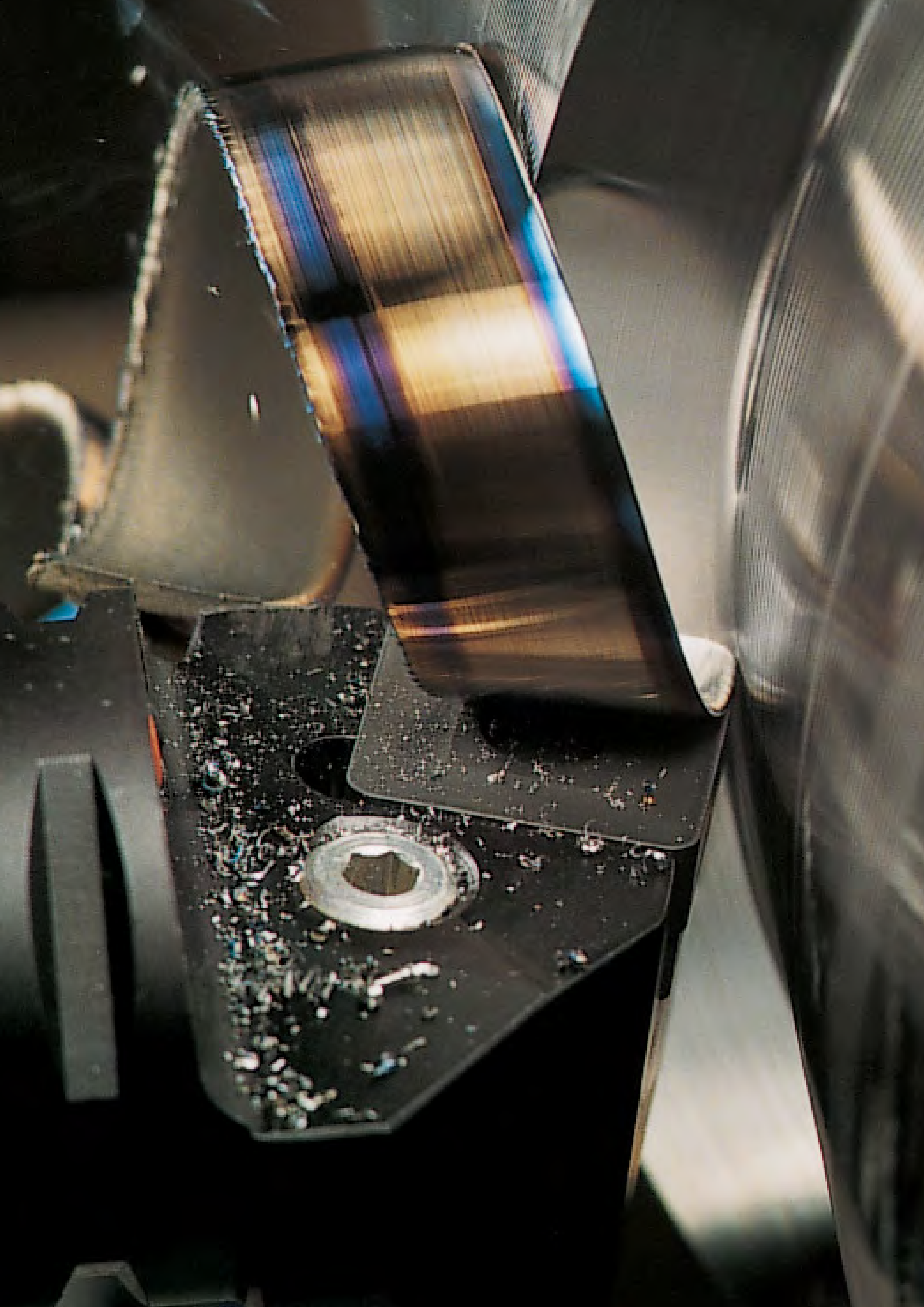
Unbalance in tool holders



Unbalance in tool holders causes:

- poor surface finish
- poor part tolerances
- reduction in tool life
- premature machine-spindle wear.


BIG-PLUS SYSTEM
8 Nm



Machinability

Matching the most suitable cutting tool material (grade) and insert geometry with the workpiece material to be machined is important for a trouble-free and productive machining process.

Other information

Workpiece materials

Six main groups

The ISO standard material groups are divided into six different types. Each type has unique properties regarding machinability and setups that make different demands on the tool.

ISO P Steel	ISO M Stainless steel	ISO K Cast iron
		
ISO N Non-ferrous	**ISO S Heat Resistant Super Alloys**	**ISO H Hardened steel**
		

P The largest variety of different types of components is probably in the P-area as it covers several different sectors in the industry.

N The aircraft industry and manufacturers of aluminum automotive wheels dominate the N-area.

M In the M-area, a big part of the application is in gas and oil, tubes, flanges, process industry and the pharmaceutical business.

S Difficult to machine S-area materials are found in the aerospace, gas turbine and power generator industries.

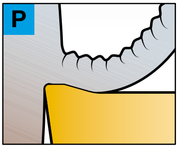
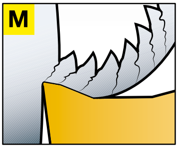
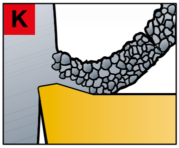
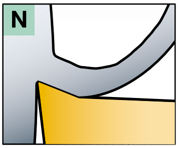
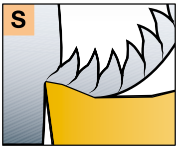
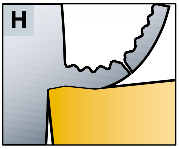
K The K-area is dominated by automotive components, the machine builders and the iron works production.

H Hardened steel in the H-area are seen in a variety of industries such as automotive and their subcontractors, as well as in machine builders and the die and mold business.

Characteristics for chip formation and removal

Factors that must be identified in order to determine a material's machinability:
- Classification, metallurgical/mechanical, of the workpiece material.
- The cutting edge micro and macro geometry to be used.
- The cutting tool material (grade), e.g. coated cemented carbide, ceramic, CBN, PCD, etc.

These selections will have the greatest influence on the machinability of the material at hand.

ISO P Steel	ISO M Stainless steel	ISO K Cast iron
		
ISO N Non-ferrous	**ISO S** Heat Resistant Super Alloys	**ISO H** Hardened steel
		

P ISO-P materials are generally long chipping and have a continuous, relatively even flow of chip formation. Variations usually depend on carbon content.
– Low carbon content = tough sticky material.
– High carbon content = brittle material.
Cutting force and power needed varies very little.

M ISO-M forms a lamellar, irregular chip formation where the cutting forces are higher compared to normal steel. There are many different types of stainless steels.
Chip breaking varies depending on the alloying properties and the heat treatment, from easy to almost impossible-to-break chips.

K Chip formation for ISO-K materials varies from near-powderlike chips to a long chip. The power needed to machine this material group is generally low.
Note that there is a big difference between gray cast iron (often near-powder) and ductile iron, which many times has a chip breaking more similar to steel.

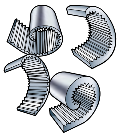
N Low power needed per mm^3 (inch^3), but due to the high metal removal rate, it is still a good idea to calculate the maximum power required.

S The range is wide, but in general high cutting forces are present.

H Often a continuous, red-glowing chip. This high temperature helps to lower the k_{c1} value and is important to help out with the application.

The complex world of metal cutting

Many parameters influence the cutting process



Workpiece material

P Steel

M Stainless steel

K Cast iron

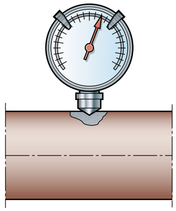
N Non-ferrous

S Heat resistant alloys

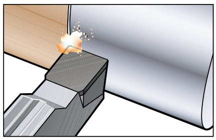
H Hardened steel

The ISO material groups are divided into 6 different types where each type has unique properties regarding machinability.

Hardness

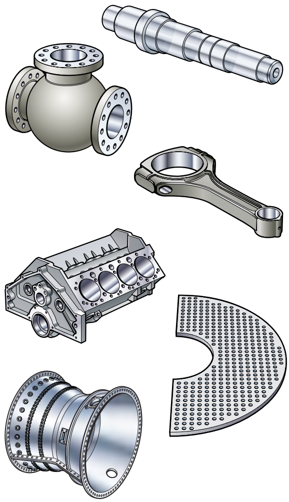


Hardness Brinell

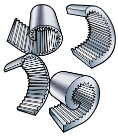


Usually there is a relation between material hardness and tool life, as well as machining data and type of geometry and grade. The higher the hardness, the shorter the tool life, with more rapid wear on the cutting edge.

Component



Depending on the type of material, set-up and way of machining, different choice of tooling is required to perform different applica-tions turning, milling, drilling etc.

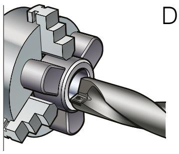


Application	Condition	Cutting environment

Application

Turning

Milling



Drilling

H R Roughing/ Heavy

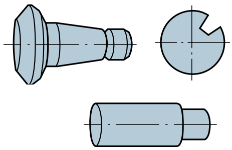
M M Medium

L F Finishing/ Light

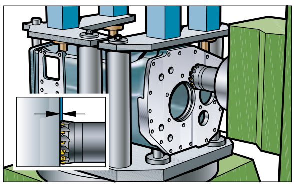
There are three major types of application, all requiring different tools, inserts and grades. These also depend on the load on the cutting edge, from finishing to roughing.

Condition

Cutting conditions



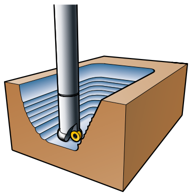
Clamping conditions



All components are different in look, shape and size. Some will need various set-ups and require special attention to the clamping conditions of the workpiece and cutting tool.

Cutting environment

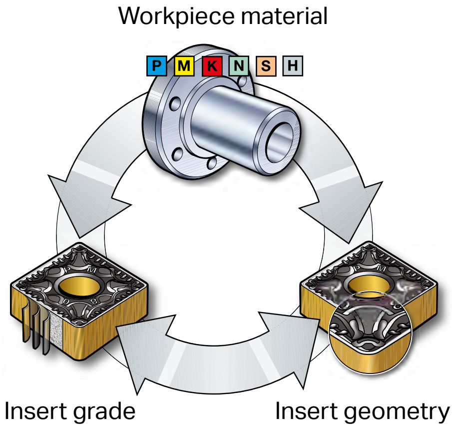
Coolant



Dry machining

Carbide performs best when machining at elevated temperatures, but needs to be constant. Dry conditions should therefore be considered first choice, depending on component requirements and machining conditions. However some grades are developed for both wet and dry conditions and used depending on component material and quality requirements.

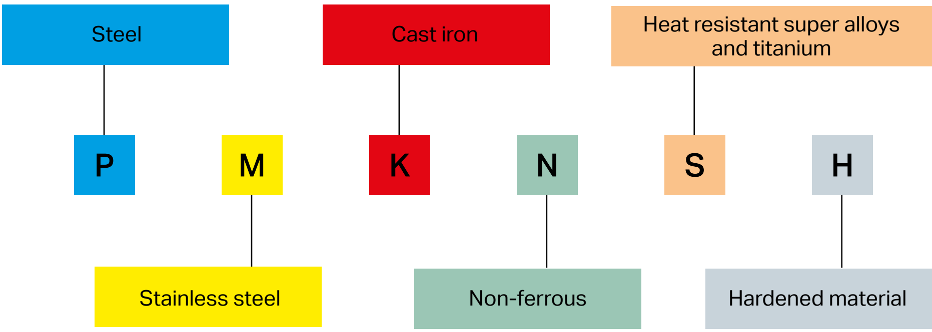
The interaction between workpiece material, geometry and grade



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

Workpiece materials, main groups

Materials are classified using MC codes



Within each material group there are subgroups depending on the hardness of the material, k_{c1} value, and metallurgical and mechanical properties.

* MC = A new material classification that replaces the CMC (Coromant Material Classification) codes.

MC code structure

The structure is set up so that the MC code can represent a variety of workpiece material properties and characteristics using a combination of letters and numbers.

Example 1:

The code P1.2.Z.AN is interpreted this way:

P = ISO code for steel

1 = material group: unalloyed steel

2 = material subgroup: carbon content ? 0.25% ≤0.55% C

Z = manufacturing process: forged/rolled/cold drawn

AN = heat treatment: annealed, supplied with hardness values

Example 2:

The code N1.3.C.UT is interpreted this way:

N = ISO code for non-ferrous metals

1 = material group: Aluminum alloys

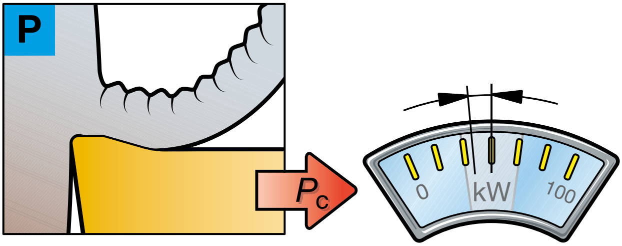
3 = material subgroup: non-ferrous with Si content 1-13%

C = manufacturing process: casting

UT = untreated

By describing not only the material composition, but also the manufacturing process and heat treatment, which influences the mechanical properties, a more exact description is available, which can be used to generate improved cutting data recommendations.

Steel ISO P – main characteristics



Machining characteristics:

- Long-chipping material.
- Relatively easy, smooth chip control.
- Low carbon steel is sticky and needs sharp cutting edges.
- Specific cutting force k_c: 1500–3100 N/mm^2 (217,500–449,500 lbs/inch^2).
- Cutting force, and the power needed to machine ISO P materials, stays within a limited range.

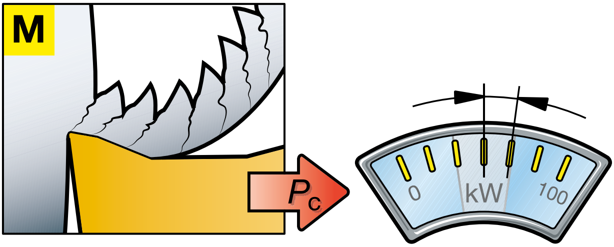
What is steel?

- Steel is the largest group in the metal cutting area.
- Steels can be non-hardened or hardened and tempered with hardness up to 400 HB.
- Steel is an alloy with the element iron (Fe) as the major component. It is produced through a melting process.
- Unalloyed steels have a carbon content lower than 0,8 %, and only Fe, with no other alloying elements.
- Alloyed steels have a carbon content which is lower than 1,7 % and alloying elements like Ni, Cr, Mo, V, W.

See product catalogs for details on MC codes.

ISO	MC	Material
P	P1	Unalloyed steel
	P2	Low-alloyed steel (≤5% alloying elements)
	P3	High-alloyed steel (>5% alloying elements)
	P4	Sintered steels

Stainless steel ISO M – main characteristics



Machining characteristics:

- Long-chipping material.
- Chip control is fair in ferritic, to difficult in austenitic and duplex.
- Specific cutting force: 1800–2850 N/mm² (261,000–413,250 lbs/inch²).
- Machining creates high cutting forces, built-up edge, heat and deformation hardening.

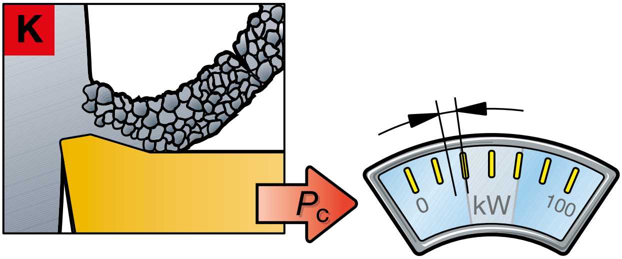
What is stainless steel?

- Stainless steels are materials alloyed with min 11–12% chromium.
- The carbon content is often low (down to max 0.01%).
- Alloys are mainly Ni (Nickel), Mo (Molybdenum), and Ti (Titanium).
- The formed Cr_2O_3 layer on the steel surface makes it non-corrosive.

See product catalogs for details on MC codes.

ISO	MC	Material
M	P5	Ferritic/Martensitic stainless steel
	M1	Austenitic stainless steels
	M2	Super-austenitic, Ni≥20%
	M3	Duplex (austenitic/ferritic)

Cast iron ISO K – main characteristics



Machining characteristics:

- Short chipping material.
- Good chip control in all conditions.
- Specific cutting force: 790–1350 N/mm² (114,550–195,750 lbs/inch²).
- Machining at higher speeds creates abrasive wear.
- Moderate cutting forces.

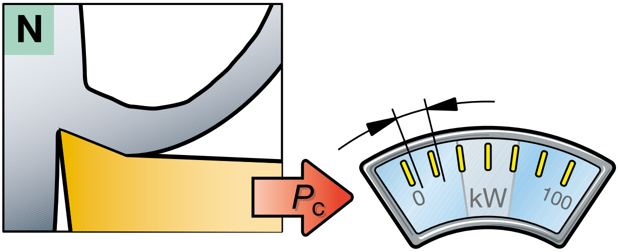
What is cast iron?

- There are 3 main forms of cast iron: gray (GCI), nodular (NCI) and compacted graphite (CGI).
- Cast iron is an Fe-C composition with relatively high content of Si (1–3%).
- Carbon content is over 2% which is the max solubility of C in the Austenitic phase.
- Cr (Chromium), Mo (Molybdenum), and V (Vanadium) form carbides which increase strength and hardness, but lower machinability.

See product catalogs for details on MC codes.

ISO	MC	Material
K	K1	**Malleable cast iron**
	K2	**Gray cast iron**
	K3	**Nodular SG iron**
	K4	**Compacted graphite iron**
	K5	**Austempered ductile iron**

Non-ferrous materials ISO N – main characteristics



Machining characteristics:

- Long-chipping material.
- Relatively easy chip control if alloyed.
- Non-ferrous (Al) is sticky and needs sharp cutting edges.
- Specific cutting force: 350–700 N/mm2 (50,750–101,500 lbs/inch2).
- Cutting force, and the power needed to machine ISO N materials, stays within a limited range.

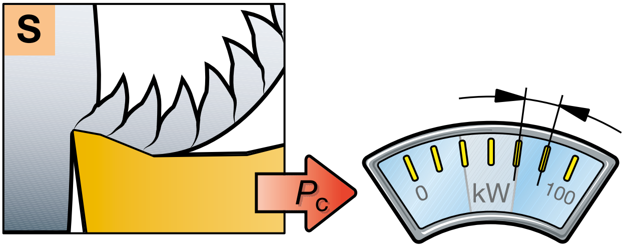
What is Non-ferrous material?

- This group contains non-ferrous, soft metals with hardness under 130 HB.
- Non-ferrous (Al) alloys with up to 22% silicon (Si) make up the largest part.
- Copper, bronze, brass.
- Plastic.
- Composites (Kevlar).

ISO	MC	Material
N	N1	Non-ferrous-based alloys
	N2	Magnesium-based alloys
	N3	Copper-based alloys
	N4	Zinc-based alloys

See product catalogs for details on MC codes.

Heat resistant super alloys and titanium ISO S – main characteristics



Machining characteristics:

- Long-chipping material.
- Difficult chip control (segmented chips).
- Negative rake angle is required with ceramics, a positive rake angle with carbide.
- Specific cutting force:
  For HRSA:
  2400–3100 N/mm^2
  (348,000–449,500 $lbs/inch^2$).

  For titanium:
  1300–1400 N/mm^2
  (188,500–203,000 $lbs/inch^2$).
- Cutting forces, and power required are quite high.

What are Heat Resistant Super Alloys?

- Heat Resistant Super Alloys (HRSA) include a great number of high alloyed iron, nickel, cobalt or titanium based materials.

