PART 2-2

SANDVIK Coromant

Training Handbook

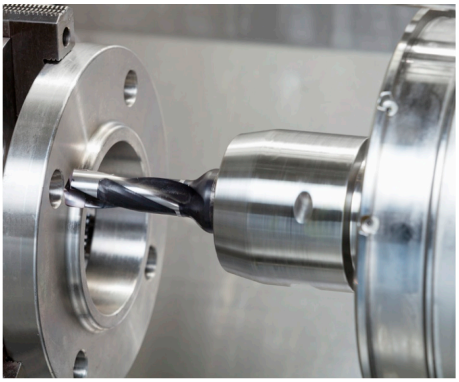
METAL CUTTING TECHNOLOGY

Drilling

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

- Theory E 4
- Selection procedure E 15
- System overview E 20
- How to apply E 26
- Hole quality and tolerances E 38
- Troubleshooting E 43

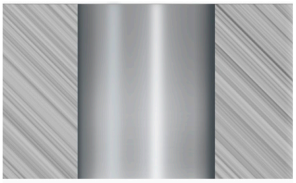
The drilling process



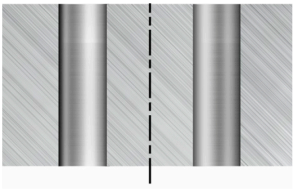
- The drill is always engulfed in the workpiece, leaving no view of the operation.
- Chips must be controlled.
- Chip evacuation is essential; it affects hole quality, tool life and reliability.

Four common drilling methods

Drilling



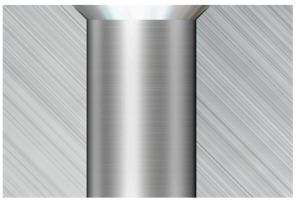
Trepanning



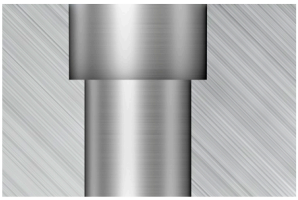
Drilling is classified into four common methods:

- Drilling
- Trepanning
- Chamfer drilling
- Step drilling.

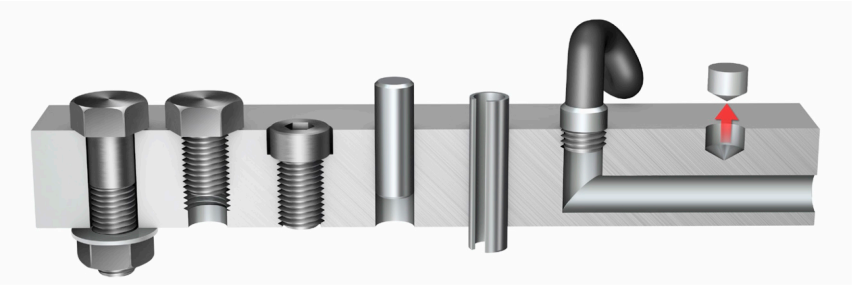
Chamfer drilling



Step drilling



The most common holes



1 2 3 4 5 6 7

The most common holes are:

1 Holes with clearance for bolts

2 Holes with a screw thread

3 Countersink holes

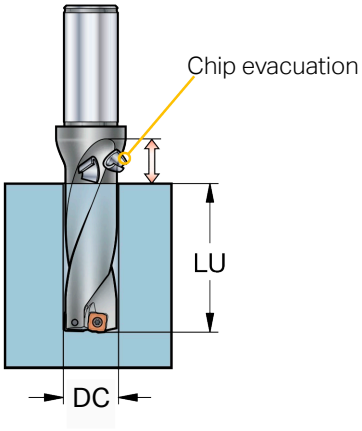
4 Pressed fit holes

5 Slip fit holes

6 Holes that form channels

7 Holes to remove weight for balancing.

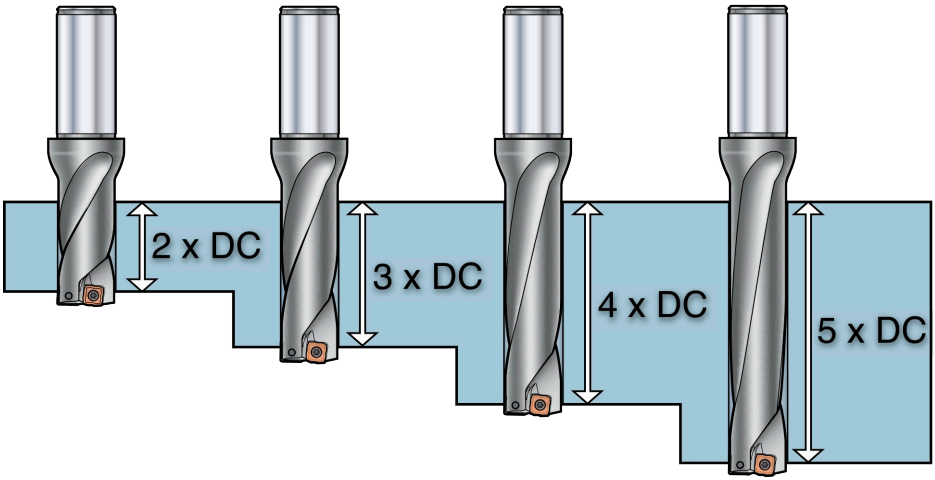
Maximum hole depth



Hole depth (LU) determines the choice of tool.

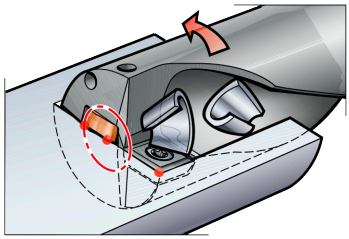
Maximum hole depth is a function of hole diameter DC and hole depth (LU).

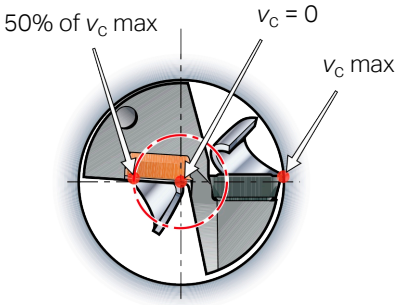
Example: max hole depth LU = 3 x DC.



Drilling theory

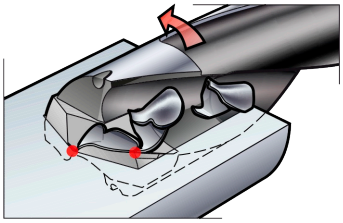
Cutting speeds for indexable drills

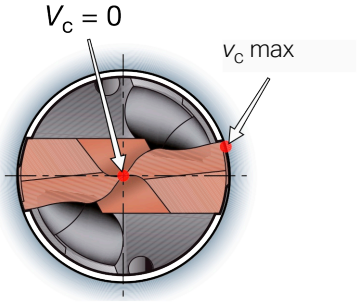




- Cutting speed (v_c) for indexable drills declines from 100% at the periphery to zero at the center.
- The central insert operates from cutting speed zero to approx. 50% of v_c max. The peripheral insert works from 50% of v_c max up to 100% of v_c max.
- One effective cutting edge/rev: = z_c.

Cutting speeds for solid and exchangeable tip drills



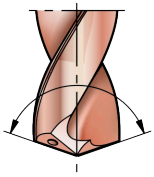


- Two effective cutting edges, from the center to the periphery.
- Two edges/rev: = z_c.

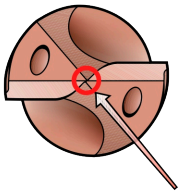
Solid carbide drill (SCD) vs. high speed drills (HSS)

Point angle and chisel edge

Solid carbide drill



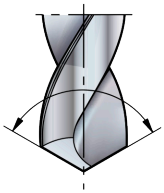
140° point angle



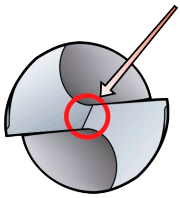
Chisel edge

- 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.

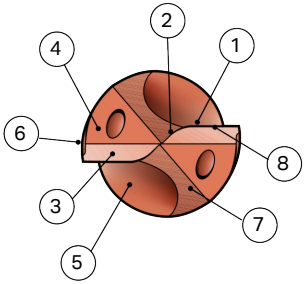
HSS drill

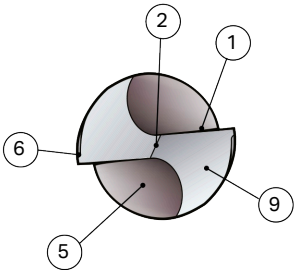


118° point angle



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.



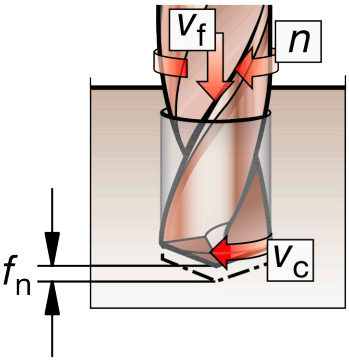


Solid carbide drill - Advantages

- 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

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

Inch

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

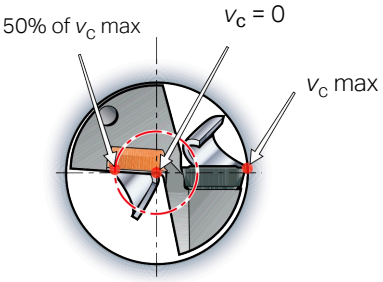
$$v_f = f_n \times n \text{ mm/min (inch/min)}$$

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.

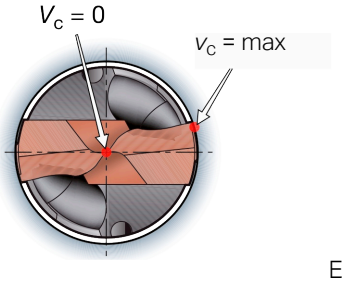
One effective cutting edge/rev: = z_c.



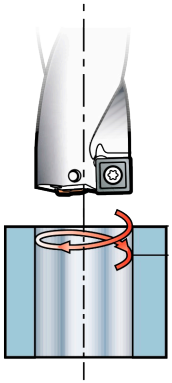
Cutting speeds for solid and exchangeable tip drills

Two edges, from the center to the periphery.

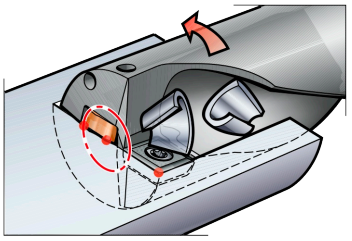
Two edges/rev: = z_c.



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

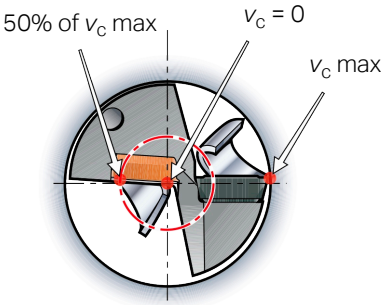


- Affects the power P_c kW (Hp) and torque M_c Nm (lbf-ft).
- The largest factor determining tool life.
- Higher speed generates higher temperature and increased flank wear, especially on the peripheral corner.
- Higher speed is beneficial for chip formation in long chipping, soft materials, i.e., low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

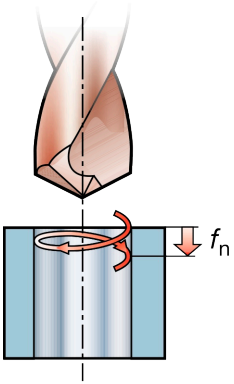


Too high cutting speed causes:
- rapid flank wear
- plastic deformation
- poor hole quality
- bad hole tolerance.

Too low cutting speed causes:
- built-up edge
- bad chip evacuation
- longer time in cut
- higher risk of drill breakage
- reduced hole quality.



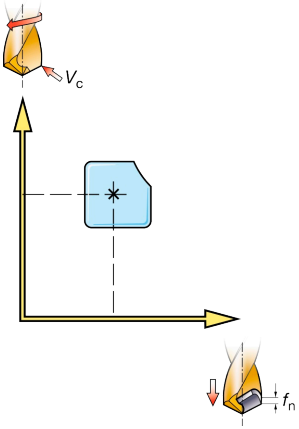
Feed rate



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.

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



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

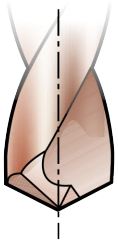
Low feed rate:
- longer, thinner chips
- quality improvement
- accelerated tool wear
- longer time in cut.

**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/mm^2 (lbf ft/inch^2)
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

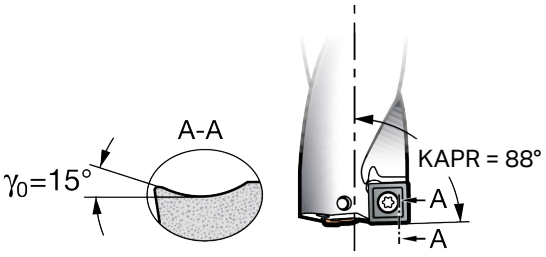
$$P_c = \frac{f_n \times v_c \times DC \times k_c}{132 \times 10^3} \text{ Hp}$$

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

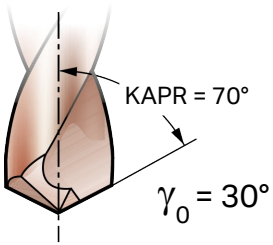
For information about the k_{c1} value, see page H16.