Groups: Fe-based, Ni-based, Co-based

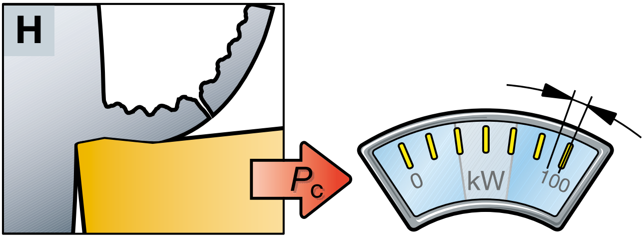
Condition: Annealed, Solution heat treated, Aged rolled, Forged, cast.

Properties:
- Increased alloy content (Co more than Ni), results in better resistance against heat, increased tensile strength and higher corrosive resistance.

ISO	MC	Material
S	S1	Iron-based alloys
	S2	Nickel-based alloys
	S3	Cobalt-based alloys
	S4	Titanium-based alloys
	S5	Tungsten-based alloys
	S6	Molybdenum-based alloys

See product catalogs for details on MC codes.

Hardened steel ISO H – main characteristics



Machining characteristics:

- Long-chipping material.
- Fair chip control.
- Negative rake angle is required.
- Specific cutting force: 2550–4870 N/mm² (369,750–706,150 lbs/inch²).
- Cutting forces and power required are quite high.

What is hardened steel?

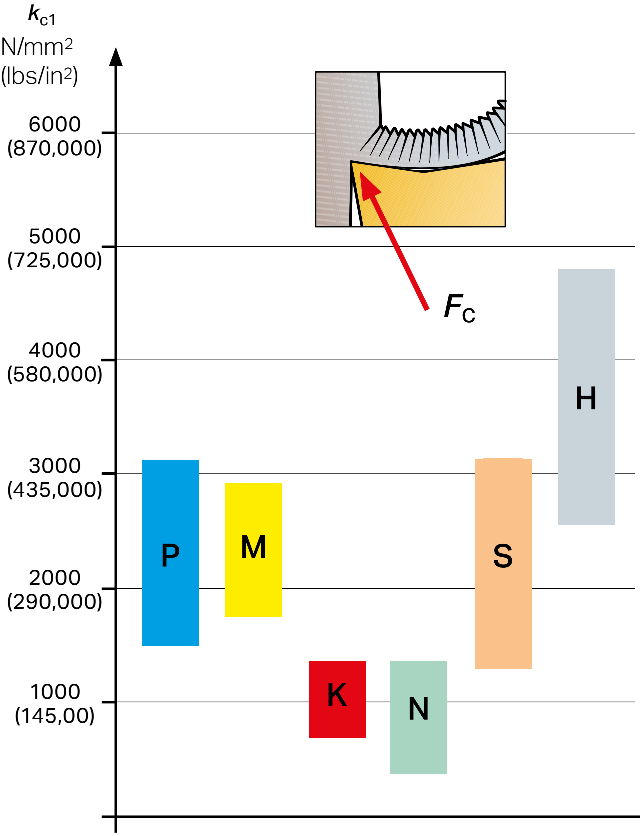
- Hardened steel is the smallest group from a machining point of view.
- This group contains hardened and tempered steels with hardness >45–65 HRC.
- Typically, however, hard part turned components can be found to be within the range of 55–68 HRC.

See product catalogs for details on MC codes.

ISO	MC	Material
H	H1	Steels (45-65 HRC)
	H2	Chilled cast iron
	H3	Stellites
	H4	Ferro-TiC

The specific cutting force k_{c1}

k_{c1} – the tabulated value of k_c for 1 mm (.0394") chip thickness



- The cutting force (F_c) is the force needed to shear off a specific chip cross-section in certain conditions
- This value (F_c) is used in the calculation of the power consumption needed for an operation
- The specific cutting force value (k_{c1}) is a material constant, expressed in N/mm2 or lbs/inch2

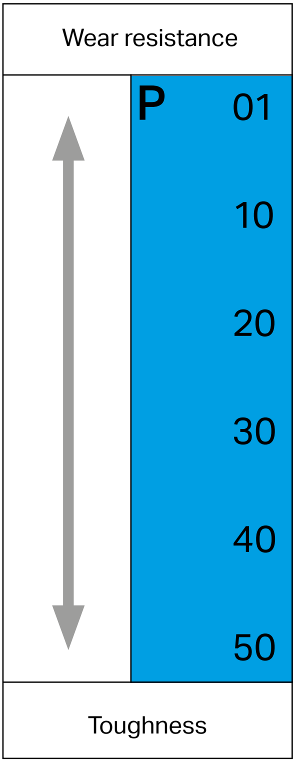
See formulas section on specific calculations.

k_{c1} values in N/mm2 (lbs/inch2)

P	1500 – 3100 (217,500 – 449,500)	N	350 – 1350 (50,750 – 195,750)
M	1800 – 2850 (261,000 – 413,250)	S	1300 – 3100 (188,500 – 449,500)
K	790 – 1350 (114,550 – 195,750)	H	2550 – 4870 (369,750 – 706,150)

The ISO nomenclature in the ISO-P area

Operations and working conditions



The above diagram is related to the ISO P area. These demands also apply to all other ISO types of material, i.e., M, K, N, S, H.

P01: Internal and external finishing turning; high cutting speed; small chip area; good surface finish; narrow tolerances; no vibrations.

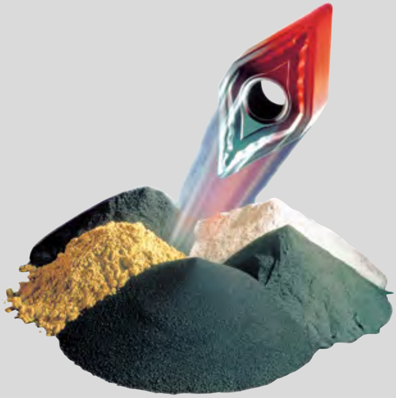
P10: Turning; copying; threading; milling; high cutting speed; small to medium chip area.

P20: Turning; copying; medium cutting speed; facing with small chip area; medium to difficult conditions.

P30: Turning; milling facing; medium to low cutting speed; medium to large chip area; includes operations with tough conditions.

P40: Turning; facing; milling; cutting; grooving; low cutting speed; large chip area; large possible chip angle; very tough conditions.

P50: When very high toughness in the tool is needed in turning, facing, grooving, cutting, low cutting speed, large chip area, large possible chip angle, extremely tough conditions.



Manufacture of cemented carbide

The manufacture of cemented carbide inserts is a carefully designed process, where geometry and grade are balanced to give a product perfectly matched to the application.

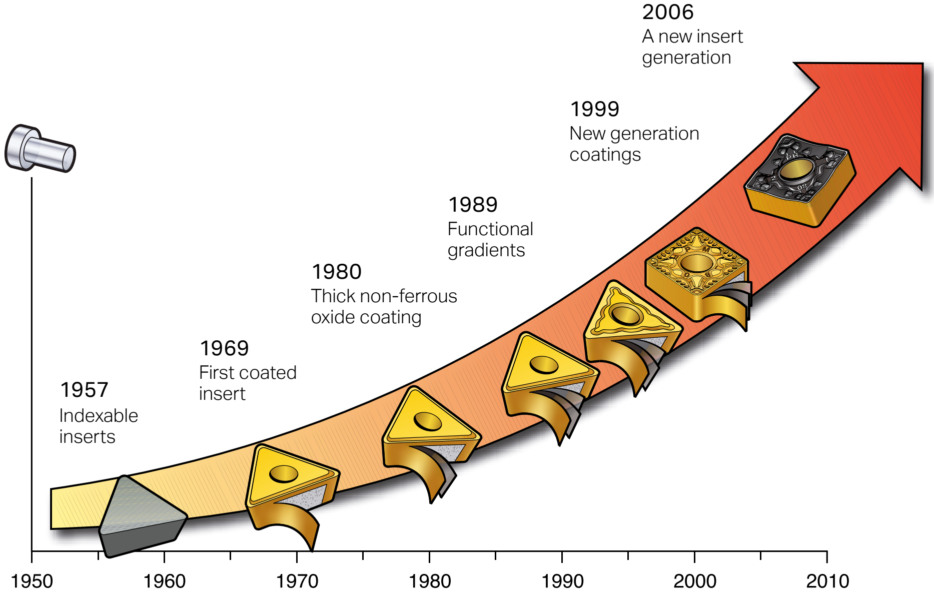
The development of cutting tool material

With the development of better carbide substrates, coatings and geometries, productivity and cost savings have improved for the end user.

Large improvements in productivity were possible in the 60s and 70s when the first coatings were developed.

After this, the developments continued - with advanced substrate design, new geometries, edge designs, new advanced coating techniques and post treatment of coated edges.

The effect on end-user productivity



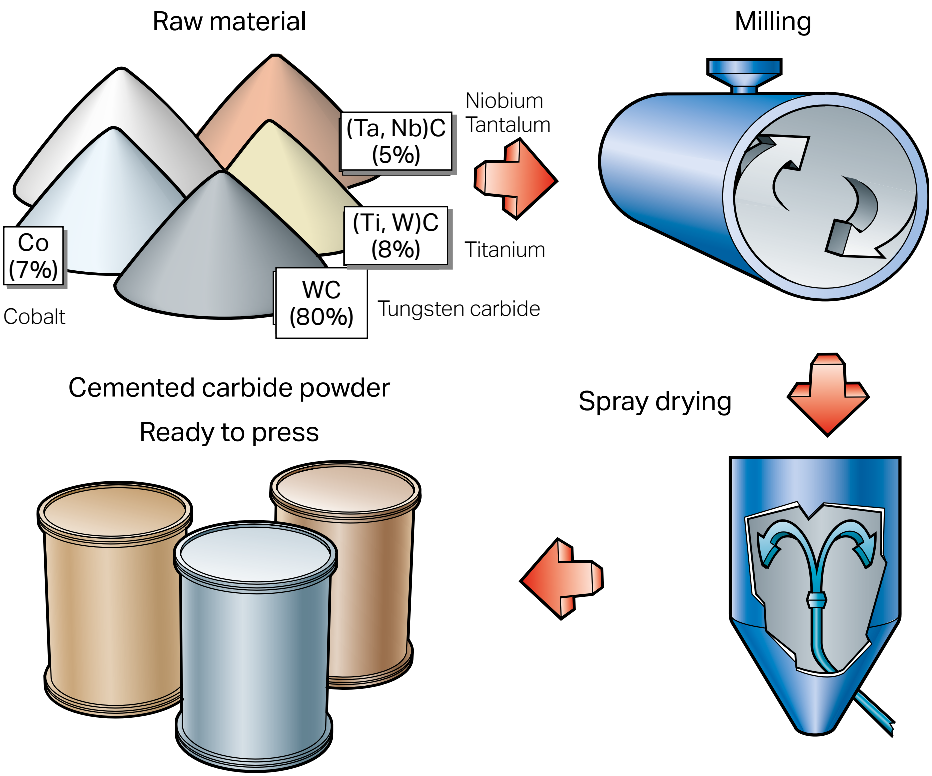
Powder production

There are two main elements of a cemented carbide insert:

- Tungsten Carbide (WC)
- Cobalt (Co)

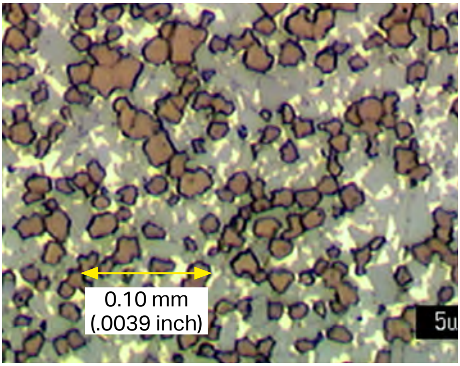
Other commonly used elements are Titanium, Tantalum and Niobium Carbides. Designing different types of powder and different percentages of the elements is what makes up the different grades.

The powder is milled and sprayed-dried, sifted and poured into containers.



Tungsten powder

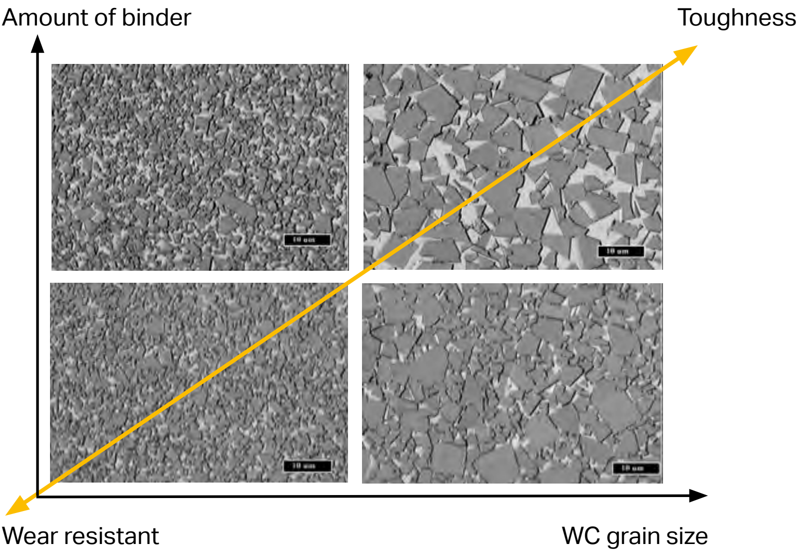
The size of the tungsten carbide grains



The main raw material for the manufacture of cemented carbide is tungsten-ore concentrate. Tungsten powder is produced from tungstic oxide derived chemically from the raw material. By varying the conditions of reduction, tungsten powder of various grain size can be manufactured. The carbide granules after spray-drying are small and vary in size depending on grade.

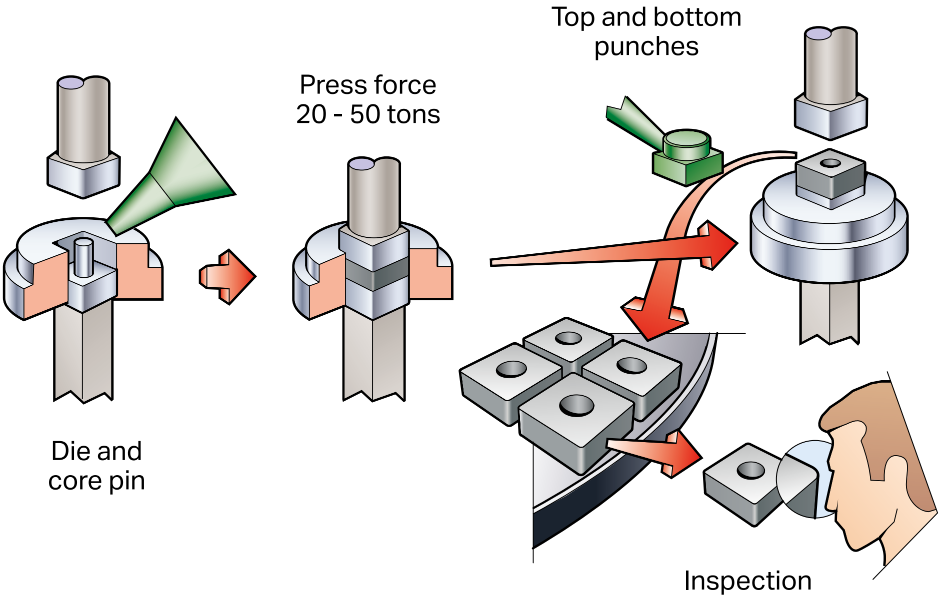
Basic properties of cemented carbide

Apart from the grain size for the Tungsten carbide (WC), the amount of binder phase is an important factor determining the characteristics of the carbide. Increasing Cobalt-content, together with increasing WC-grain size, contributes to increasing toughness but also to a lower hardness which reduces the wear resistance of the substrate.



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

Pressing powder compacts



The pressing operation consists of several pieces of tooling:

- Top and bottom punches
- Core pin
- Cavity.

The pressing procedure:

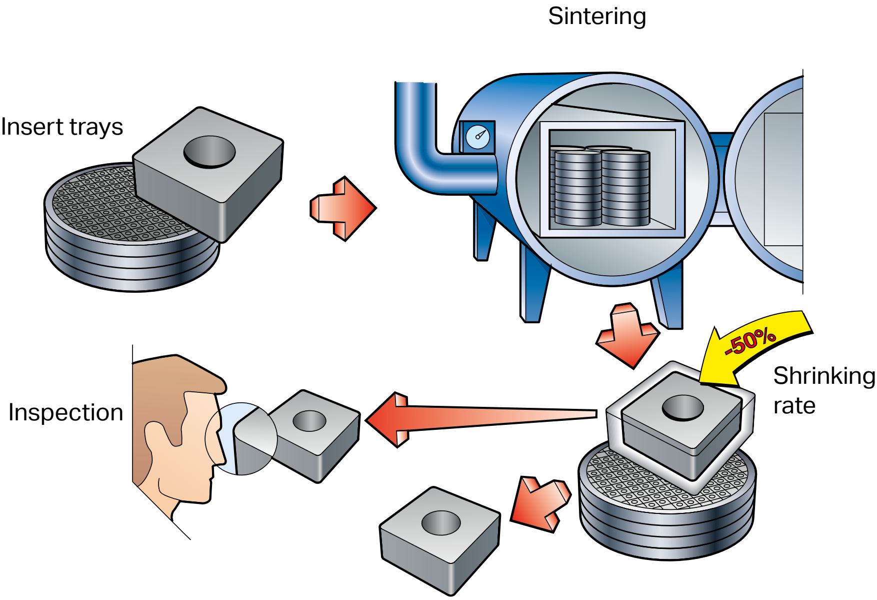
- Powder is poured into the cavity
- Top and bottom punches come together (20-50 tons)
- The insert is picked and placed via robot onto a graphite tray
- Random SPC is performed, to check for weight.

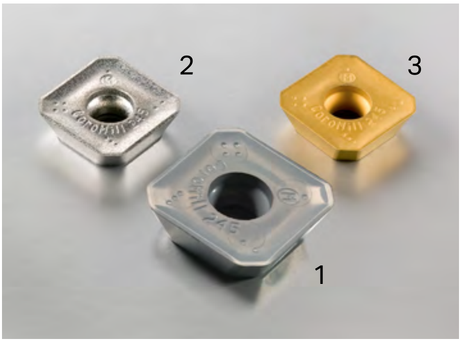
The insert is 50% porous at this stage.

Sintering the pressed inserts

Sintering consists of the following:

- Loading trays of inserts into a sintering furnace.
- The temperature is raised to ~1400° C (~2550° F) .
- This process melts the cobalt and the cobalt acts as a binder.
- The insert will shrink 18% in all directions during the sintering; this corresponds to about 50% in volume.

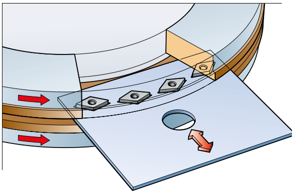




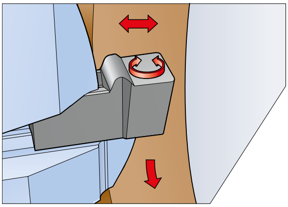
1. Unsintered insert
2. Sintered insert
3. Coated insert

Different types of grinding operations

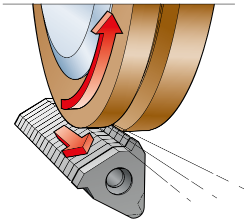
Top and bottom



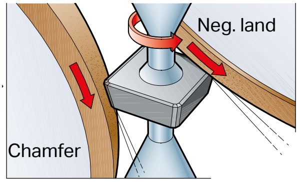
Free profiling



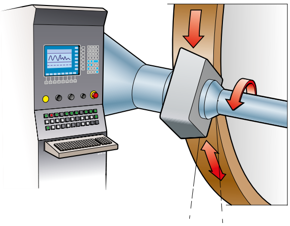
Profiling



Chamfer – negative land

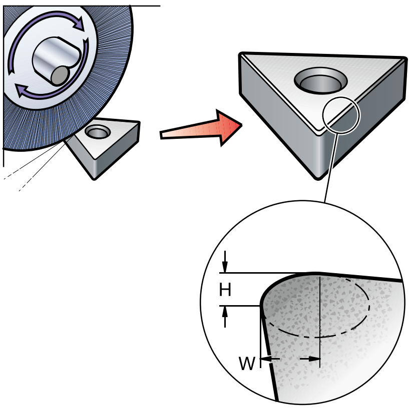


Periphery



The reinforcement of the cutting edge

The ER-treatment gives the cutting edge the final micro-geometry.

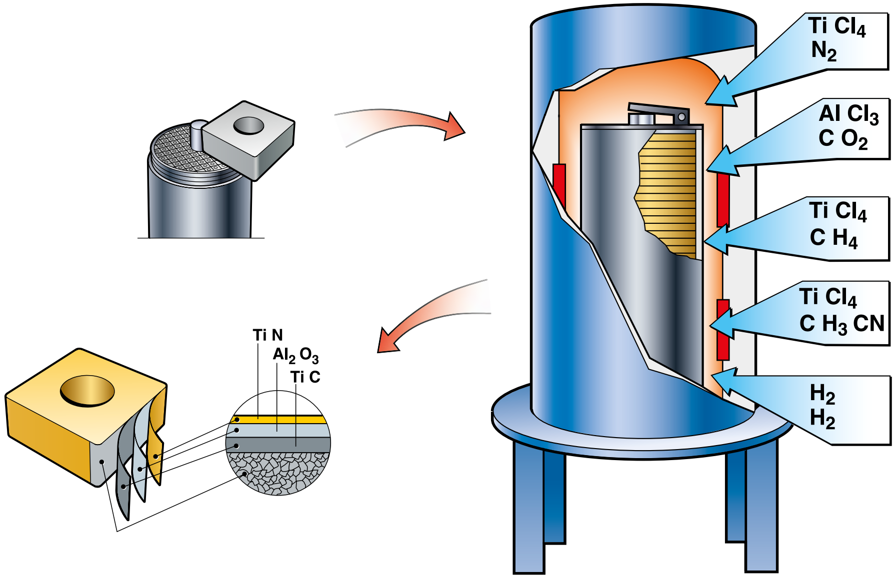


- ER-treatment (Edge Roundness) is done before coating.
- The relation between W/H depends on the application.

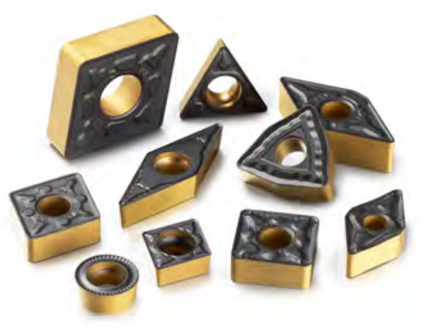
Generally the ER corresponds to the thickness of a hair, diameter: ~80 µm (~.0031 inch).

CVD – Chemical Vapor Deposition

Stacks of inserts are placed into a furnace, a series of gases are introduced to the chamber, lines are purged and another series of gases introduced. This is repeated until the layers of coating are complete. The process is carried out at approx. 900° C (1650° F) for 30 hours. Thickness is approx 2-20 microns (.00008-.0008 inch).



The advantages of CVD coatings

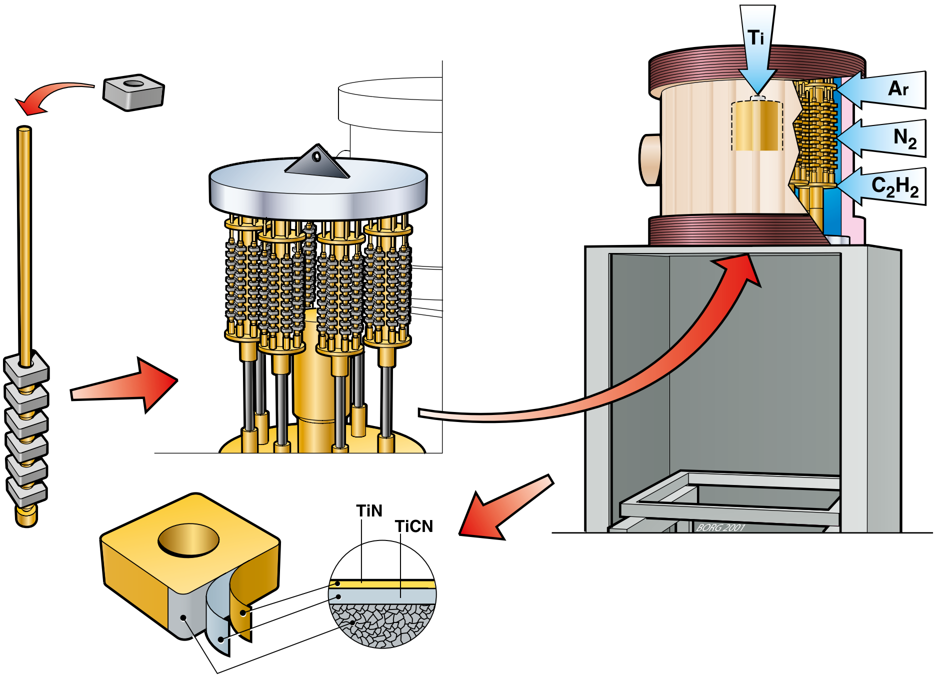


- The ability to making thick coatings.
- Ability to make even coating thickness.
- Very good adherence to the carbide substrate.
- Very good wear resistance.
- Possibility to make oxide coatings.

PVD – Physical Vapor Deposition

The inserts are loaded into the coating chamber on trays. Metal source targets are placed on the reactor chamber walls. The most common source is titanium (Ti). The targets are heated to a temperature where the solid metal ionizes.

By using a gas as carrier, the ions can then be transported from the targets to the inserts. As the inserts are cooler, the ions will condensate on the insert surface to form a coating.



The coating thickness is in the range of 2-6 microns (.00008-.0002 inch) depending on application area for the insert.

The most common PVD layers today are TiN, Ti(C,N), (Ti,Al)N, (Ti,Al,Cr)N and now also non-ferrous oxides.

The advantages of PVD coating

- PVD provides good edge line toughness.
- PVD coatings can maintain a “sharp” cutting edge.
- PVD can be used on brazed tips.
- PVD can be used on solid carbide tools.

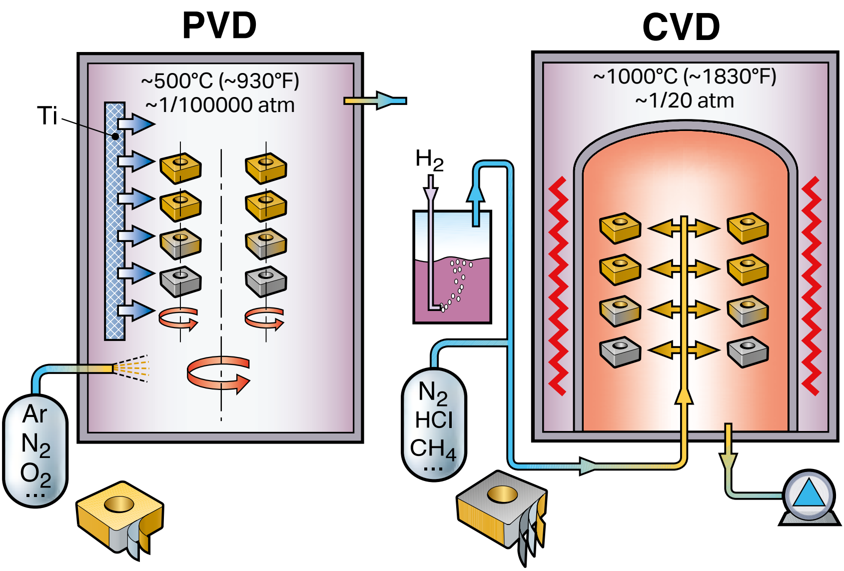
PVD vs. CVD coating process

PVD (Physical Vapor Deposition)

In a PVD coating process, the coating is formed by metal vapor condensating on insert surfaces. PVD works the same way as when humid air condensates on cold roads and forms an ice layer on the road. PVD is formed at a much lower temperature than CVD. Normal PVD process temperatures are around 500° C (930° F). The coating thickness is in the range of 2-6 microns (.00008-.0002 inch) depending on application area for the insert.

CVD (Chemical Vapor Deposition)

In a CVD coating process, the coating is formed by a chemical reaction of different gases. Temperature, time, gas flow, gas atmosphere, etc., are carefully monitored to steer the deposition of the coating layers. Depending on the type of coating, the temperature in the reactor is about 800 to 1100 degrees C (1470 to 2000 degrees F). The thicker the coating the longer the process time. The thinnest CVD coating today is below 4 microns (.00016 inch) and the thickest is above 20 microns (.0008 inch).

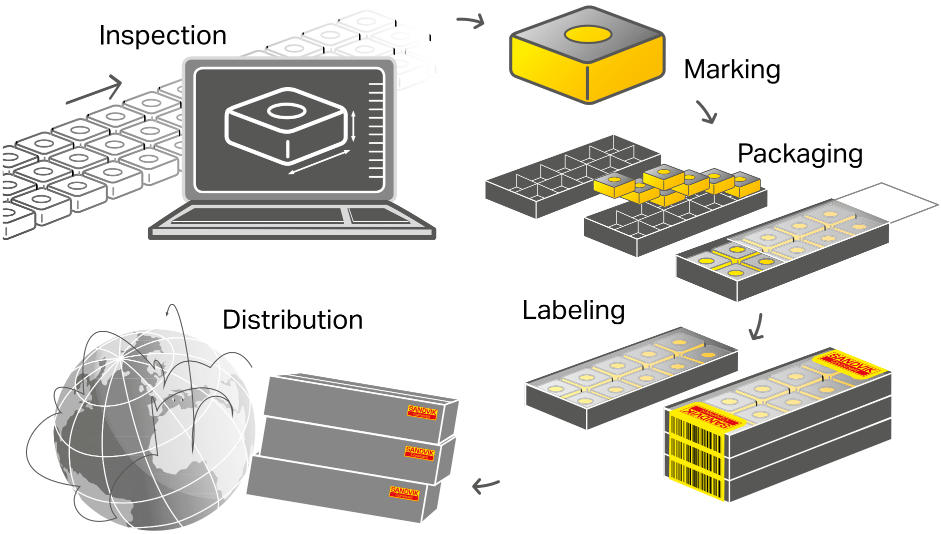


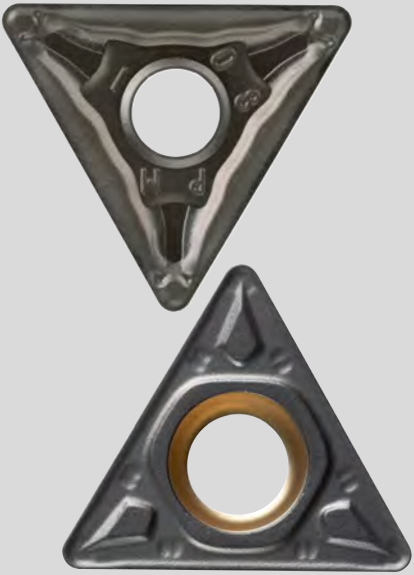
- Thinner coating
- Sharper edges
- Tougher

- Thicker coating
- More wear resistant
- Thermal resistant

Vision control, marking and packaging

Before being packaged, each insert is inspected again and compared with th blueprints and batch rder. A laser marks the insert with the correct grade, and it's placed in a grey box with a printed label. It's now ready to be distributed to customers.





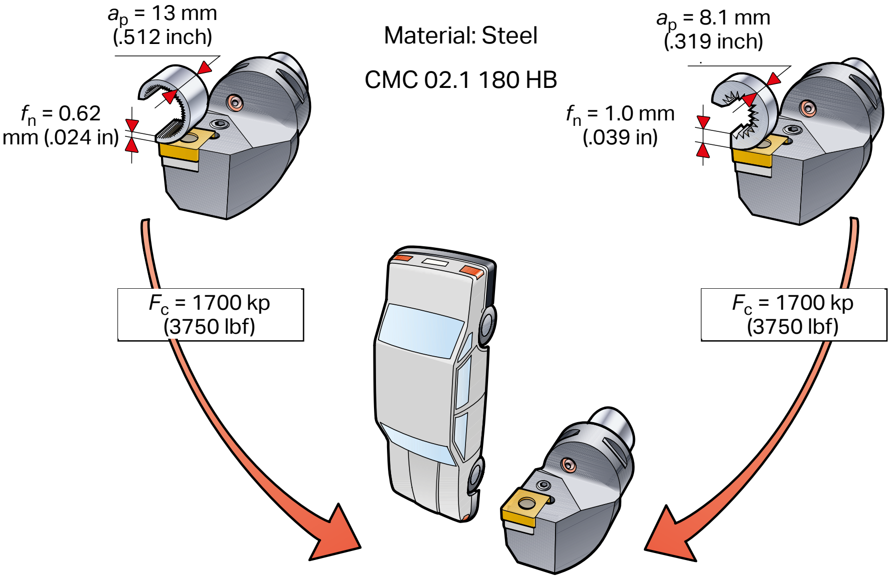
The cutting edge

The design of the cutting edge and insert geometry is of vital importance for the chip formation, tool life and feed rate data in metal cutting process.

The high cutting force on a cutting edge

Cemented carbide has a high compressive strength resistance and can also work at high temperatures without plastic deformation. It can also resist high cutting forces (F_c) without breaking, as long as the insert is well supported.

In order to understand the tough environment of the cutting edge, you can find two different cutting data conditions for a cutting unit below. They generate about the same cutting force (F_c) on the cutting edge.



The cutting force in these two cases is equivalent to the weight of a passenger car.

Calculation of F_c Material: MC P2 (low alloyed steel) 180 HB
Specific cutting forces k_{c1} = 2100N/mm² (304,563 lbs/in²)

$F_c = k_{c1} \times a_p \times f_n$
F_c = 2100N/mm² x 13 mm x 0.62 mm = 16926 Newton (N) = 1700 kp
F_c = 304,563 lbs/in² x .512" x .024" = 3742 pound force (lbf) = 1700 kp

1 lbf = 0.4535 kilogram force (kg),
1N = 0.101 kg
kp = kilopond or kilogram force

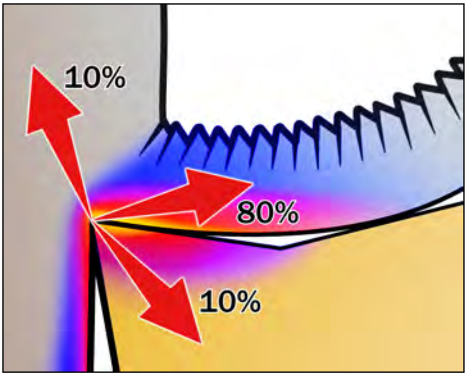
The machining starts at the cutting edge

Typical chip breaking sequences with high speed imaging.

Cutting zone temperatures

The maximum heat generated during cutting is on the top part of the insert, 1000° celsius (1832° fahrenheit), in the chip breaker, and close to the cutting edge.

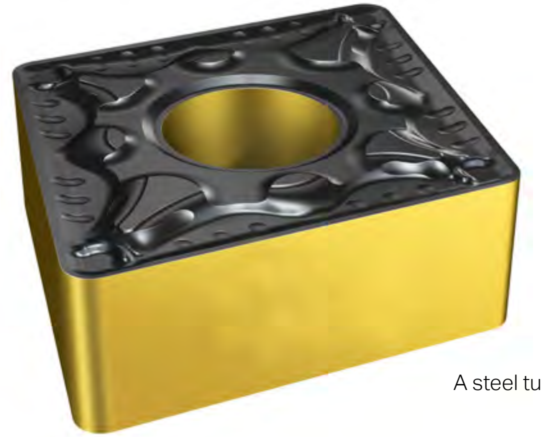
This is where the maximum pressure from the material is, and, with the friction between chip and carbide, causes these high temperatures.



- The 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 factor.
- The rest of the heat is usually evenly distributed between the workpiece and the tool.

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

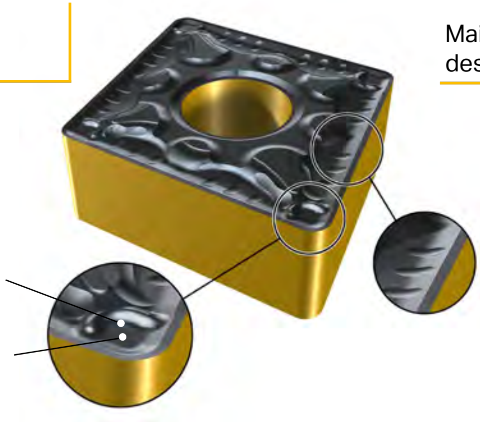
The design of a modern insert

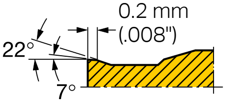


A steel turning insert for medium turning.

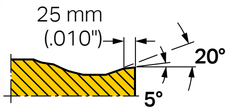
Definitions of terms and geometry design

Nose cutting edge design





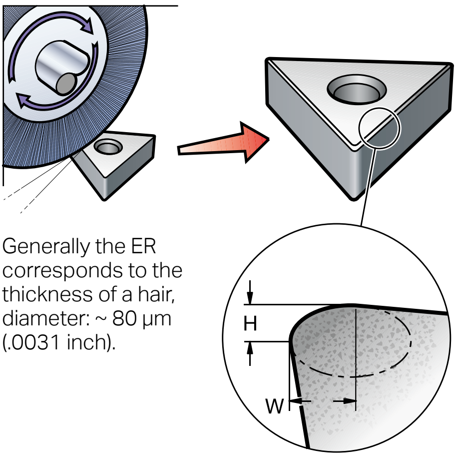
Main cutting edge design



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

The reinforcement of the cutting edge

The ER treatment gives the cutting edge the final micro-geometry

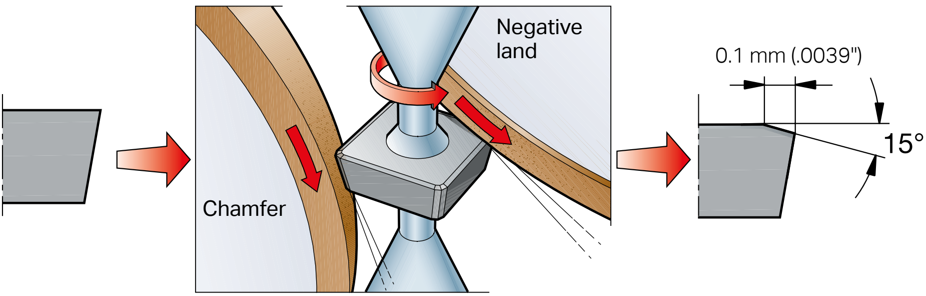


Generally the ER corresponds to the thickness of a hair, diameter: ~ 80 µm (.0031 inch).

- ER treatment (Edge Roundness) is done before coating, and gives the final shape of the cutting edge (micro-geometry).
- The relation between W/H depends on the application.

A negative land increases the strength of the cutting edge

In some cases inserts have a negative land and reinforced insert corners, making them stronger and more secure in the intermittent cutting action.

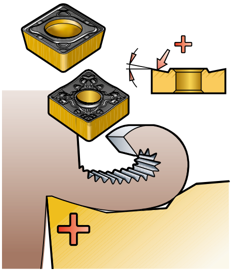


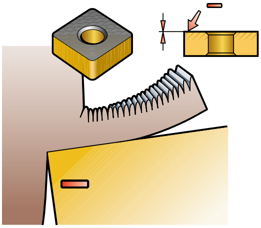
- A negative land increases the strength of the cutting edge, but also creates higher cutting forces.