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

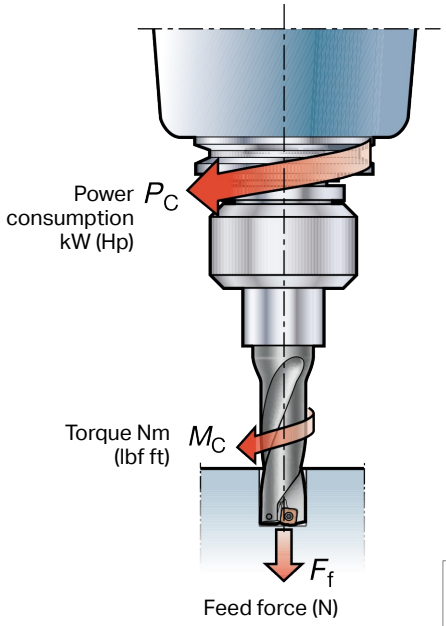
$$P_c = \frac{f_n \times v_c \times DC \times k_c}{132 \times 10^3} \text{ Hp}$$

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

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

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

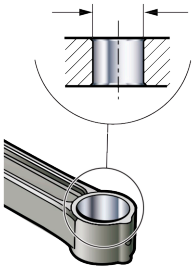
Tool selection procedure

Production planning process

1 Component

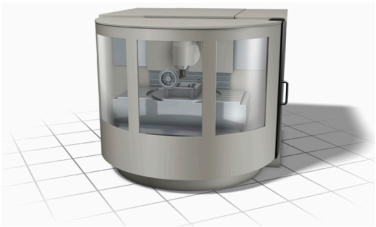
Hole dimension and quality

Workpiece material, shape and quantity



2 Machine

Machine parameters

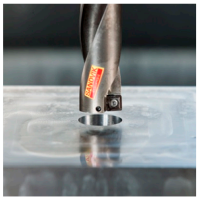


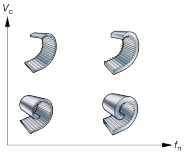
3 Choice of tool

Type of tool and machining method

4 How to apply

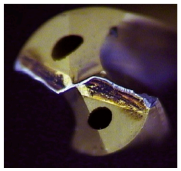
Cutting data, coolant, etc.

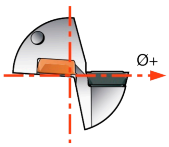




5 Troubleshooting

Remedies and solutions

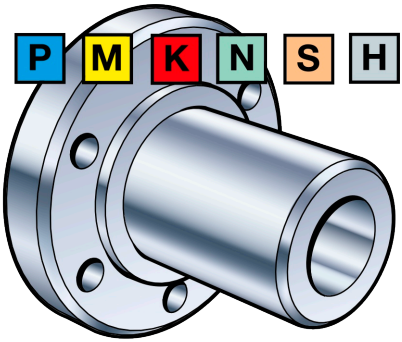




1. Component and the workpiece material

Material:

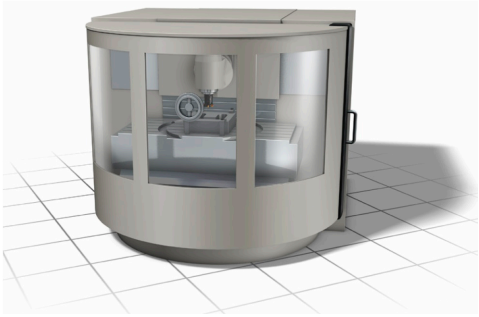
- Machinability
- Chip breaking
- Hardness
- Alloy elements.



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

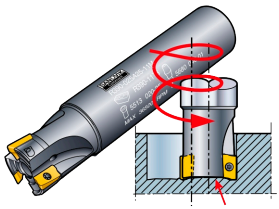
Advantages

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

Disadvantages

- Requires more power and stability
- Less flexibility.

Helical interpolation



Advantages

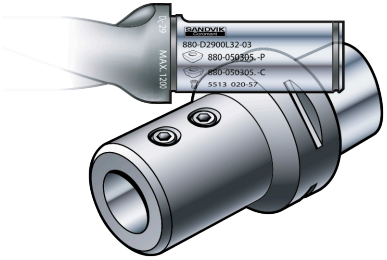
- Simple standard tools
- Very flexible
- Low cutting forces.

Disadvantages

- Longer cycle times.

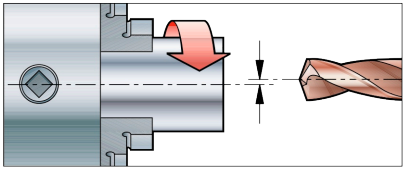
4. How to apply

Important application considerations



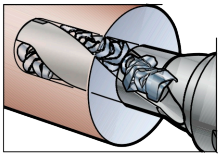
Tool holding

- 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.



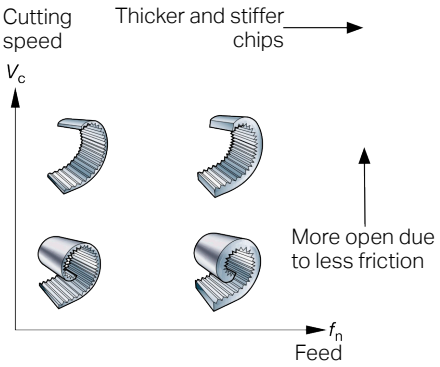
Tool runout

- 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

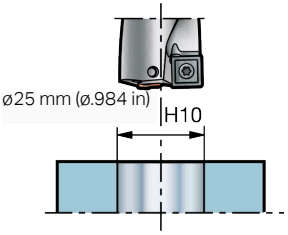
- Use recommended grade and geometry.
- Use recommended cutting parameters.
- To ensure a stable process, make sure to achieve good chip formation by adjusting cutting parameters.

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

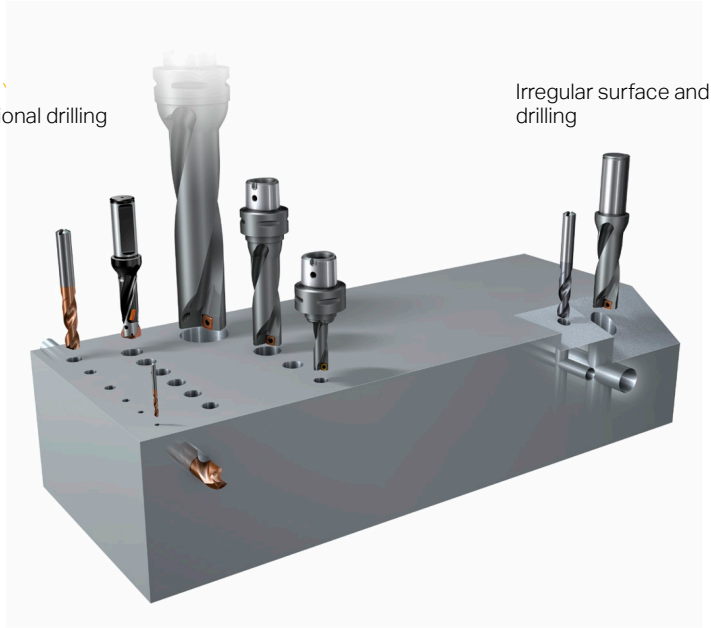
- Check clamping of drill/workpiece, feed rate, machine conditions and chip evacuation.

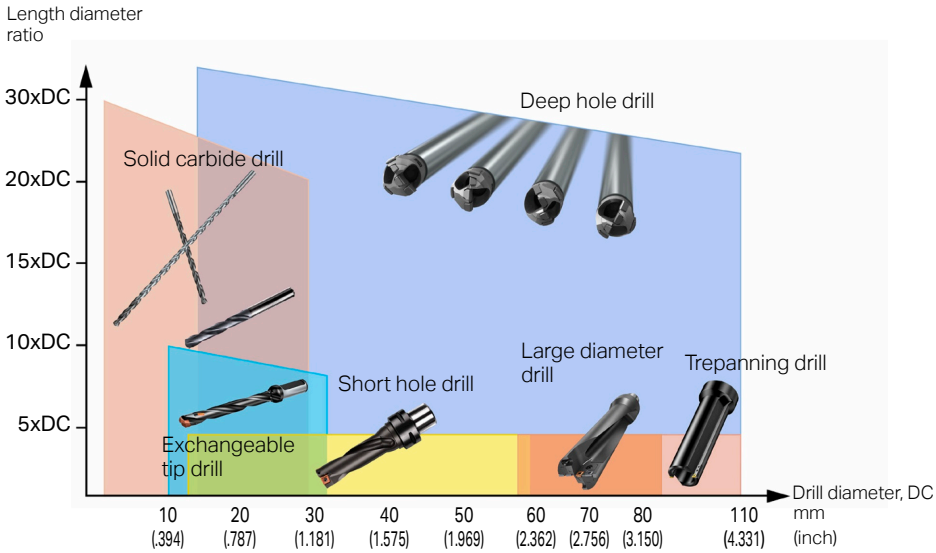
Cutting data

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

Drilling tools

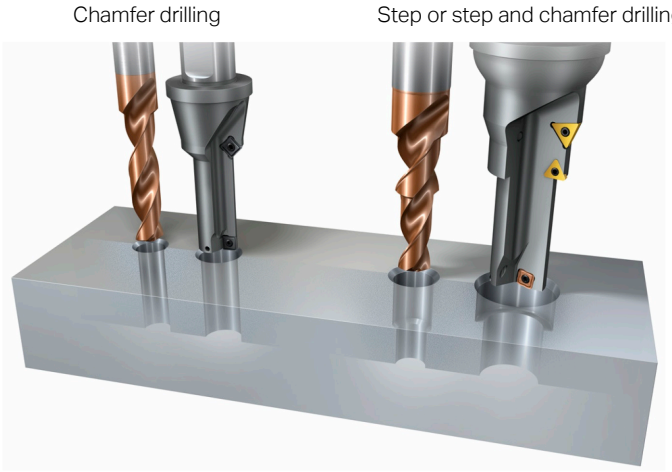
Drilling tools covering diameters from 0.30 mm up to 110 mm (.0118 inch up to 4.331 inch) and even larger as engineered products.



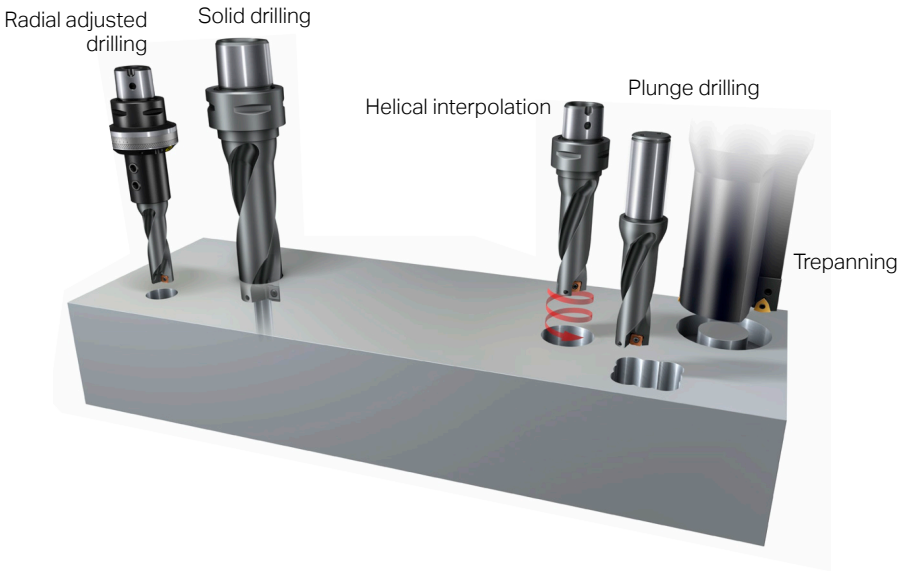


Choice of drilling tools

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

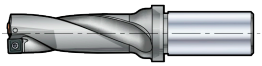
Step Technology™

- 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.

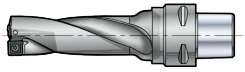
Mounting options

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.

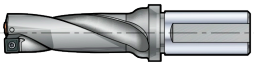
Cylindrical shank



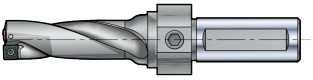
Coromant Capto® coupling



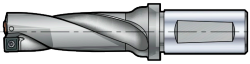
Cylindrical with flat



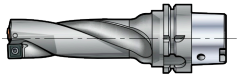
P-shank



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

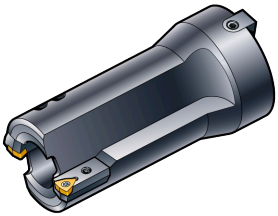
ISO material group		P	M	K	N	S	H
	Solid carbide drills	+++	+++	+++	+++	+++	+++
	Exchangeable tip drills	+++	+++	+++	++	++	+
	Indexable insert drills	+++	+++	+++	+++	+++	+++

Large hole diameters

Large diameter drill

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

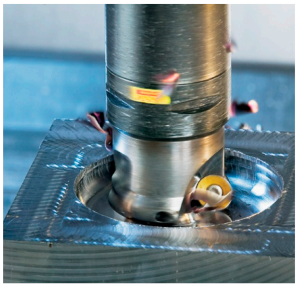
Trepanning drill



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.