Insert rake angle

The rake angle can be either negative or positive.

Based on that, 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, and results in either a negative or positive cutting action.





- The insert rake angle is the angle between the top face of the insert and the horizontal axis of the workpiece.

Positive and negative cutting action

Turning needs a durable edge that can perform for a long time and often in continuous cuts at high temperature. This condition requires an edge with among other things good chip breaking ability, good resistance against different types of wear and against plastic deformation.

In milling, which always has an intermittent cutting action, the edge needs to have good bulk strength to resist breakage. A large variation in cutting edge temperature due to interrupted cuts also makes resistance to thermal cracks of vital importance.

In drilling, the edge must be strong enough to last at very low cutting speeds, and even at zero speed in the center of the drill.

In most drilling applications there is also coolant present, mainly for chip transportation reasons which puts the edge under extra stress from temperature variations. To be able to transport the chips from the narrow chip flutes and from inside the hole, good chip breaking into short chips is an important factor.

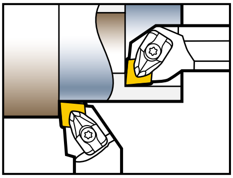
Peak performance in machining

Dedicated inserts for different applications

There are major differences in insert geometry and grade requirements between applications in turning, milling and drilling.

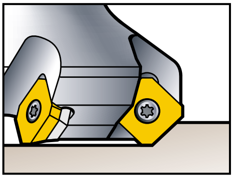
P
M
K
N
S
H

Turning



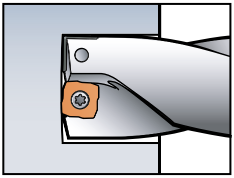
- Needs a durable edge that can perform for a long time, and often in continuous cuts at high temperature.
- Good chip breaking ability.
- Good resistance against different types of wear and against plastic deformation.

Milling



- The cutting action is always intermittent and the edge needs to have good bulk strength to resist breaking.
- Variations in cutting edge temperature due to the interrupted cuts also mean that the resistance to thermal cracks is of vital importance.

Drilling

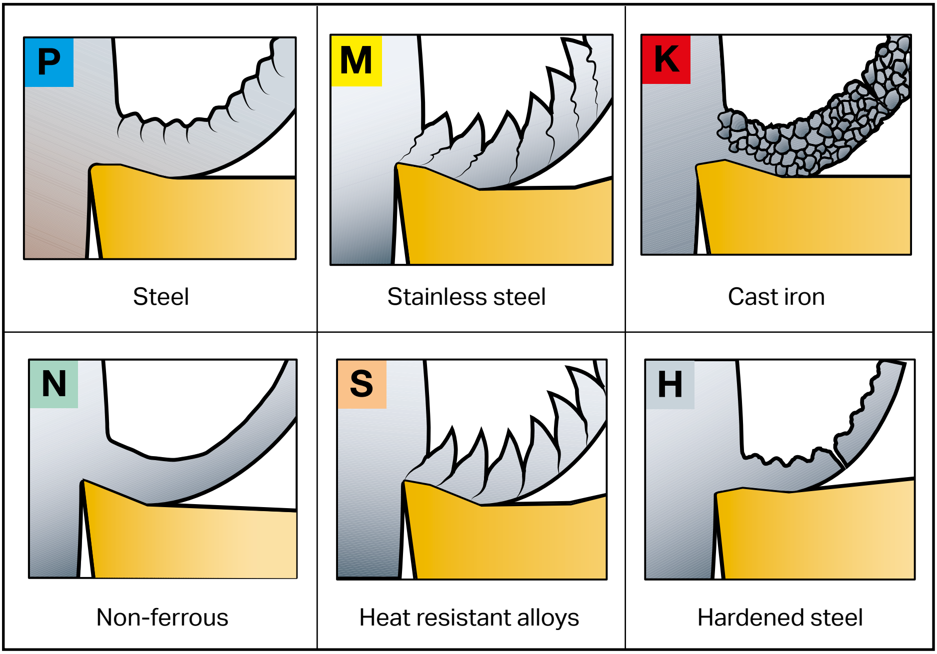


- The edge must be strong enough to last at very low cutting speeds; in fact, at zero speed in the center of the drill.
- Coolant is present, mainly for chip transportation reasons, which puts the edge under extra stress from temperature variations.
- To transport the chips from the narrow chip flutes and from inside the hole, good chip breaking is important.

Six main groups of workpiece materials

Different characteristics for removing chips

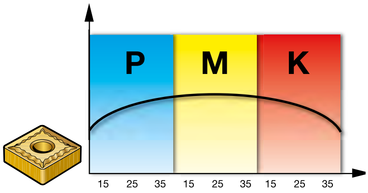
Good chip forming usually results in high cutting forces and excess heat, depending on the material. This can lead to low cutting speeds with adhesive stresses as a result. On the other hand, materials like non-ferrous, unalloyed steels and low-strength cast iron produce less cutting force.



From universal to optimized turning inserts

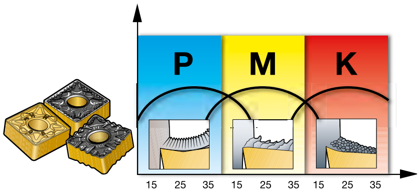
General inserts

- General geometry
- Optimizing with grades
- Performance compromised



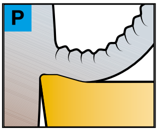
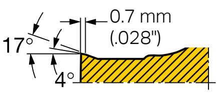
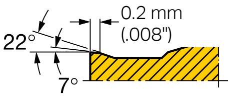
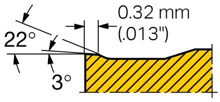
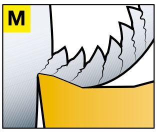
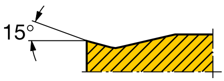
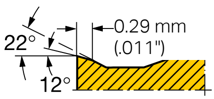
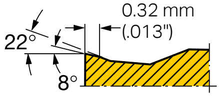
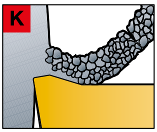
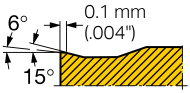
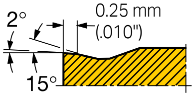
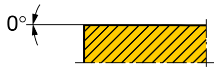
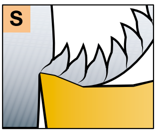
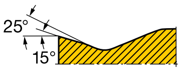
Dedicated inserts

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

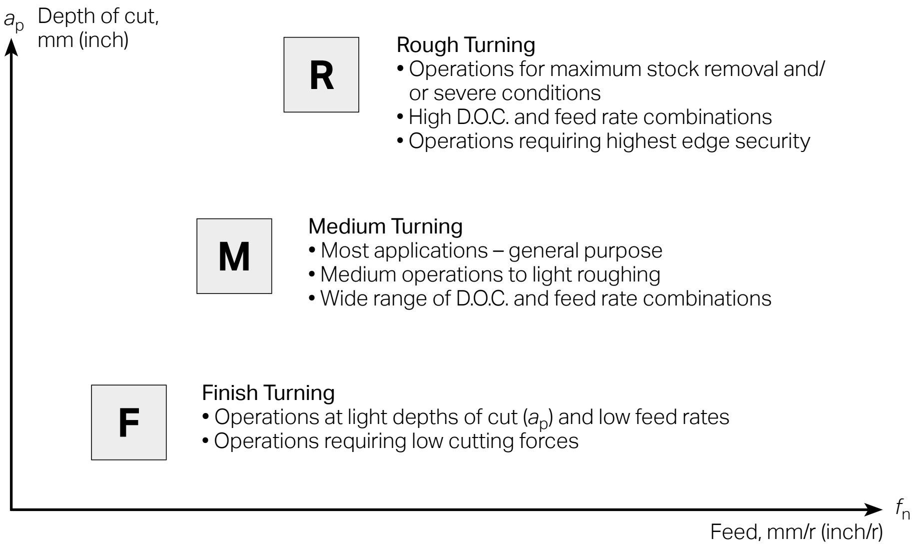


Dedicated inserts for the ISO P, M, K and S areas

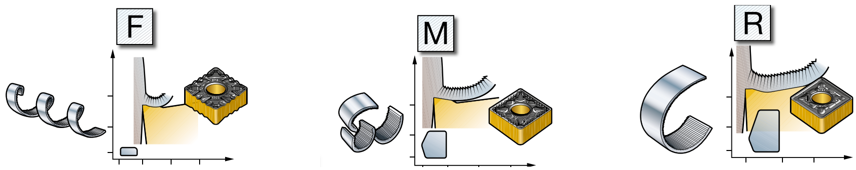
The different micro- and macro-geometries are adapted to the various requirements in the applications and materials.

Workpiece material	Finishing	Medium	Roughing
			
			
			
			

Type of application - Turning



Selecting the insert geometry in turning



Finishing (F)

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

Medium (M)

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

Roughing (R)

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

Type of application - Milling

a_p Depth of cut, mm (inch)

H

Heavy Milling
- Operations at maximum stock removal and/or severe conditions
- Larger depth of cut and feed rate
- Operations requiring highest edge security

M

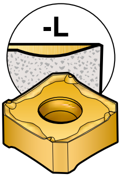
Medium Milling
- Most applications – general purpose milling
- Medium operations to light roughing
- Medium depth of cut and feed rate

L

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

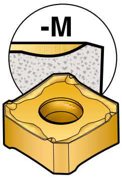
Feed f_z, mm/tooth (inch/tooth)

Selecting the insert geometry in milling



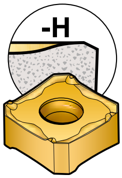
Light (-L)

- 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)

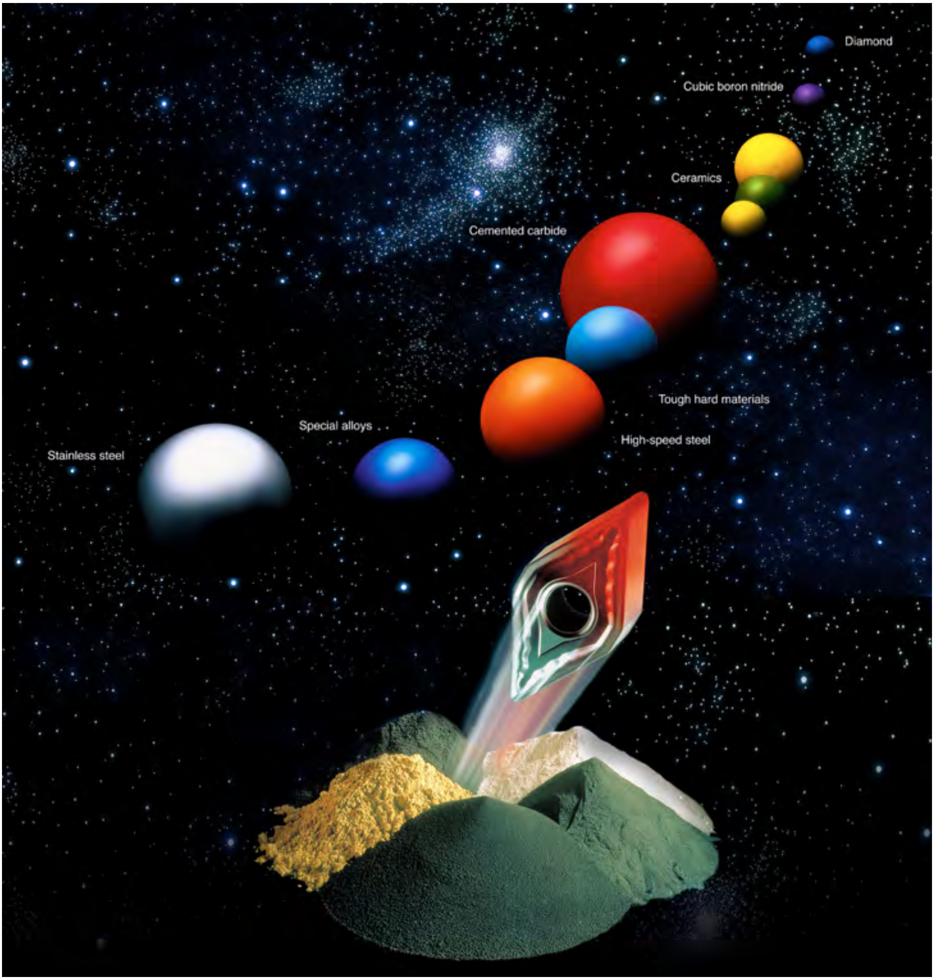
- Reinforced cutting edge
- Heavy machining
- Highest edge security
- High feed rates.

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

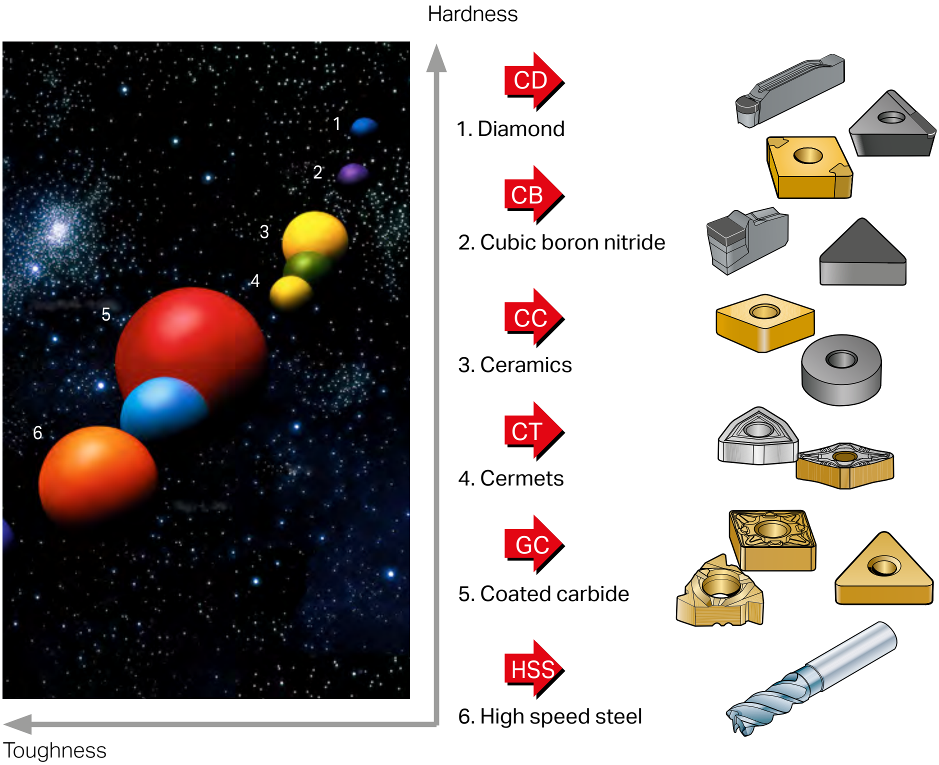
Cutting tool materials

The selection of cutting tool material and grade is an important factor to consider when planning a successful metal cutting operation.

A basic knowledge of each cutting tool material and its performance is therefore important to be able to make the correct selection for each application. This should take into consideration the workpiece material to be machined, the component type and shape, machining conditions and the level of surface quality required for each operation.



Different types of cutting tool materials



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

- Uncoated cemented carbide (HW)
- Coated cemented carbide (HC)
- Cermet (HT, HC)
- Ceramic (CA, CN, CC)
- Cubic boron nitride (BN)
- Polycrystalline diamond (DP, HC)

- (HW) Uncoated hard metal containing primarily tungsten carbide (WC).
- (HT) Uncoated hard metal, also called cermet, containing primarily titanium carbides (TIC) or titanium nitrides (TIN) or both.
- (HC) Hard metals as above, but coated.
- (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.
- (DP) Polycrystalline diamond [1]
- (BN) Cubic boron nitride [1]

[1]) Polycrystalline diamond and cubic boron nitride are also called superhard cutting materials.

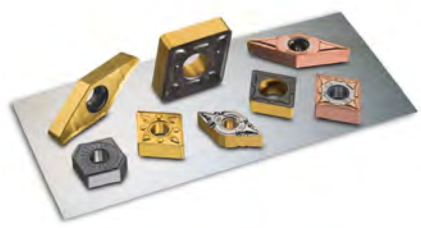
Uncoated cemented carbide

Characteristics, features and benefits

- Used in moderate to difficult applications related to steel, HRSA, titanium, cast iron and non-ferrous in turning, milling and drilling.
- Good combination of abrasive wear resistance and toughness.
- Gives sharp cutting edges.
- Good edge security but limited wear resistance at higher speeds.
- Represents a small portion of the total grade program.

Coated cemented carbide

Characteristics, features and benefits

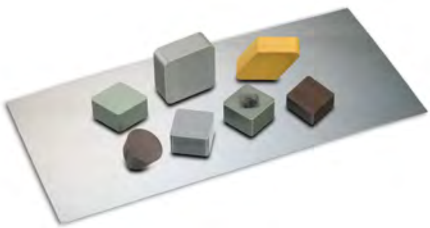


- General use in all kinds of components and materials for turning, milling and drilling applications.
- Extremely good combination of wear resistance and toughness in a variety of jobs.
- Consists of a large variety of grades with hard to tough substrates, usually with gradient sintering, and various coatings of CVD and PVD-type.
- Shows very good wear characteristics with long tool life.
- Dominates the insert program, with increasing share.

Cermet

- Used in finishing and semi-finishing applications where close tolerance and good surface finish is required.
- Chemically stable with a hard and wear resistant substrate.
- Consists of Titanium based (TiC, TiCN) cemented carbide with cobalt as a binder.
- PVD-coating adds wear resistance and tool life. "Self sharpening " properties. Limited toughness behavior.
- Quite low share of total insert program.

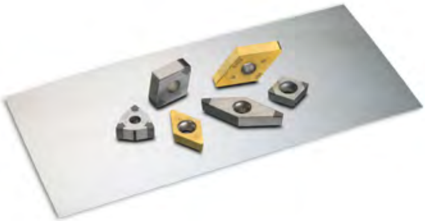
Ceramic



- Depending on type of ceramic, the grades are mainly used in cast iron and steel, hardened materials and HRSA.
- Ceramic grades are generally wear resistant and with good hot-hardness. Wide application area in different types of material and component.
- Ceramics are considered brittle and need stable conditions. With additions in the mix and whisker reinforced ceramic, toughness is improved.
- Fairly low share of total insert usage, but increased usage in the aerospace and hardened steel-cast iron areas.

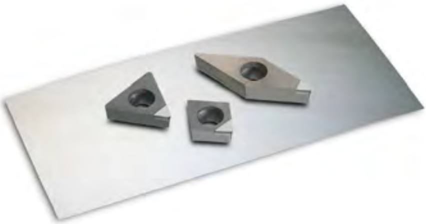
Cubic boron nitride

Characteristics, features and benefits



- For finish turning of hardened steel. Roughing of gray cast iron at high cutting speeds. Rough turning of rolls in white/chilled cast iron.
- Applications that require extreme wear resistance and toughness.
- CBN consists of Boron nitride with Ceramic or Titanium nitride binder.
- Resists high cutting temperatures at high cutting speeds.
- Special application area with small volume inserts. Trend is towards a higher volume of hard materials to be cut.

Polycrystalline diamond



- Turning of normal non-ferrous at low temperature and very abrasive hypereutectic non-ferrous. Used in non-metal and non-ferrous materials.
- Extremely wear resistant grades. Sensitive to chipping.
- Brazed-in corners of polycrystalline diamond (PCD tip) to an insert or thin diamond coated film on a substrate.
- Long tool life and extremely good wear resistance. Decomposes at high temperatures. Dissolves easily in iron.
- Fairly low portion of the insert program, with special limited applications.