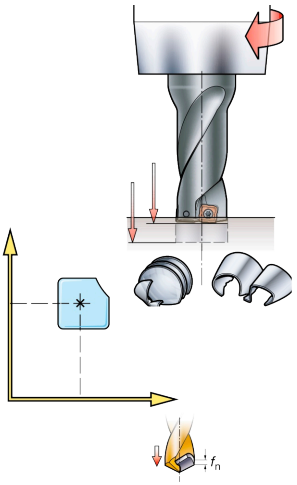
Milling, helical interpolation



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

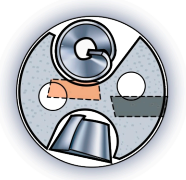


Setup routine

- 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

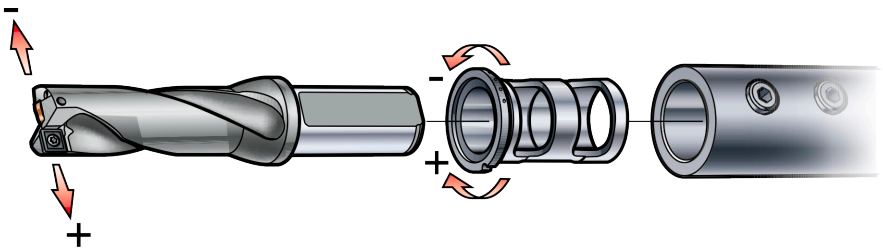
- If over- or under-sized holes are produced or if the center insert tends to chip, it is often because the drill is off center.
- Turning the drill 180° in its holder may solve this problem.
- But it is important to ensure that the center axis of the drill and the axis of rotation are parallel in order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

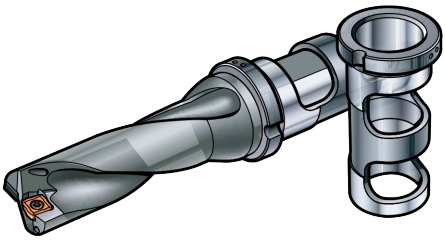
Radial adjustment

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





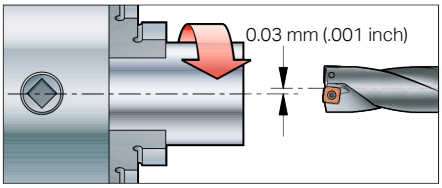
Rotating drill – eccentric sleeve

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).

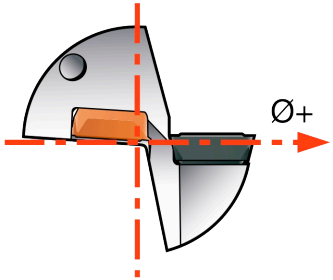
- One dot increases/decreases the diameter by 0.10 mm (.004 inch).
- Increase the diameter by turning the sleeve clockwise.
- Decrease the diameter by turning the sleeve counterclockwise.
- Use both screws to clamp the drill in the fixture and make sure the bolts in the holder are long enough.

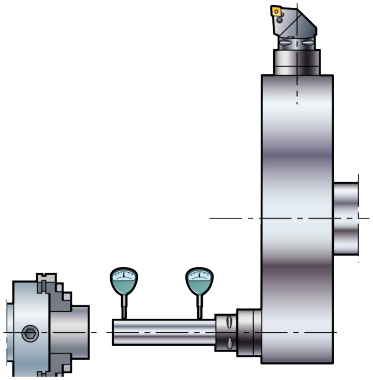
Non-rotating drill

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).





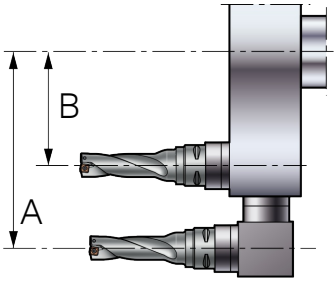
Dial indicator and test bar

- 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

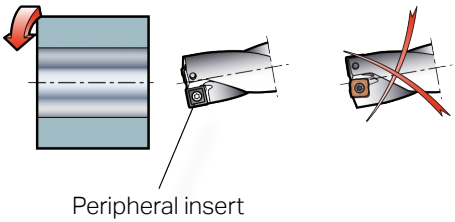
- Another way is by making a drill with four flats equally positioned around the drill shank.
- Make holes with the drill mounted in each of the four flat positions. Hole measurement will indicate the state of machine alignment.

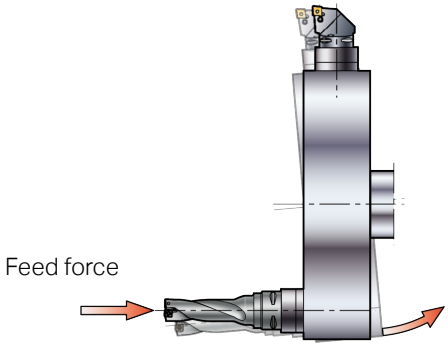
Deflection of turret



Problem solving

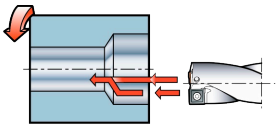
- Deflection of the turret on a CNC lathe can be caused by the feed force.
- First, check if you can minimize torque by mounting the tool differently.
  Position B is preferable to position A.

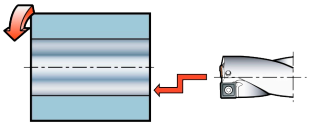


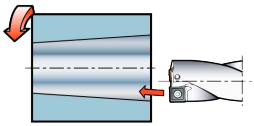


- To avoid wear on the drill body and retraction marks in the hole, mount the drill with the peripheral insert as shown in the picture.
- Finally, a reduction of the feed/revolution (f_n) can be made to minimize the feed force.

Radial offset

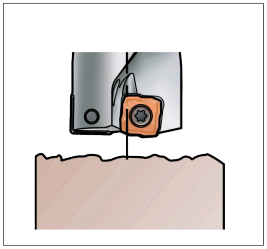


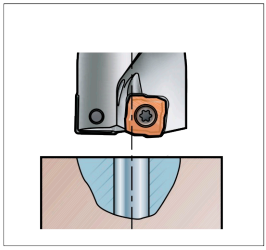




- Holes can be drilled larger than the nominal size of the drill as well as enlarged and finished with a subsequent boring pass.
- Non-rotating indexable insert drills can also be used to generate tapered holes.
- Also chamfering and reliefs can be machined with the drill.
- A hole which is to be threaded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

Irregular surfaces and pre-drilled holes

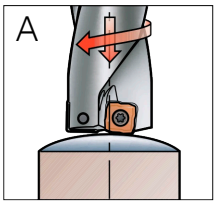




When entering or exiting an irregular surface there is a risk of the inserts chipping.

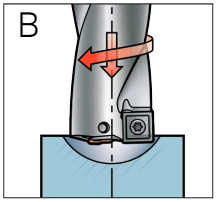
- The feed rate should therefore be reduced.
- A pre-drilled hole should be small rather than large - not more than 25% of the drill diameter - to avoid drill deflection.
- However, reduced feed does allow broad machining of pre-drilled holes.

Entering non-flat surfaces



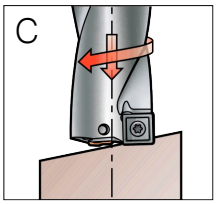
Convex surface

- Normally no feed reduction needed.



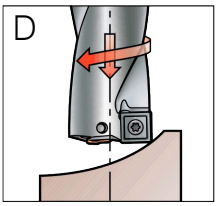
Concave surface

- 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.



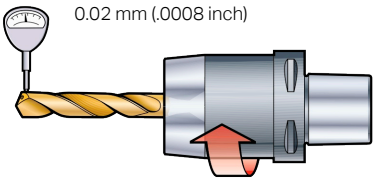
Irregular surfaces

- Reduce feed 1/3 of original feed rate.

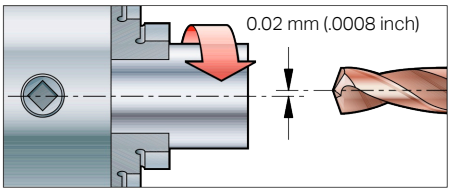
Solid carbide and exchangeable tip drills

Alignment

Rotating drill



Stationary drill

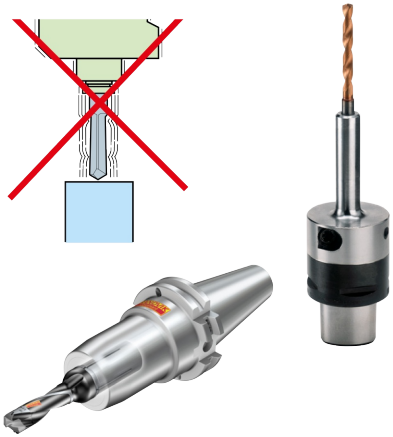


Minimum tool runout is one of the main criteria for successful use of solid carbide drills.

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

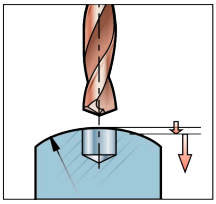
- Not recommended due to risk of chipping on cutting edge.

Exchangeable-tip drills

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

Entering non-flat surfaces

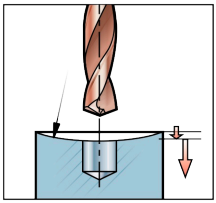
When entering non-flat surfaces there is a risk of drill deflection. To avoid this, the feed can be reduced when entering.



Convex surface

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

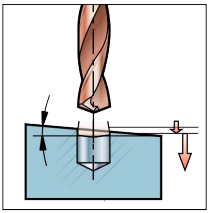
Reduce feed 50% of normal rate during entrance.



Concave surface

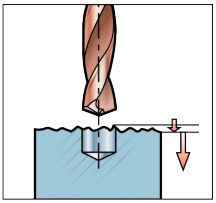
Drill if radius is > 15 times drill diameter and the hole is perpendicular to the radius.

Reduce feed 25% of normal rate during entrance.



Inclined surface

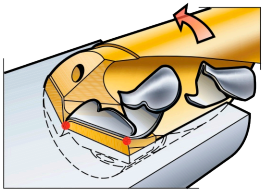
Inclinations up to 10°, reduce the feed to 1/3 of normal feed rate during entrance. More than 10°, not recommended. Mill a small flat on surface, then drill the hole.



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).

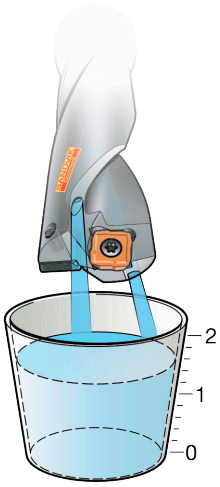
External coolant supply

- 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.

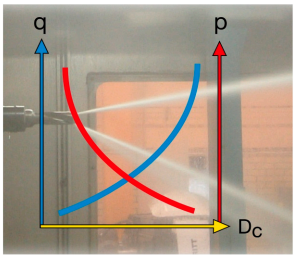
Mist cutting fluid or minimal lubrication

- 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

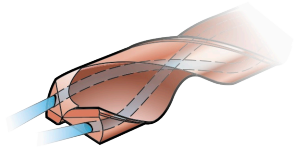


Coolant supply is essential in drilling and influences:
- chip evacuation
- hole quality
- tool life.

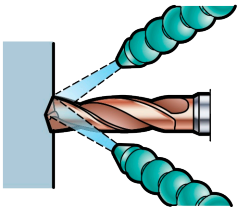
- The cubic capacity of the coolant tank should be between 5-10 times larger than the volume of coolant that the pump supplies per minute.
- The volume capacity can be checked using a stopwatch and a suitably-sized bucket.

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").

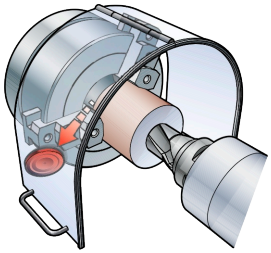


External coolant supply
- Can be acceptable in short-chipping materials.
- To improve chip evacuation at least one coolant nozzle (two if drill is stationary) should be directed close to the tool axis.
- Can sometimes help to avoid built-up edge formation due to a higher edge temperature.

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

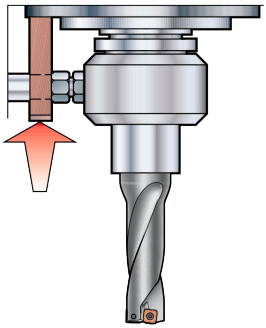
Internal coolant supply



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

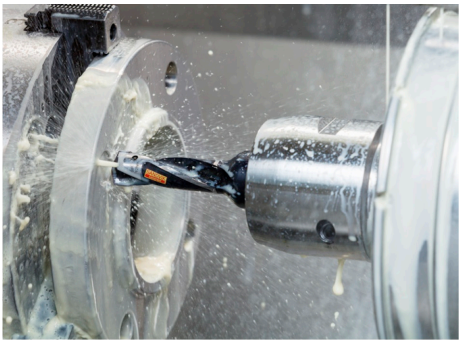


Rotating stop is an important measure

- 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.

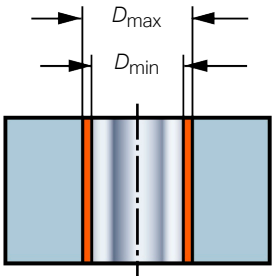
Hole quality and tolerance

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).

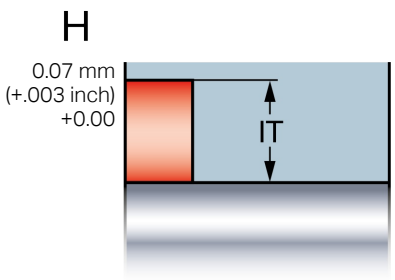
D_{max} minus D_{min} is the tolerance width, also called, e.g., IT 7.