The development of cutting tool material

The development of cutting tool material through the years can be seen in the reduced time taken to machine a component 500 mm long, with 100 mm diameter (19.685 inch long, with 3.937 inch diameter) from 1900 to today.

At the beginning of the last century, cutting tool material was only slightly harder than the material which needed to be cut. Therefore tool life was poor, and cutting speed and feed had to be kept very low.

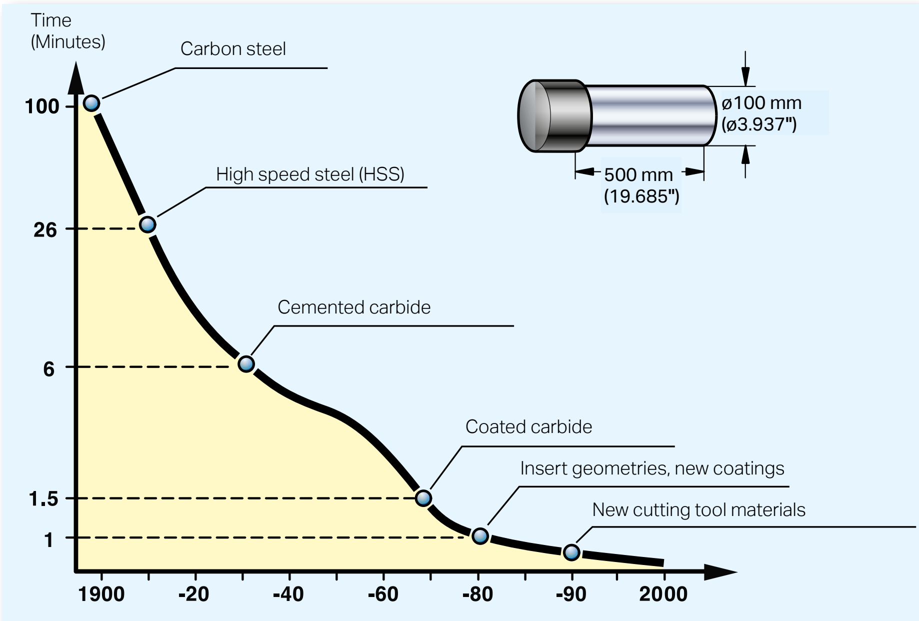
The introduction of HSS brought major improvements, which resulted in reduced cutting time.

20 years later uncoated cemented carbide brought down the required time in cut to a staggering 6 minutes.

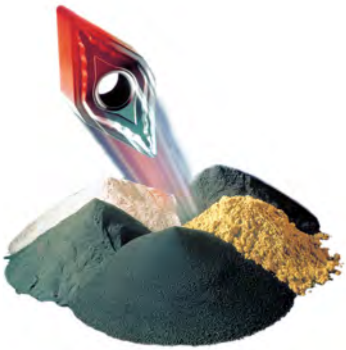
The introduction of coated carbide again lowered the cutting time to 1.5 minutes.

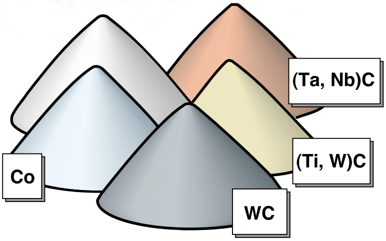
Today with improved geometries and new coating technique we have reached below 1 minute in cutting time for the 500 mm (19.685 inch) steel bar.

In addition to traditional uncoated and coated carbide, new cutting tool materials like cermet, ceramic, cubic boron nitride and diamond, have contributed to optimized and improved productivity.



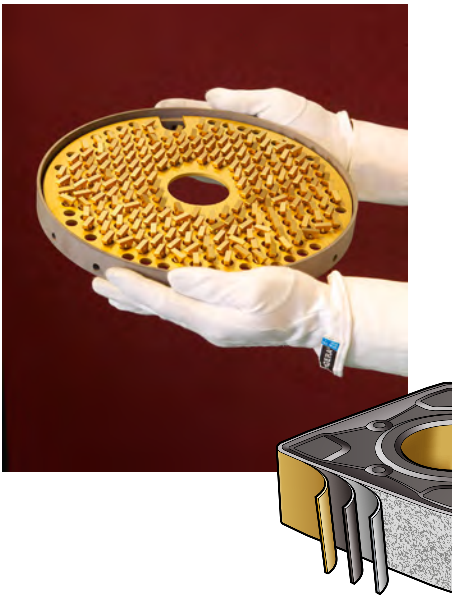
What is cemented carbide and a grade?





- Cemented carbide is a powder metallurgical material consisting of:
  - hard-particles of tungsten carbide (WC)
  - a binder metal, cobalt (Co)
  - hard-particles of Ti, Ta, Nb (titanium, tantalum, niobium-carbides).
- A grade represents the hardness or toughness of the insert, and is determined by the mixture of ingredients which make up the substrate.

Coating of cemented carbide



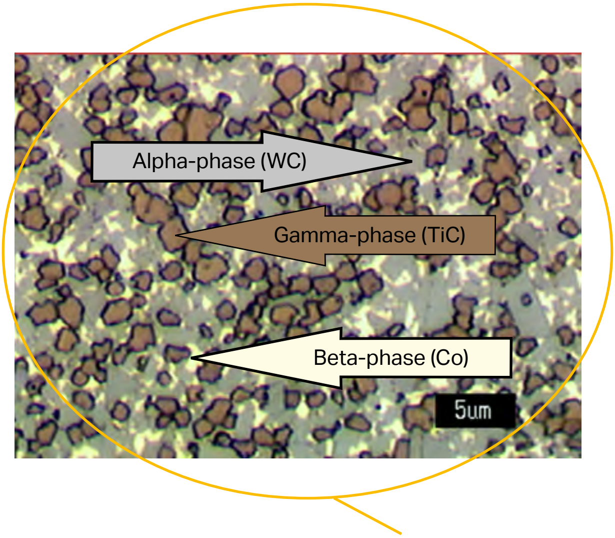
- Coating of cemented carbide was developed in the 1960s.
- A thin Titanium Nitride coating layer was added, only a few microns thick. This improved the performance of carbide overnight.
- Coatings offer improved wear resistance giving longer tool life and possibility to use higher cutting data.
- Today modern grades are coated with different carbide, nitride and oxide layers.

Microstructure of cemented carbide

Cemented carbide consists of hard particles (carbides) in a binder matrix.

The binder is more or less in all cases cobalt (Co) but could also be Nickel (Ni). The hard particles consist mainly of tungsten carbide (WC) with a possible addition of gamma phase (Ti-, Ta- Nb- carbides and nitrides).

The gamma phase has a better hot hardness and is less reactive at elevated temperatures, so is often seen in grades where the cutting temperature can get high. WC has a better abrasive wear resistance.



Hair diameter
= 50-70 µm (.0020-.0028")

Elements:

Alpha-phase
WC (tungsten carbide)

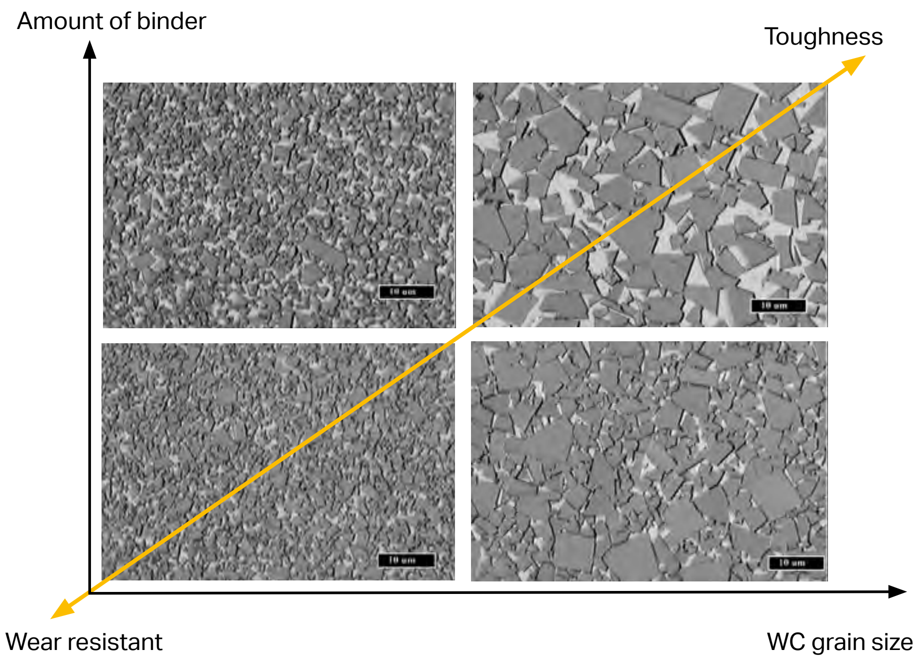
Gamma-phase
(Ti,Ta,Nb)C
(titanium, tantalum, niobium-carbides)

Beta-phase
Co (cobalt)

Fundamental characteristics

Apart from the grain size of the tungsten carbide (WC), the amount of binder phase cobalt (Co) is an important factor determining the characteristics of the carbide. The Co content in Sandvik Coromant grades is generally 4–15% of the total weight.

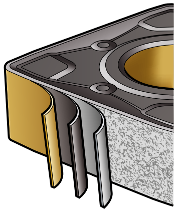
An increase in Co content and WC grain size contributes to an increase in bulk toughness, but also lowers the hardness. As a result, the substrate has less resistance to plastic deformation, which means less wear resistance/lower practical tool life.



Coating design

Many factors influence the behavior of the insert:

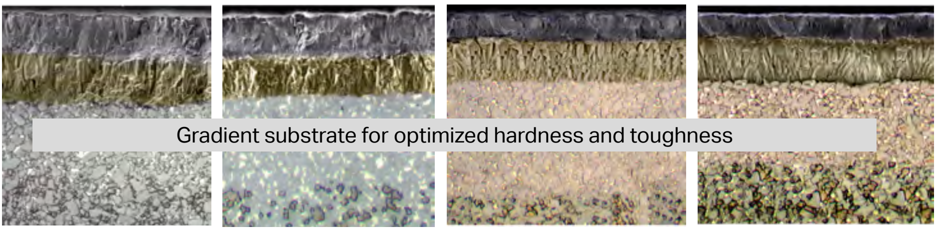
- Coating process
- Coating material
- Coating thickness
- Post treatment
- Surface morphology.



Example of modern steel turning grades

Structure and build-up of the coating layers

Wear resistance | P | Toughness



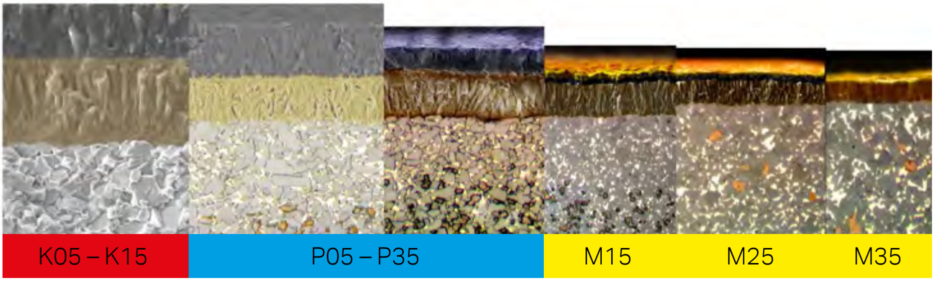
ISO P01 – P15 | ISO P05 – P30 | ISO P10 – P35 | ISO P20 – P45

Thicker coatings mean more wear resistance.

Harder substrates mean more deformation resistance.

Grade design

Coatings and substrates vary with the type of application

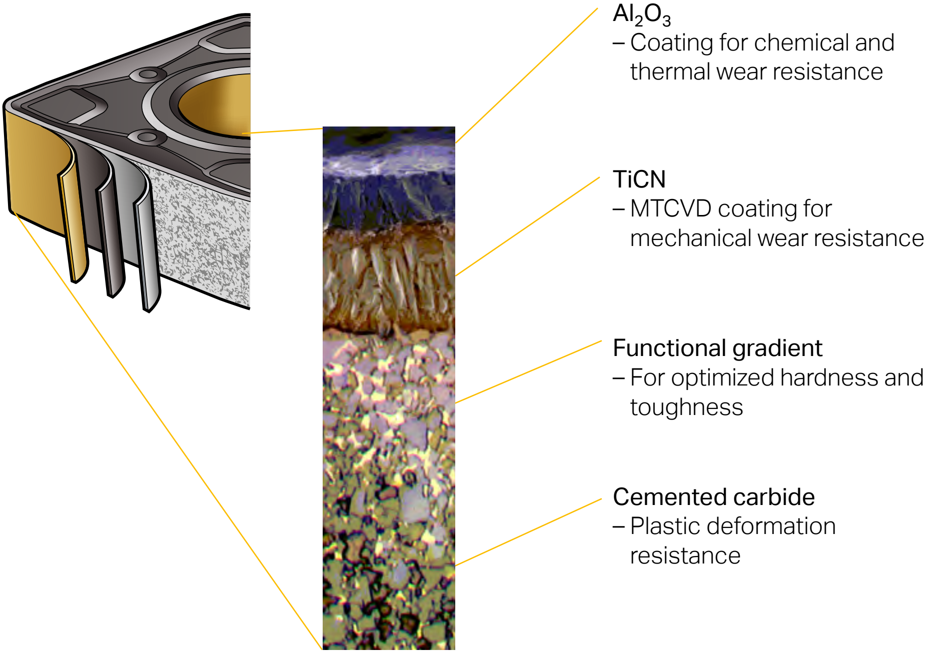


Thicker coatings mean more wear resistance.

Harder substrates mean more deformation resistance.

The coating of a modern turning grade

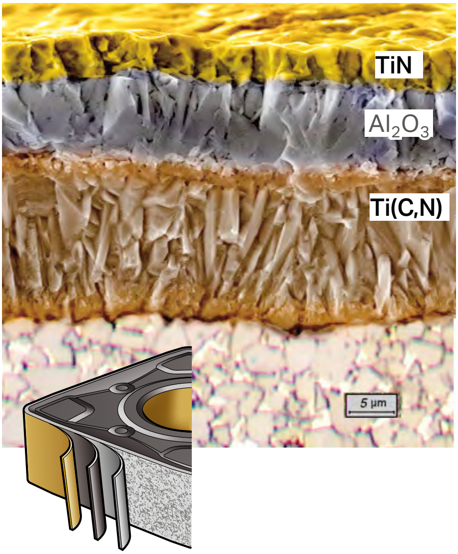
The grade plays a very important part of the performance



Properties of different coating materials

CVD coating of inserts

Chemical Vapor Deposition

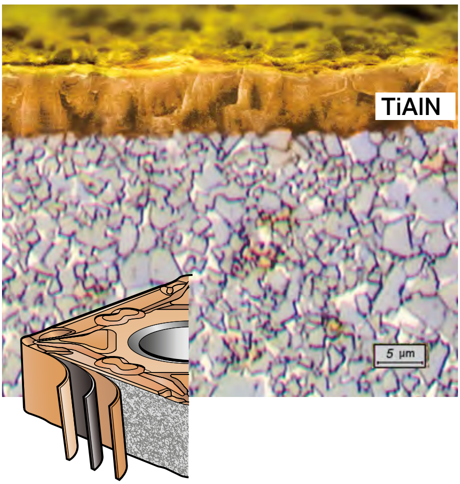


- The most common CVD layers today are TiN, Ti(C,N) and Al_2O_3.
- TiCN provides flank wear resistance.
- Al_2O_3 provides temperature protection (plastic deformation resistance).
- TiN provides easy wear detection.

TiN = Titanium nitride
Ti(C,N) = Titanium carbonitride
Al_2O_3 = Non-ferrous oxide

PVD coating of inserts

Physical Vapor Deposition



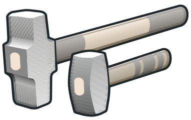
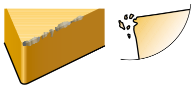
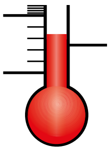
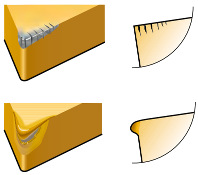
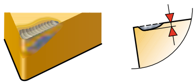
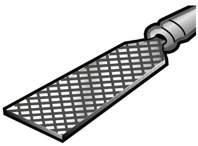
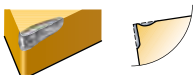
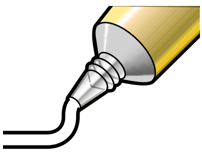
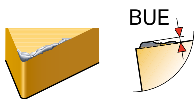
- PVD coatings are generally tougher than CVD coatings.
- PVD coatings are often used in combination with fine-grained substrates to coat "sharp" cutting edges.
- Total thickness of the PVD layers is often between 3 – 6 µm (.0001 – .0002 inch).
- The coating is applied at approx. 500° C (932° F).

TiAlN = Titanium aluminum nitride

Tool wear & maintenance

The tough environment in metal cutting

Different wear mechanisms on the inserts

Type of load	Symbol	Wear picture	Cause
Mechanical			Mechanical stress on the insert edge causes breakage.
Thermal			Temperature variations cause cracks and heat generates plastic deformation (PD) on the insert edge.
Chemical			A chemical reaction between carbide and working material causes wear.
Abrasive			In cast iron the SiC inclusions can wear on the insert edge.
Adhesive			With sticky material, built-up layers/edges are formed.

BUE = Built-Up Edge

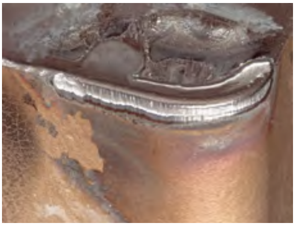
PD = Plastic Deformation

Wear pictures, cause and remedy

Some of the most common wear patterns

Flank wear (abrasive)

Flank wear is one of the most common wear types and it occurs on the flank face of the insert (tool). This is the preferred wear pattern.



Cause
During cutting, tool material is lost on the flank face due to friction against the surface of the work piece material. Wear typically begins at the edge line and gradually develops downward.

Remedy
Reducing the cutting speed and simultaneously increasing the feed will result in increased tool life with retained productivity.

Crater wear (chemical)

Cause
Crater wear occurs as a result of chip contact with the rake face of the insert (tool).

Remedy
Lowering the cutting speed and choosing an insert (tool) with the right geometry and a more wear resistant coating will increase the tool life.

Plastic deformation (thermal)

Plastic deformation is a permanent change in the shape of the cutting edge, where the edge has either suffered an inward deformation (edge impression) or a downward deformation (edge depression).

Edge depression

Edge impression

Cause
The cutting edge is subjected to high cutting forces and temperatures resulting in a stress state, exceeding the tool materials yield strength and temperature.

Remedy
Plastic deformation can be dealt with by using grades with higher hot hardness. Coatings improve the plastic deformation resistance of the insert (tool).

Flaking

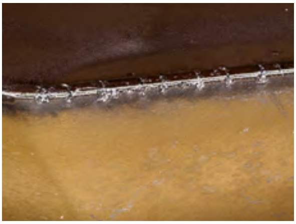
Flaking usually occurs when machining in materials with smearing properties.

Cause
An adhesive load can develop, where the cutting edge is subjected to tensile stresses. This may lead to the detachment of the coating, exposing sublayers or substrate.

Remedy
Increasing the cutting speed as well as selecting an insert with a thinner coating will reduce the flaking on the tool.

Cracks (thermal)

Cracks are narrow openings in which new boundary surfaces have been formed through rupture. Some cracks are confined to the coating, while others extend down into the substrate. Comb cracks are roughly perpendicular to the edge line and most often thermal cracks.