Hole tolerance according to ISO

Tolerance	Diameter range, mm/*inch*							Examples
	3–6	6–10	10–18	18–30	30–50	50–80	80–120	
	.118–.236	*.236–.394*	*.394–.709*	*.709–1.181*	*1.181–1.969*	*1.969–3.150*	*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*	

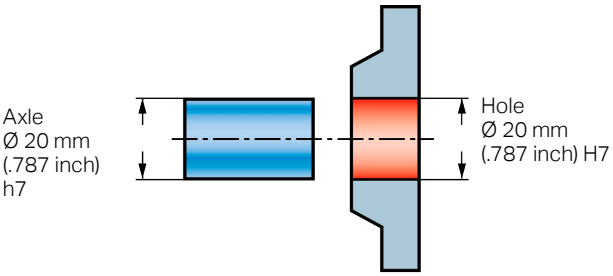
[1] Holes for threading with fluteless taps (rolled threads)

- 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)

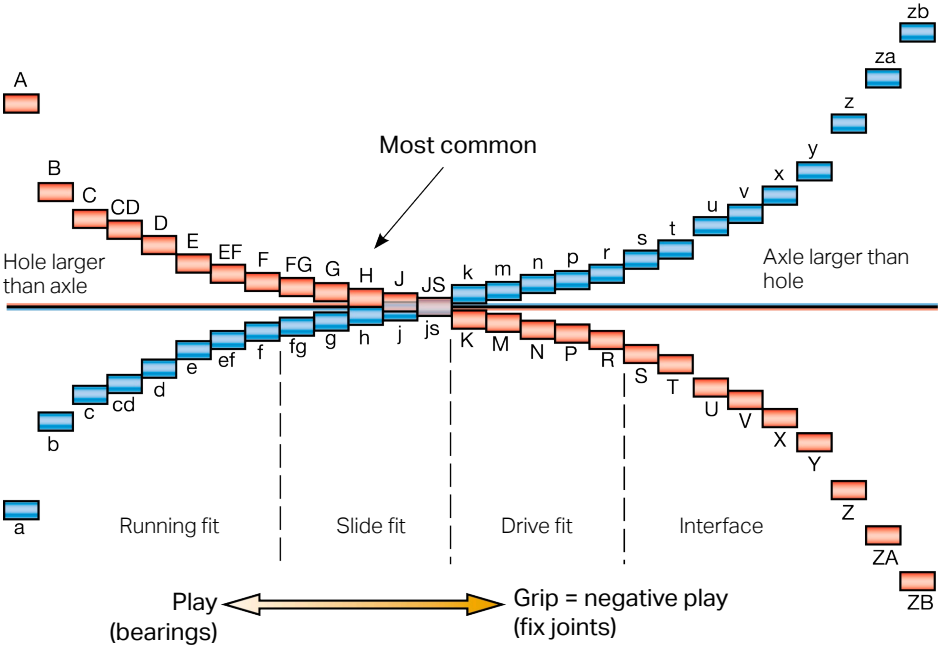
Hole tolerances according to ISO



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

Hole and axle tolerance according to ISO

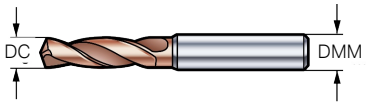
Axle tolerance position is denominated by lower case letters corresponding to the hole tolerance in upper case letters. The figure below gives a complete picture.



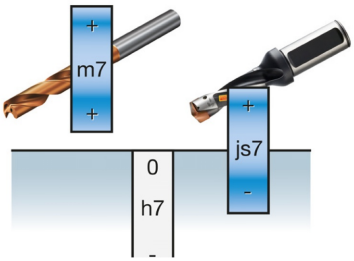
Hole and tool tolerance

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 exchangeable tip drills, the hole tolerance is very close to the drill tolerance.

Tolerance	Solid carbide drills	Exchangeable tip drills	Indexable insert drill
IT6			
IT7			
IT8			
IT9			With pre-setting
IT10			
IT11			
IT12			
IT13			

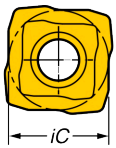
Indexable insert drills

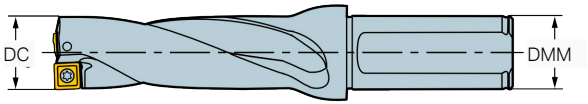
Drill tolerance

- 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.



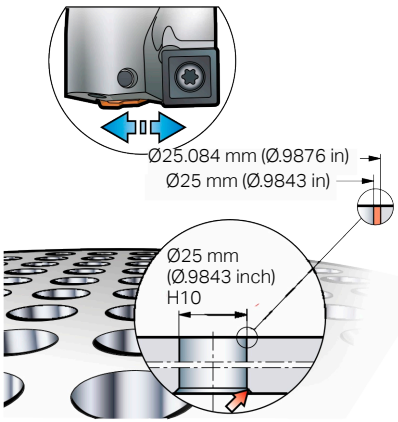


Drill depth 2-3 x DC

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.

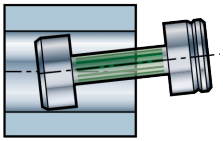
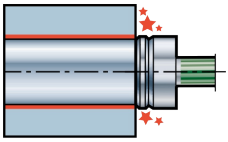
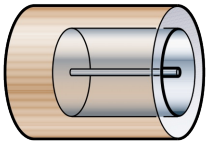
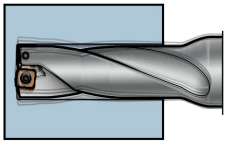
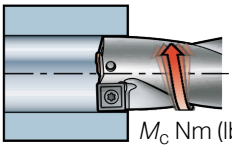
This can be done in a lathe or with an adjustable holder/sleeve, see page E28.

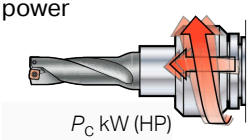
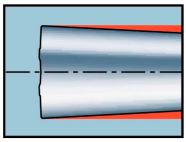
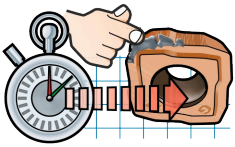
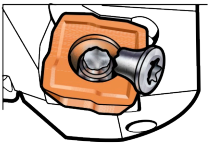
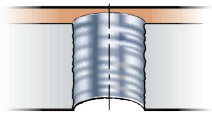
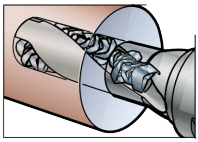
A tolerance width (IT) inside 0.10 mm (.004 inch) can then be obtained.

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  M_c Nm (lbf-ft)	1. Reduce feed. 2. Choose a light cutting geometry to lower the cutting force.	