Cause
Comb cracks form as a result of rapid fluctuations in temperature.

Remedy
To prevent this, a tougher insert grade can be used and the coolant should be applied in large amounts or not at all.

Chipping (mechanical)

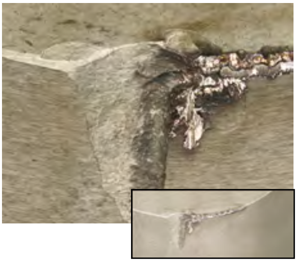
Chipping consists of minor damage to the edge line. The difference between chipping and fracture is that with chipping the insert can still be used.

Cause
There are many combinations of wear mechanisms that can cause chipping. However, the most common are thermo-mechanical and adhesive.

Remedy
Different preventative measures can be taken to minimize chipping, depending on which wear mechanism/mechanisms that caused it.

Notch wear

Notch wear is characterised by excessive localised damage at maximum cutting depth but can also occur on secondary edge.



Cause
Depending upon if the chemical wear dominates the notch wear, which proceeds more regularly, as in the picture, compared to irregular growth of adhesive or thermal wear. In the latter case work hardening and burr formation are important factors for notch wear.

Remedy
For work-hardening materials, select a smaller entering angle and/or vary the depth of cut.

Fracture

Fracture is defined as the breakout of a large part of the cutting edge, where the insert can no longer be applied.

Cause
The cutting edge has been exposed to a greater load than it can resist. This could be the result of allowing the wear to progress too far leading to increased cutting forces. It can also be caused prematurely due to the wrong cutting data or stability issues in the setup.

Remedy
Identify and prevent the original wear type, selecting proper cutting data and checking stability of setup.

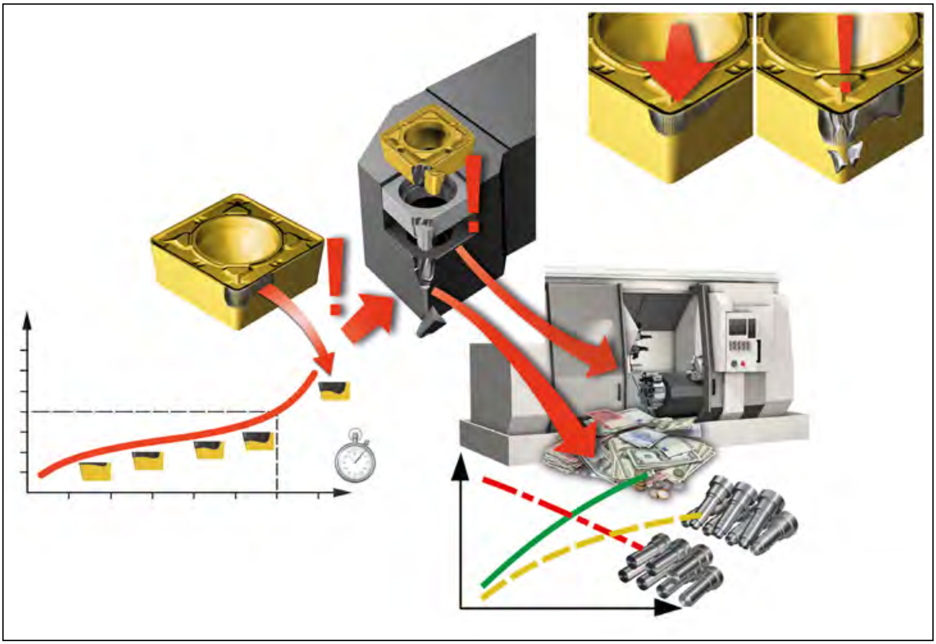
Built up edge (adhesive)

Built up edge (BUE) is an accumulation of material against the rake face.

Cause
Built up material can form on the top of the cutting edge, which separates the cutting edge from the material. Resulting in increased cutting forces, leading to failure or releasing and taking away parts of the coating and even substrate layers,

Remedy
Increasing the cutting speed can prevent the formation of BUE. In softer, stickier materials a sharper edge will help.

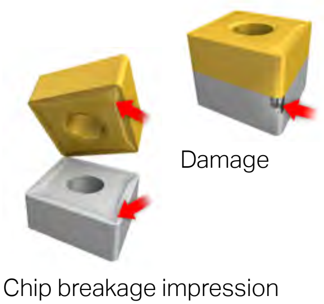
Consequences of poor tool maintenance



- Damaged inserts
- Damaged shims
- Damaged tool holders
- Damaged components
- Damaged machine

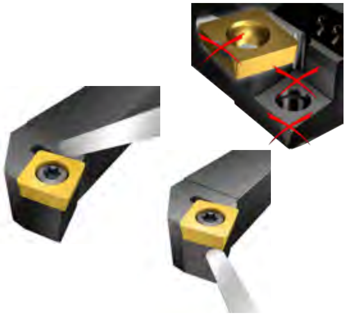
Result:
- Reduced production
- Higher production costs

Inspection of tool



Visually inspect shims & shim seats

- Check shim damage.
- Clean insert seat and damaged location and support for cutting edge.
- If necessary index or replace shim.
- Ensure correct insert location against support points.
- It is important to ensure that shim corners have not been knocked off during machining or handling.



Inspect pockets

- Pockets damaged or plastic deformation.
- Oversized pockets due to wear. The insert does not sit properly in the pocket sides. Use a 0.02 mm (.0008 inch) shim to check the gap.
- Small gaps in the corners, between the shim and the bottom of the pocket.

The importance of using the correct wrench

Why use the proper wrenches?

- Extends life of screw and wrench.
- Reduces risk of stripping screw.

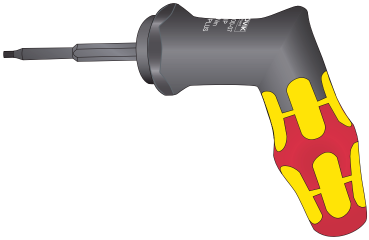
What is the proper way to tighten an insert screw?

- Important to use the proper wrench.
- Always use correct torque. Values are marked on tool and shown in product catalog.
- Common sense!

Torx Plus® wrenches

Torx Plus from Sandvik Coromant

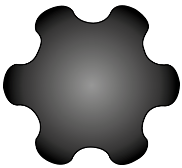
Nm (lbs-in)



Torx Plus® vs. Torx

Cross section

Torx Plus®



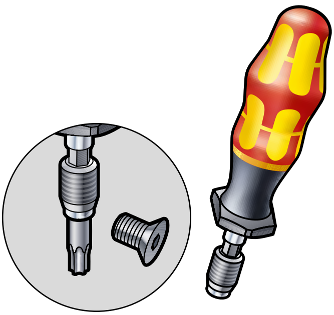
Torx

Torx Plus®

Standard Torx screw

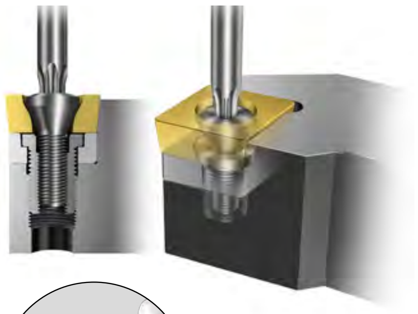
Torx Plus is a registered trademark of Camcar-Textron (USA)

Torx Plus® wrenches with adjustable torque



- On parting and grooving tools an adjustable torque wrench is required, as the torque is not related to screw size.
- It should of course be used on all products with a clamp screw.

Insert screws / clamping screws



- Screw threads, heads and Torx sockets should be in good condition.
- Use correct keys.
- Ensure correct screw-tightening torque.
- Apply sufficient screw lubrication to prevent seizure. Lubricant should be applied to the screw thread as well as the screw-head face.
- Replace worn or exhausted screws.

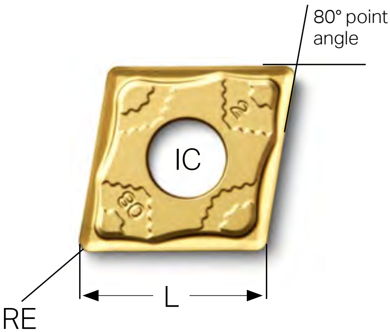
Important!

Use Anti-seize for screw heads and threads

Tool maintenance

Contact faces

- Always check supporting and contact faces of tool holders, milling cutters and drills, making sure there is no damage or dirt.
- In boring operations it is especially important to have the best possible clamping. If the bar is not supported to the end of the holder, overhang will be increased and create vibration.



L = cutting edge length (insert size)

RE = nose radius

Production security

- It is important to select the correct insert size, insert shape and geometry and insert nose radius to achieve good chip flow.
  - Select largest possible point angle on the insert for strength and economy.
  - Select largest possible nose radius for insert strength.
  - Select a smaller nose radius if there is a tendency for vibration.

Stability

- Stability is the key factor for successful metal cutting, affecting machining costs and productivity.
- Make sure that any unnecessary play, overhang, weakness, etc., has been eliminated and that correct types and sizes of tools are employed for the job.

Insert handling

Inserts are placed in segregated packages in order to prevent insert to insert contact, as this may damage the carbide with micro fracturing and/or chipping. Which may reduce insert performance and life. It's recommended that inserts remain in their original packaging until they are applied in the machining process.

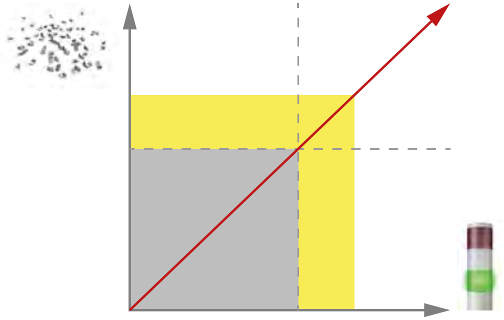
Summary of maintenance checklist

- ☐ Check tool wear and shims for damage.
- ☐ Make sure insert seat is clean.
- ☐ Make sure of correct insert location.
- ☐ Make sure correct keys and drivers are used.
- ☐ Insert screws should be correctly tightened.
- ☐ Lubricate screws before tool assembly.
- ☐ Make sure contact faces are clean and undamaged on tools, holding tools and machine spindles.
- ☐ Make sure boring bars are clamped well and that holders are undamaged at the end.
- ☐ A well organized, maintained and documented tool inventory is a production cost saver.
- ☐ Stability is always a critical factor in any metal cutting operation.

Machining economy

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

Doing more machining in the same production time



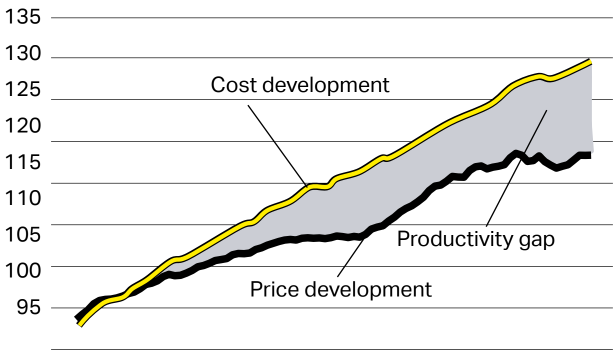
Productivity definition

The value of output produced divided by the value of input or resources.

= Output / Input

Attack the productivity gap

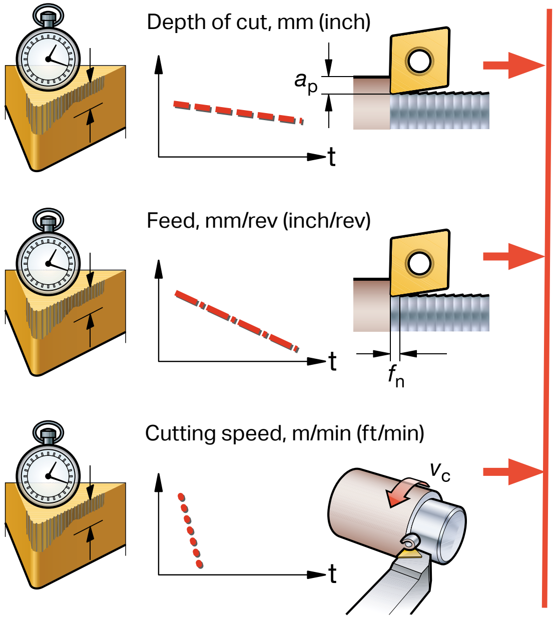
In all industrial operations, the cost of running the operation, e.g. for labor, raw material, equipment, etc., is increasing at a faster rate than the price of the goods that are sold. In order to bridge that gap, one needs to continuously increase efficiency, resulting in higher productivity. Bridging this gap is the only way to stay competitive and ultimately to stay in business.

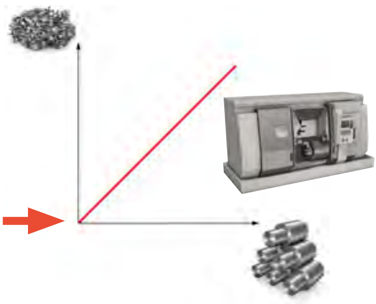


Source: Mechanical Industry in OECD.

Maximizing productivity

The three main machining parameters, cutting speed, feed, and depth of cut, have an effect on tool life. The depth of cut has the smallest effect followed by the feed rate. Cutting speed has the largest effect by far on insert tool life.

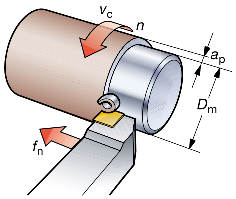




Optimizing the depth of cut, feed rate and then the cutting speed is the best way to maximize the productivity.

Productivity "Q" is measured as the amount of material removed in a fixed time period, cm³/min (inch³/min).

Turning



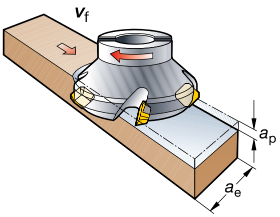
Metric

$$Q = v_c \times a_p \times f_n$$

Inch

$$Q = v_c \times a_p \times f_n \times 12$$

Milling



Metric

$$Q = \frac{a_p \times a_e \times v_f}{1000}$$

Inch

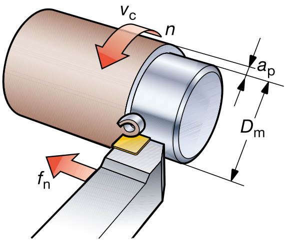
$$Q = a_p \times a_e \times v_f$$

Maximizing productivity – examples

Metal removal rates for a fixed depth of cut of 3.0 mm (.118 inch) using:

P Low alloy steel, MC P2

Hardness, HB 180



Insert: CNMG 432-PM 4225 (CNMG 120408-PM 4225)

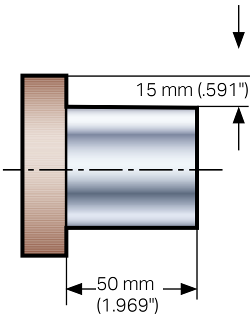
a_p, mm (inch)	3.0 (.118)	3.0 (.118)	3.0 (.118)
f_n, mm/r (inch/r)	0.15 (.006)	0.3 (.012)	**0.5 (.020)**
v_c, m/min (ft/min)	425 (1394)	345 (1132)	**275 (902)**
Q, cm3/min (inch3/min)	191 (12)	310 (19)	412* (25)*

* Slowest cutting speed with the highest feed = highest productivity

Using a trigon W-style insert, versus a C-style double-sided or single-sided insert

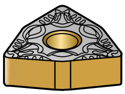
P Low alloy steel, MC P2

Hardness, HB 180



Trigon shape

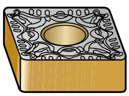
Insert: double-sided for medium machining.



No of passes / cutting depth, a_p	3/4 mm (.118 /.157 inch) 1/3 mm (.039/.118 inch)
Machining time, T_c	22 seconds

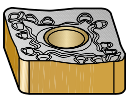
Rhombic shape

Insert: double sided for medium machining.



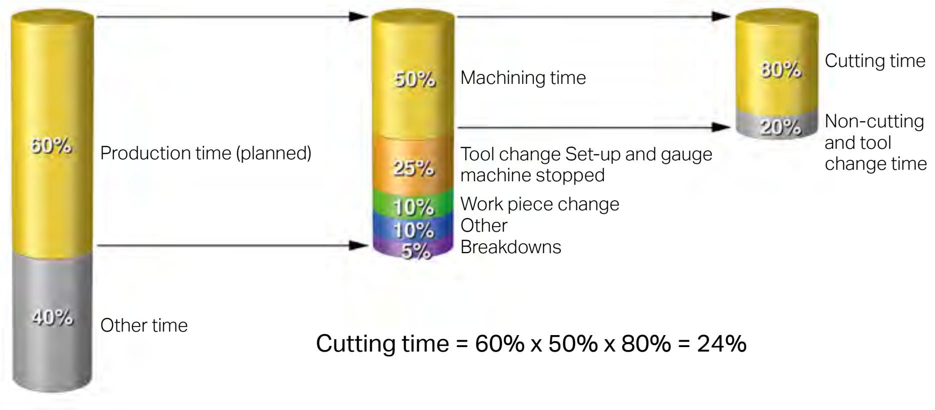
No of passes / cutting depth, a_p	3/5 mm (.118/.197 inch)
Machining time, T_c	16 seconds

Insert: Single sided for rough machining.



No of passes / cutting depth, a_p	2/7.5 mm (.079/.295 inch)
Machining time, T_c	8 seconds

Value adding time



Machining economy

- Variable costs
  Costs incurred only during production:
  - cutting tools, consumables (3%)
  - workpiece materials (17%).

- Fixed costs
  Costs which exist even when not in production:
  - machine and tool holders (27%)
  - labor (31%)
  - buildings, administration, etc. (22%).

Machine tool utilization

Cost, tool life or productivity

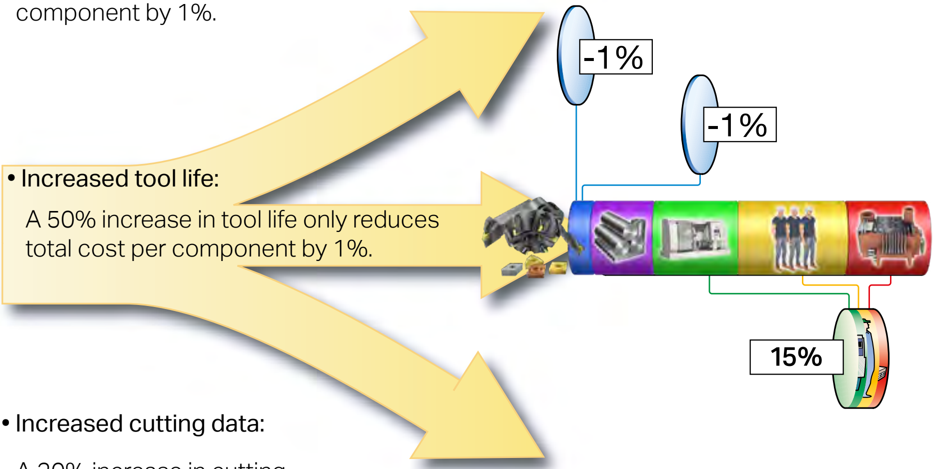
The cost of the tooling, an easily measured value, is always under price or discount pressure, but even when the price is reduced by 30% it only influences the component cost by 1%.

We have a similar result of a 1% cost saving when tool life is increased by 50%.

Increasing the cutting data by only 20% will dramatically reduce component costs and lead to a 15% component saving.

- Decreased cost:

  A 30% decrease in price only reduces total cost per component by 1%.



- Increased cutting data:

  A 20% increase in cutting data reduces total cost per component by more than 15%.

Machine tool utilization

Cost, tool life or productivity

Example:
Shop spends $10,000 to make 1000 parts.

Machine cost is $10.00 per part.

Variable	Today	Lower price	Tool life	Increase cutting data
– Tooling	$ .30	$ .21	$ .20	$ .45
– Material	$ 1.70	$ 1.70	$ 1.70	$ 1.70
Fixed				
– Machinery	$ 2.70	$ 2.70	$ 2.70	$ 2.16
– Labor	$ 3.10	$ 3.10	$ 3.10	$ 2.48
– Building	$ 2.20	$ 2.20	$ 2.20	$ 1.76
Cost per part	$ 10.00	$ 9.91	$ 9.90	$ 8.55
Savings		1%	1%	15%

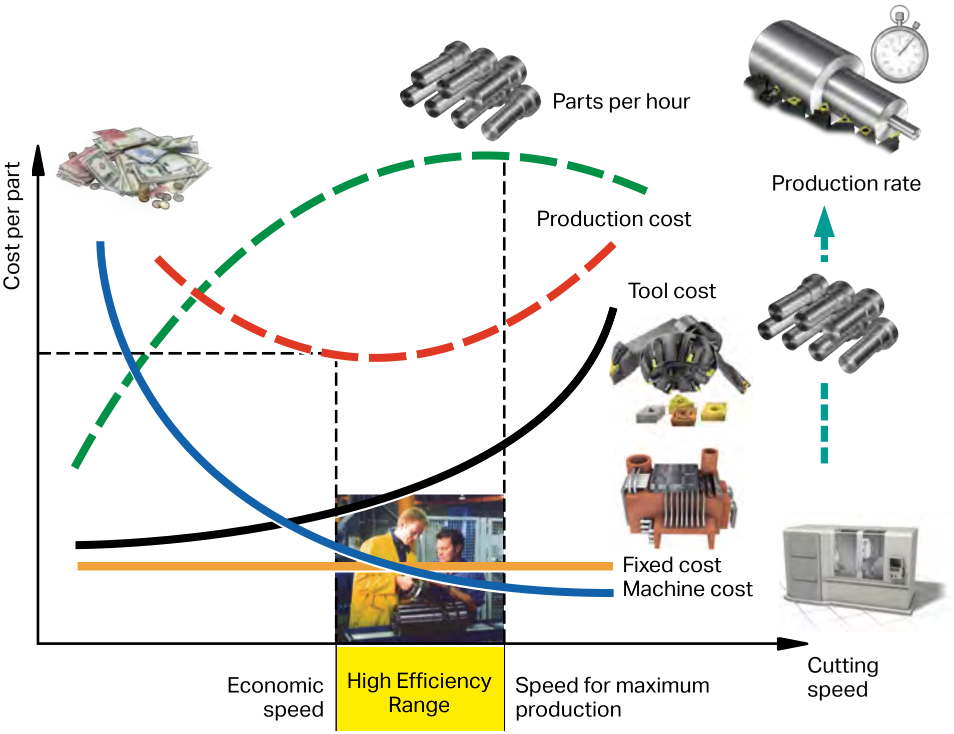
Machining ecomomy

Cutting data and cost

- Cutting speed has no effect on fixed costs.
- As cutting speed increases more parts are produced per hour and therefore cost per part is reduced.
- As cutting speed increases more tools are used and therefore cost per part increases.
  If we add all costs together we will get the curve of total **Production cost**.

1. As speed increases the **Parts per hour** increase until we reach a point where we are spending a disproportionate amount of time changing tools and production rate will start to decrease.
2. The lowest point on the **Production cost curve** corresponds to the economic cutting speed.
3. The highest point on the **Production cost curve** corresponds to the maximum cutting speed.

The speed between these two points is the **High Efficiency Range**, which is where we should be trying to operate.



Base for cutting data recommendations

Compensation of cutting speed for increased tool life or higher metal removal

Tool life

- All recommended cutting data is based on 15 minutes of tool life.
- Looking at the chart below 15 min tool life = a factor of 1.0.
- Multiple the factor for desired minutes by the recommended cutting speed.

Increase tool life (example)

- Our Recommended cutting data is 225 m/min (738 ft/min).
- To increase tool life by 30%, we look at the factor for 20 minutes of tool life = 0.93.
- Multiple the factor for desired minutes by the recommended cutting speed.
- 225 m/min x 0.93 = 209 m/min (738 ft/min x 0.93 = 686 ft/min).

Tool life (min)	10	15	20	25	30	45	60
Correction factor	1.11	1.0	0.93	0.88	0.84	0.75	0.70

Higher metal removal rate

- Recommended cutting data is based on 15 minutes of tool life.
- To obtain higher metal removal rates, we would move in the opposite direction on the chart. Decreasing the minutes of tool life to gain higher metal removal.
- Multiple the factor for desired minutes by the recommended cutting speed.

Higher metal removal rate (example)

- The Recommended cutting data is 225 m/min (738 ft/min).
- To increase metal removal by 10%, we look at the factor for 10 minutes = 1.11.
- Multiple the factor for desired minutes by the recommended cutting speed.
- 225 m/min x 1.11 = 250 m/min (738 ft/min x 1.11 = 819 ft/min).

Compensation of cutting speed for differences in material hardness

Hardness

- Cutting speed recommendations are based on the material reference and their respective hardness.
- Metal material hardness is measured in Hardness Brinell (HB) or Hardness Rockwell "C" scale (HRC) example: ISO/ ANSI **P** = 180 HB, ISO/ANSI **H** = 60 HRC.
- The hardness (HB) column is the base hardness for each material group and cutting speeds are recommended for this base hardness (note: your material could be harder/softer).
- Each ISO/ANSI material group is associated with a multiplying factor for reduced/ increased hardness of material (example ISO/ANSI **P** = 180 HB and has a factor of 1.0).
- Use the chart below for correction factors and multiply by the recommended cutting speed for the chosen insert grade.

			Reduced hardness ←				Increased hardness →				
ISO/ ANSI	**MC(1)**	**HB(2)**	**-60**	**-40**	**-20**	**0**	**+20**	**+40**	**+60**	**+80**	**+100**
P	P2	HB 180	1.44	1.25	1.11	1.0	0.91	0.84	0.77	0.72	0.67
M	M1	HB 180	1.42	1.24	1.11	1.0	0.91	0.84	0.78	0.73	0.68
K	K2	HB 220	1.21	1.13	1.06	1.0	0.95	0.90	0.86	0.82	0.79
	K3	HB 250	1.33	1.21	1.09	1.0	0.91	0.84	0.75	0.70	0.65
N	N1	HB 75			1.05	1.0	0.95				
S	S2	HB 350			1.12	1.0	0.89				
H	H1	HRC(3) 60			1.07	1.0	0.97				

1) MC = material classification code
2) HB = Hardness Brinell
3) HRC = Hardness Rockwell

Example of Conversion table for hardness scale

Material specifications maybe given in different forms, example: HB, HRC, Tensile Strength or Specific Cutting forces.

Tensile strength		Vickers	Brinell	Rockwell	
N/mm2	lbs/inch2)	HV	HB	HRC	HRB
255	36,975	80	76.0	–	–
270	39,150	85	80.7	–	41.0
285	41,325	90	85.5	–	48.0
305	44,225	95	90.2	–	52.0
320	46,400	100	95.0	–	56.2
350	50,750	110	105	–	62.3
385	55,825	120	114	–	66.7
415	60,175	130	124	–	71.2
450	65,250	140	133	–	75.0
480	69,600	150	143	–	78.7
510	73,950	160	152	–	81.7
545	79,025	170	162	–	85.0
575	83,375	180	171	–	87.5
610	88,450	190	181	–	89.5
640	92,800	200	190	–	91.5
660	95,700	205	195	–	92.5
675	97,875	210	199	–	93.5
690	100,050	215	204	–	94.0
705	102,225	220	209	–	95.0
720	104,400	225	214	–	96.0
740	107,300	230	219	–	96.7
770	111,650	240	228	20.3	98.1
800	116,000	250	238	22.2	99.5
820	118,900	255	242	23.1	–
835	121,075	260	247	24.0	(101)
850	123,250	265	252	24.8	–
865	125,425	270	257	25.6	(102)
900	130,500	280	266	27.1	–
930	134,850	290	276	28.5	(105)
950	137,750	295	280	29.2	–
965	139,925	300	285	29.8	–
995	144,275	310	295	31.0	–

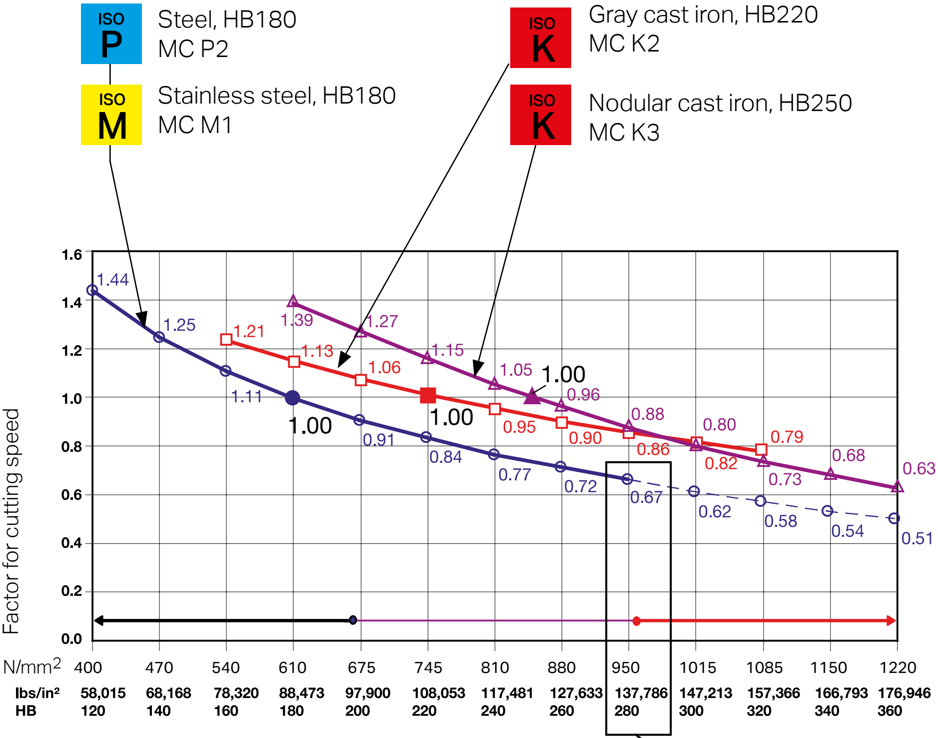
Customer workpiece material (match to information on chart)

Tensile strength = 950 N/mm2 (137,750 lbs/inch2)

HB = 280, HRC = 29.2

Example of conversion table, finding the factor for hardness

Diagram form for P, M and K



Specific cutting forces N/mm2 (lbs/in2)
Hardness Brinell (HB)

Customer workpiece material
4140 Steel
Tensile strength = 950 N/mm2 (137,786 lbs/inch2)
HB = 280, HRC = 29.2

Calculating hardness factor = 0.67

Compensation of cutting speed for differences in material hardness

Example:

- Recommended cutting data is 415 m/min (1360 ft/min) for **P** Steel material 180 HB.
- Customer workpiece material = 280 HB **P** Steel material.
- Calculating hardness factor, Customer material =280 HB – Material reference 180 HB = +100 HB in increased hardness (factor = 0.67).
- Use the factor to recalculate cutting speed for the material hardness 415 m/min x 0.67 = 278 m/min (1360 ft/min x 0.67 = 911 ft/min).

ISO/ ANSI	MC(1)	HB(2)	Reduced hardness ← -60	-40	-20	0	Increased hardness → +20	+40	+60	+80	+100
P	P2	HB 180	1.44	1.25	1.11	1.0	0.91	0.84	0.77	0.72	0.67
M	M1	HB 180	1.42	1.24	1.11	1.0	0.91	0.84	0.78	0.73	0.68
K	K2	HB 220	1.21	1.13	1.06	1.0	0.95	0.90	0.86	0.82	0.79
	K3	HB 250	1.33	1.21	1.09	1.0	0.91	0.84	0.75	0.70	0.65
N	N1	HB 75			1.05	1.0	0.95				
S	S2	HB 350			1.12	1.0	0.89				
H	H1	HRC(3) 60			1.07	1.0	0.97				

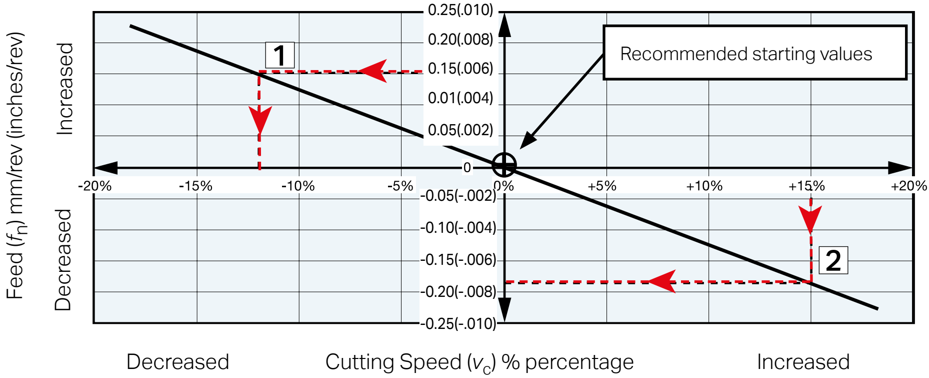
1) MC = material classification code
2) HB = Hardness Brinell
3) HRC = Hardness Rockwell

Compensation of cutting speed and feed data for Turning

How to use the diagram

This diagram shows a simple method of adjusting the starting values for cutting speed and feed recommendations.

Recommended cutting data for inserts are based on 15 minutes of tool life (in cut time), as well as maintaining chip formation and this will remain the same with the values taken from this diagram.



Example 1: Productivity increase

- Increasing the feed rate by 0.15 (.006") to give a new starting value of 0.45 mm/r (.018 in/r).
- Calculate the new cutting speed of -12% from the diagram by intersecting feed with Start value line and cutting speed axis.
- New cutting data = 0.45 mm/r (.018 in/r) and 415 x .88 = 365 m/min (1360 x .88 = 1197 ft/min) Metal removal +30%.

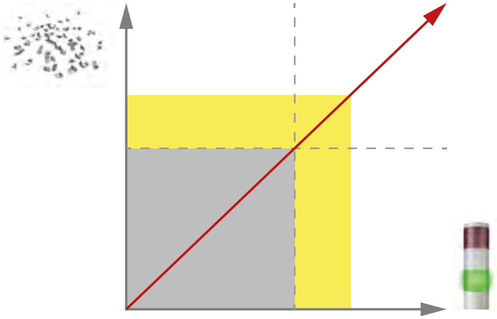
Example 2: Better Surface finish

- Increasing the cutting speed by 15% to give a new starting value of 477 m/min (1564 ft/min).
- Calculate the new cutting feed of -0.175 (-.0075") from the diagram by intersecting speed with Start value line and feed axis.
- New cutting data = 477 m/min (1564 ft/min) and 0.3 – 0.175 = 0.125 mm/r (.012"- .0075" = .0045 in/r) improved Surface finish.

⊕ Recommended starting values

CNMG 12 04 08-PM
(CNMG 432 – PM)
P15 grade
3 mm (.118") - Depth of cut
0.3 mm/r (.012 in/r) – Feed Rate
415 m/min (1360 ft/min) – Cutting speed

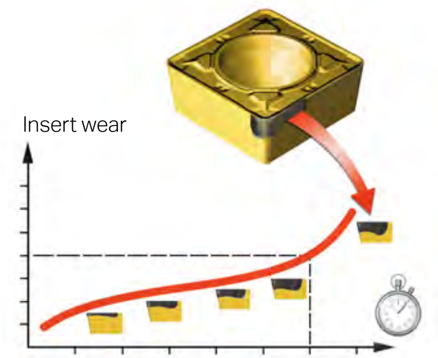
How can you improve productivity?



Things to consider

- Identify material hardness HB, Specific cutting forces or Tensile strength N/mm2 (lbs/inch2).
- Choose the correct geometry.
- Choose the correct grade.
- Use given cutting data values, compensate for material hardness factor.
- Create a stable environment for component and tools.

Machining tips for improved tool life



- Identify material hardness HB, Specific cutting forces or Tensile strength N/mm2 (lbs/inch2).
- Use given cutting data values, compensate for material hardness factor.
- Create a stable environment for component and tools.
- Choose the right combination of nose radius and geometry.
- Use climb milling over conventional, when ever possible.
- Make use of all available insert corners
- Consider chamfering operations with worn inserts.

Good stability = Successful metal cutting

ISO 13399 The industry standard

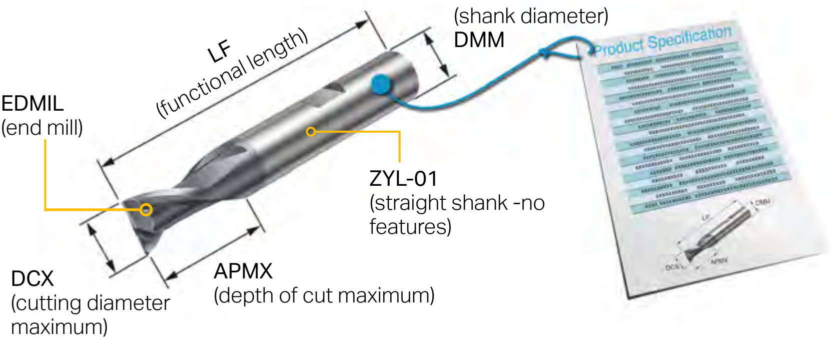
ISO 13399 - The industry standard

Variations in terminology among cutting tool suppliers make collection and transfer of information complex. At the same time, more and more advanced functionality in modern manufacturing systems rely on access to relevant and exact information.

A common language is valuable from a system to system point of view, but will also make life easier for users. ISO 13399 is the international standard simplifying exchange of data for cutting tools and is a globally recognized way of describing cutting tool data.

ISO 13399 - What it means for the industry

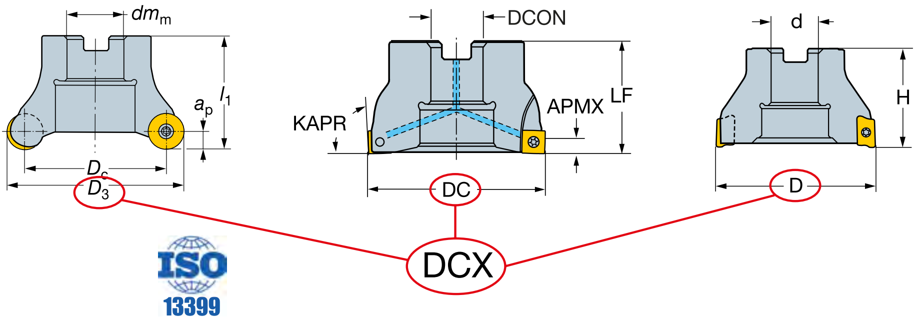
The international standard defines attributes of the tool, for example functional length, cutting diameter, maximum depth of cut in a standard way. Each tool is defined by the standardized parameters.



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

ISO 13399 - What it means for the industry

When the industry share the same parameters and definitions, communicating tool information between software systems becomes very straight forward. In the picture you see that three different suppliers call a diameter D3, DC and D respectively. It creates a lot of confusion for programmers. In the ISO 13399 standard, the diameter will always be named DCX.



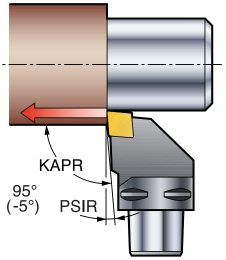
A full list of parameters is available on www.sandvik.coromant.com

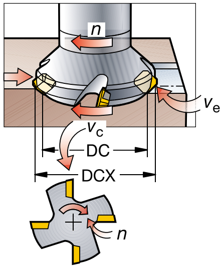
Formulas & definitions

E-learning

Glossary of terms

$$v_c = \frac{\pi \times D_m \times n}{1000}$$





a_e **(Working engagement)** working engagement of the cutting tool with the workpiece, measured in a direction parallel to the plane Pfe (Primary motion/Resultant cutting direction) and perpendicular to the direction of feed motion. Measured in millimeters (mm) or inches.

a_p **(Cutting depth)** cutting width perpendicular to direction of feed motion. Note: When drilling, radial cutting depth is denoted with a_p, the same symbol as for axial cutting depth/cutting width when milling. Measured in millimeters (mm) or inches.

DC (Cutting diameter) diameter of a circle created by a cutting reference point revolving around the tool axis of a rotating tool item. Note: The diameter refers to the machined peripheral surface. Measured in millimeters (mm) or inches.

D_{cap} **(Cutting diameter at depth of cut)** diameter at the distance a_p from the plane Pfe through point PK, measured in base plane 1 (Bp1). Measured in millimeters (mm) or inches.

D_m **(Machined diameter)** machined diameter of the workpiece. Measured in millimeters (mm) or inches.

F_f **(Feed force)** component of the total force obtained by perpendicular projection on the direction of the feed motion (i.e. in direction of vector v_f). Feed force for a given engagement and is measured in newton (N) and pound-force (lbf).

f_n **(Feed per revolution)** the transportation of the tool in the direction of feed motion during one revolution of rotation. Regardless of the number of effective cutting edges on the tool. In the case of turning, the distance is measured as the workpiece makes one complete revolution. Measured in mm/revolution or inches/revolution.

f_z **(Feed per tooth)** the transportation of an effective cutting edge (Z_c) in the direction of feed motion for rotation center of the tool which moves through the workpiece as the tool makes one complete revolution. In the case of turning, the distance is measured as the workpiece makes one complete revolution. Measured in mm/tooth or inches/tooth.

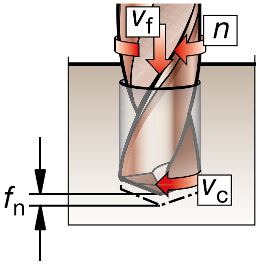
h_{ex} **(Maximum chip thickness)** is the maximum thickness of the non-deformed chip at the right angles of the cutting edge, and it is influenced by the radial engagement, edge preparation of the insert and feed per tooth. Keep in mind, however, that different radial widths of cut and different entering (lead) angles require feed rate adjustments to maintain proper chip thickness. Measured in millimeters (mm) or inches.

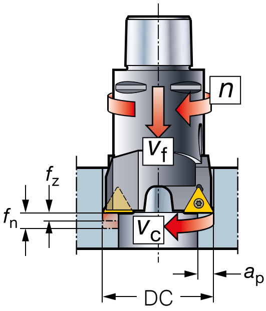
h_m **(Average chip thickness)** is the average thickness of the non-deformed chip at the right angles of the cutting edge, and it is influenced by the radial engagement, edge preparation of the insert and feed per tooth. Keep in mind, however, that different radial widths of cut and different entering (lead) angles require feed rate adjustments to maintain proper chip thickness. Measured in millimeters (mm) or inches.

KAPR (Entering angle) Angle between the cutting edge plane and the tool feed plane measured in a plane parallel the xy – plane.

k_c **(Specific cutting force)** cutting force/area for a given chip thickness in tangential direction. (Specific cutting force coefficient for material and tool combination) and is measured in newton/square millimeters (N/mm^2) and pounds/square inch (lbs/in^2).

k_{c1} **(Specific cutting force coefficient)** cutting force/area for a chip thickness of 1 mm (.0394") in tangential direction. (Material constant: specific cutting force coefficient. Traditionally named k_c 1.1) and is measured in newton/square millimeters (N/mm^2) and pounds/square inch (lbs/in^2).





$$P_c = \frac{v_c \times DC \times f_n \times k_c}{240 \times 10^3}$$

l_m **(Machined length)** length of cutting engagement over all passes. Measured in millimeters (mm) or inches.

M_c **(Rise in specific cutting force)** rise in specific force as a function of reduced chip thickness. Can be found in the work material property from cutting data tables and is measured as a ratio. Is also closely associated with specific cutting force coefficient (k_{c1}).

n **(Spindle speed)** frequency of the spindle rotation. Measured in revolutions/minute (rpm).

P_c **(Cutting power)** cutting power generated by the removal of chips. Measured in kilowatts (kW) and/or horsepower (Hp)

PSIR (Lead angle) Angle between the cutting edge plane and a plane perpendicular to the tool feed plane measured in a plane parallel the xz – plane.

Q (Material removal rate) defined as the volume of material removed divided by the machining time. Another way to define Q is to imagine an "instantaneous" material removal rate as the rate at which the cross-section area of material being removed moves through the work piece. It is measured in cubic centimeters/minute (cm^3/min) and cubic inches/minute (in^3/min).

T_c **(Cutting time total)** period of time for cutting engagement over all passes. Measured in minutes.

v_c **(Cutting speed)** the instantaneous velocity of the cutting motion of a selected point on the cutting edge relative to the workpiece. Measured in surface meter/minute or feet/minute.

v_f **(Table feed / Penetration rate)** the distance, in millimeters or inches, that a cutting tool moves through the workpiece in one minute. Measured in mm/minute or inches/minute.

$\gamma 0$ **(effective rake angle)** The specific force gets reduced by one percent for each degree of rake angle. Measured in degrees.

Z_c **(effective cutting edge count)** number of cutting edges that are effective around the tool item.

Z_n **(mounted insert count)** number of cutting edges of the tool item axis.

Formulas and definitions for turning - METRIC

Cutting speed, m/min

$$v_c = \frac{\pi \times D_m \times n}{1000}$$

Spindle speed, rpm

$$n = \frac{v_c \times 1000}{\pi \times D_m}$$

Machining time, min

$$T_c = \frac{l_m}{f_n \times n}$$

Metal removal rate, cm³/min

$$Q = v_c \times a_p \times f_n$$

Specific cutting forces

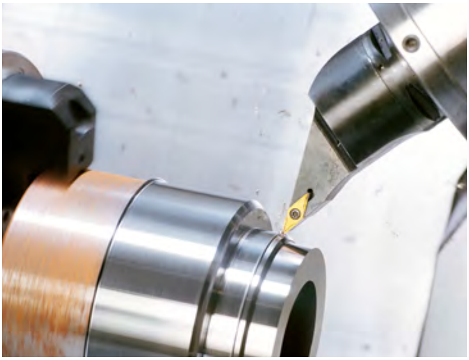
$$k_c = k_{c1} \times \left(\frac{1}{h_m}\right)^{m_c} \times \left(1 - \frac{\gamma 0}{100}\right)$$

Average chip thickness

$$h_m = f_n \times \sin \text{KAPR}$$

Net power, kW

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



Symbol	Designation/ definition	Unit
D_m	Machined diameter	mm
f_n	Feed per revolution	mm/r
a_p	Cutting depth	mm
v_c	Cutting speed	m/min
n	Spindle speed	rpm
P_c	Net power	kW
Q	Metal removal rate	cm³/min
h_m	Average chip thickness	mm
h_{ex}	Maximum chip thickness	mm
T_c	Period of engagement	min
l_m	Machined length	mm
k_c	Specific cutting force	N/mm²
KAPR	Entering angle	degree
$\gamma 0$	Effective rake angle	degree

Formulas and definitions for turning - INCH

Cutting speed, ft/min

$$v_c = \frac{\pi \times D_m \times n}{12}$$

Spindle speed, rpm

$$n = \frac{v_c \times 12}{\pi \times D_m}$$

Machining time, min

$$T_c = \frac{l_m}{f_n \times n}$$

Metal removal rate, inch³/min

$$Q = v_c \times a_p \times f_n \times 12$$

Specific cutting forces

$$k_c = k_{c1} \times \left(\frac{0.0394}{h_m}\right)^{m_c} \times \left(1 - \frac{\gamma 0}{100}\right)$$

Average chip thickness

$$h_m = f_n \times \sin \text{KAPR}$$

Net power, HP

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

Symbol	Designation/definition	Unit
D_m	Machined diameter	inch
f_n	Feed per revolution	inch/r
a_p	Cutting depth	inch
v_c	Cutting speed	ft/min
n	Spindle speed	rpm
P_c	Net power	HP
Q	Metal removal rate	inch³/min
h_m	Average chip thickness	inch
h_{ex}	Maximum chip thickness	inch
T_c	Period of engagement	min
l_m	Machined length	mm
k_c	Specific cutting force	lbs/inch²
PSIR	Lead angle	degree
γ0	Effective rake angle	degree

Formulas and definitions for milling - METRIC

Table feed, mm/min

$$v_f = f_z \times n \times z_c$$

Cutting speed, m/min

$$v_c = \frac{\pi \times D_{cap} \times n}{1000}$$

Spindle speed, r/min

$$n = \frac{v_c \times 1000}{\pi \times D_{cap}}$$

Feed per tooth, mm

$$f_z = \frac{v_f}{n \times z_c}$$

Feed per revolution, mm/rev

$$f_n = \frac{v_f}{n}$$

Metal removal rate, cm3/min

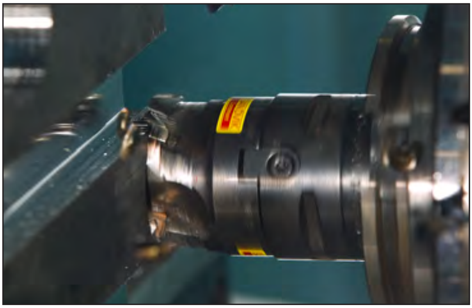
$$Q = \frac{a_p \times a_e \times v_f}{1000}$$

Net power, kW

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

Torque, Nm

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



Symbol	Designation/ definition	Unit
a_e	Working engagement	mm
a_p	Cutting depth	mm
D_{cap}	Cutting diameter at cutting depth a_p	mm
DC	Cutter diameter	mm
f_z	Feed per tooth	mm
f_n	Feed per revolution	mm/r
n	Spindle speed	rpm
v_c	Cutting speed	m/min
v_f	Table feed	mm/min
z_c	Number of effective teeth	pcs
h_{ex}	Maximum chip thickness	mm
h_m	Average chip thickness	mm
k_c	Specific cutting force	N/mm^2
P_c	Net power	kW
M_c	Torque	Nm
Q	Metal removal rate	cm^3/min
KAPR	Entering angle	degree

Specific cutting forces

$$k_c = k_{c1} \times \left(\frac{1}{h_m}\right)^{m_c} \times \left(1 - \frac{\gamma_0}{100}\right)$$

Formulas and definitions for milling - INCH

Table feed, inch/min

$$v_f = f_z \times n \times z_c$$

Cutting speed, ft/min

$$v_c = \frac{\pi \times D_{cap} \times n}{12}$$

Spindle speed, rpm

$$n = \frac{v_c \times 12}{\pi \times D_{cap}}$$

Feed per tooth, inch

$$f_z = \frac{v_f}{n \times z_c}$$

Feed per revolution, inch/rev

$$f_n = \frac{v_f}{n}$$

Metal removal rate, inch³/min

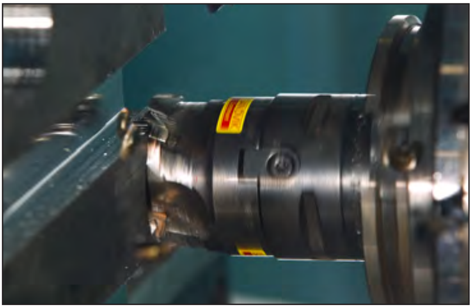
$$Q = a_p \times a_e \times v_f$$

Net power, HP

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

Torque, lbf ft

$$M_c = \frac{P_c \times 16501}{\pi \times n}$$



Symbol	Designation/ definition	Unit
a_e	Working engagement	inch
a_p	Cutting depth	inch
D_{cap}	Cutting diameter at cutting depth a_p	inch
DC	Cutter diameter	inch
f_z	Feed per tooth	inch
f_n	Feed per revolution	inch
n	Spindle speed	rpm
v_c	Cutting speed	ft/min
v_f	Table feed	inch/min
z_c	Number of effective teeth	pcs
h_{ex}	Maximum chip thickness	inch
h_m	Average chip thickness	inch
k_c	Specific cutting force	lbs/inch²
P_c	Net power	HP
M_c	Torque	lbf ft
Q	Metal removal rate	inch³/min
PSIR	Lead angle	degree

Specific cutting forces

$$k_c = k_{c1} \times \left(\frac{0.039}{h_m}\right)^{m_c} \times \left(1 - \frac{\gamma_0}{100}\right)$$

Formulas and definitions for drilling - METRIC

Penetration rate, mm/min

$$v_f = f_n \times n$$

Cutting speed, m/min

$$v_c = \frac{\pi \times DC \times n}{1000}$$

Spindle speed, r/min

$$n = \frac{v_c \times 1000}{\pi \times DC}$$

Feed force, N

$$F_f \approx 0.5 \times k_c \times \frac{DC}{2} f_n \times \sin KAPR$$

Metal removal rate, cm³/min

$$Q = \frac{v_c \times DC \times f_n}{4}$$

Net power, kW

$$P_c = \frac{v_c \times DC \times f_n \times k_c}{240 \times 10^3}$$

Torque, Nm

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

Symbol	Designation/ definition	Unit
DC	Drill diameter	mm
f_n	Feed per revolution	mm/r
n	Spindle speed	rpm
v_c	Cutting speed	m/min
v_f	Penetration rate	mm/min
F_f	Feed force	N
k_c	Specific cutting force	N/mm²
M_c	Torque	Nm
P_c	Net power	kW
Q	Metal removal rate	cm³/min
KAPR	Entering angle	degree

Formulas and definitions for drilling - INCH

Penetration rate, inch/min

$$v_f = f_n \times n$$

Cutting speed, ft/min

$$v_c = \frac{\pi \times DC \times n}{12}$$

Spindle speed, rpm

$$n = \frac{v_c \times 12}{\pi \times DC}$$

Feed force, N

$$F_f \approx 0.5 \times k_c \times \frac{DC}{2} \times f_n \times \sin PSIR$$

Note: DC needs to be converted into millimeters

Metal removal rate, inch³/min

$$Q = v_c \times DC \times f_n \times 3$$

Net power, HP

$$P_c = \frac{v_c \times DC \times f_n \times k_c}{132 \times 10^3}$$

Torque, lbf ft

$$M_c = \frac{P_c \times 16501}{\pi \times n}$$

Symbol	Designation/ definition	Unit
DC	Drill diameter	inch
f_n	Feed per revolution	inch/r
n	Spindle speed	rpm
v_c	Cutting speed	ft/min
v_f	Penetration rate	inch/min
F_f	Feed force	N
k_c	Specific cutting force	lbs/inch²
M_c	Torque	lbf ft
P_c	Net power	HP
Q	Metal removal rate	inch³/min
PSIR	Lead angle	degree

Formulas and definitions for boring - METRIC

Penetration rate, mm/min

$$v_f = f_n \times n$$

Cutting speed, m/min

$$v_c = \frac{\pi \times DC \times n}{1000}$$

Spindle speed, r/min

$$n = \frac{v_c \times 1000}{\pi \times DC}$$

Feed per revolution, mm/r

$$f_n = z_c \times f_z$$

Metal removal rate, cm3/min

$$Q = \frac{v_c \times DC \times f_n}{4}$$

Net power, kW

$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{60 \times 10^3}\left(1 - \frac{a_p}{DC}\right)$$

Torque, Nm

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

Symbol	Designation/ definition	Unit
DC	Drill diameter	mm
f_n	Feed per revolution	mm/r
n	Spindle speed	rpm
v_c	Cutting speed	m/min
v_f	Table speed	mm/min
F_f	Feed force	N
k_c	Specific cutting force	N/mm^2
M_c	Torque	Nm
P_c	Net power	kW
Q	Metal removal rate	cm^3/min
KAPR	Entering angle	degree
z_c	Number of effective teeth (z_c = 1 for step boring)	pcs

Feed force, N

$$F_f \approx 0.5 \times k_c \times a_p \times f_n \times \sin KAPR$$

Formulas and definitions for boring - INCH

Penetration rate, inch/min

$$v_f = f_n \times n$$

Cutting speed, ft/min

$$v_c = \frac{\pi \times DC \times n}{12}$$

Spindle speed, rpm

$$n = \frac{v_c \times 12}{\pi \times DC}$$

Feed per revolution, inch/rev

$$f_n = z_c \times f_z$$

Metal removal rate, inch³/min

$$Q = v_c \times DC \times f_n \times 3$$

Net power, HP

$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{132 \times 10^3}\left(1 - \frac{a_p}{DC}\right)$$

Torque, lbf ft

$$M_c = \frac{P_c \times 16501}{\pi \times n}$$

Symbol	Designation/ definition	Unit
DC	Drill diameter	inch
f_n	Feed per revolution	inch/r
n	Spindle speed	rpm
v_c	Cutting speed	ft/min
v_f	Table speed	inch/min
F_f	Feed force	N
k_c	Specific cutting force	lbs/inch²
M_c	Torque	lbf ft
P_c	Net power	HP
Q	Metal removal rate	inch³/min
PSIR	Lead angle	degree
z_c	Number of effective teeth (z_c = 1 for step boring)	pcs

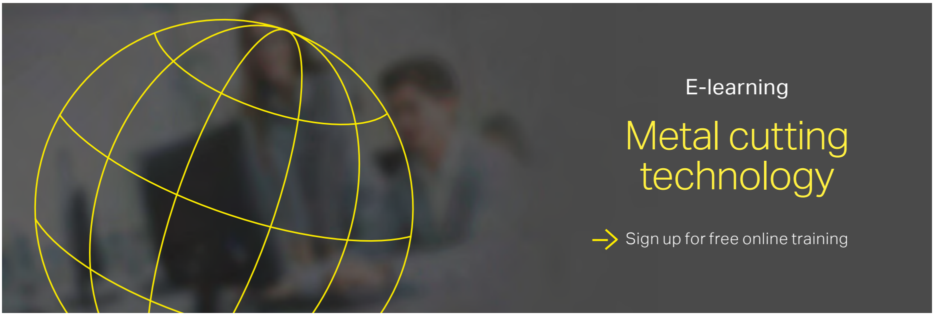
Feed force, N

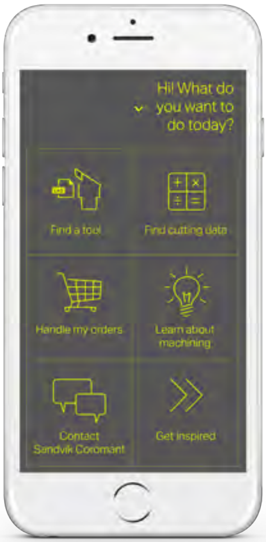
$$F_f \approx 0.5 \times k_c \times a_p \times f_n \times \sin KAPR$$

University level online training

The Metal Cutting Technology e-learning is the ideal complement to a classroom education - or the perfect solution to develop new skills in whatever location you may be. Learn the fundamentals of Metal Cutting Technology through the extensive 75 courses, covering most application areas of metal cutting. Save your progress and track your achievements as you advance.

Register at metalcuttingknowledge.com and start your free training.





Support in the palm of your hand

Learning metal cutting requires the mindset of an engineer and strong support along the way. Sandvik Coromant provides several advanced and useful calculators and apps for your mobile devices, free of charge.

Find the support you need on sandvik.coromant.com/apps

Head office:
AB Sandvik Coromant
SE-811 81 Sandviken, Sweden
E-mail: info.coromant@sandvik.com
www.sandvik.coromant.com

SANDVIK
Coromant