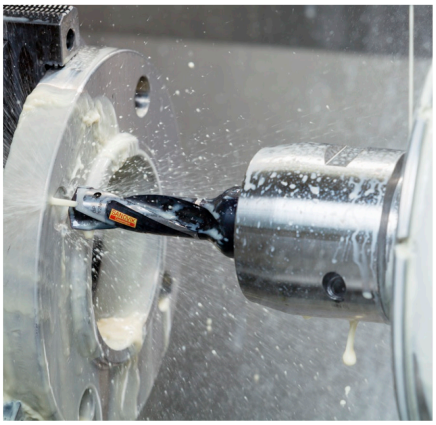
Problem	Solution
Insufficient machine power  P_C kW (HP)	1. Reduce cutting speed. 2. Reduce cutting feed. 3. Choose a light cutting geometry to lower the cutting force.
Hole not symmetrical 	Hole widens at bottom (due to chip jam on center insert) 1. Increase coolant flow, clean filter, clear coolant 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

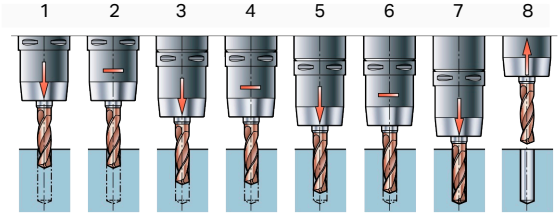


Checkpoints and remedies

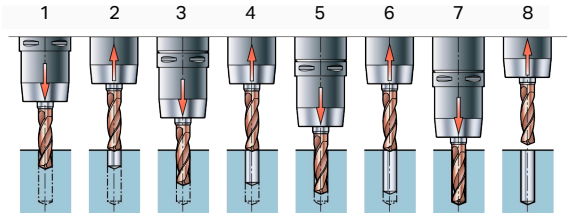
1. Make sure the right cutting data and drill geometry are used.
2. Inspect chip form (compare with picture on page E 26).
3. Check if the cutting fluid flow and pressure can be increased.
4. Inspect the cutting edges. Chipping on the edge can cause long chips because the chip is divided. Also a large 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:

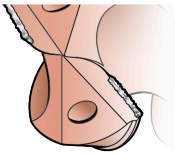
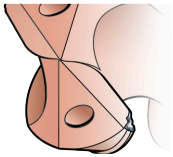
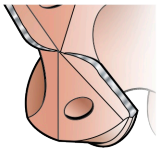
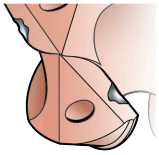
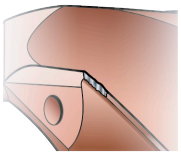


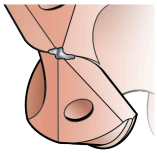
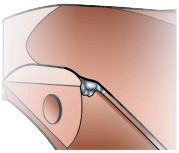
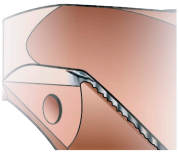
- Method 1 for best productivity
Do not retract the drill more than approx. 0.3 mm (.012 inch) from the hole bottom. Alternatively, make a periodical stop, while the drill is still rotating, before continuing to drill.



- 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

	Cause	Solution
Built-up edge		
	1. Cutting speed too low and edge temperature too high 2. Negative land too large 3. No coating 4. Percentage of oil in the cutting fluid too low	1. Increase cutting speed or use external cutting fluid 2. Sharper cutting edge 3. Coating on the edge 4. Increase the percentage of oil in the cutting fluid
Chipping on the edge corner		
	1. Unstable fixturing 2. TIR too large 3. Intermittent cutting 4. Insufficient cutting fluid (thermal cracking) 5. Unstable tool holding	1. Check fixture 2. Check radial runout 3. Decrease the feed 4. Check cutting fluid supply 5. Check the tool holder
Flank wear on the cutting edges		
	1. Cutting speed too high 2. Feed too low 3. Grade too soft 4. Lack of cutting fluid	1. Decrease the cutting speed 2. Increase the feed 3. Change to harder grade 4. Check for proper cutting fluid supply
Chipping on the cutting edge		
	1. Unstable conditions 2. Maximum allowed wear exceeded 3. Grade too hard	1. Check the setup 2. Replace drill sooner 3. Change to softer grade
Wear on the circular lands		
	1. TIR too large 2. Cutting fluid too weak 3. Cutting speed too high 4. Abrasive material	1. Check the radial runout 2. Use neat oil or stronger emulsion 3. Decrease cutting speed 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

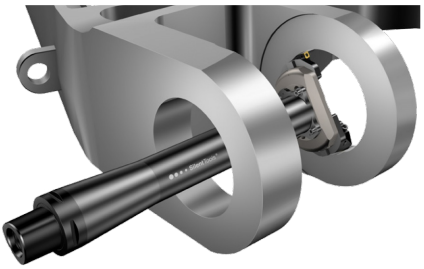
Boring operations involving rotating tools are applied to machine holes that have been made through methods such as pre-machining, casting, forging, extrusion, flame-cutting, etc.

- Theory F 4
- Selection procedure F 8
- System overview F 13
- Choice of tool F 16
- How to apply F 22
- Troubleshooting F 27

Boring theory

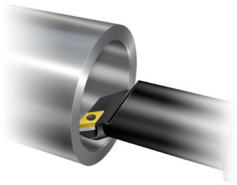
The boring process

- 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

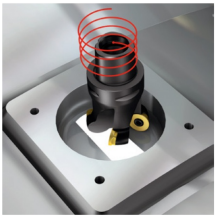


- To be used only for symmetrical components in a turning lathe.
- Profiling can be carried out with standard boring bars.
- Very flexible tool solutions with interchangeable cutting heads.

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.

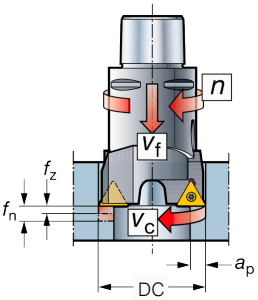
Milling, helical interpolation



- 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.

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

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

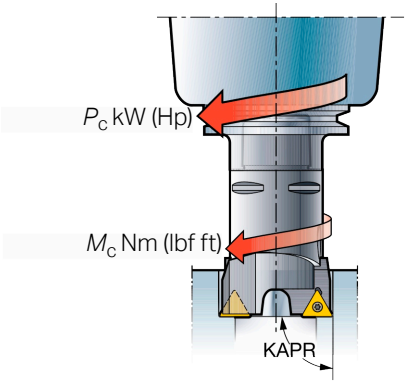
Penetration rate

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



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.

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.

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).

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

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, kW

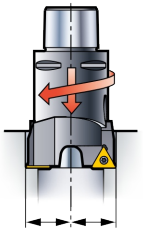
$$P_c = \frac{v_c \times a_p \times f_n \times k_c}{60 \times 10^3} \left(1 - \frac{a_p}{DC}\right)$$

Net power, HP

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

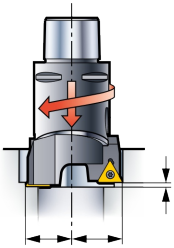
Hole making methods

Productive boring



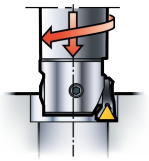
Productive boring involves 2-3 cutting edges and is used for roughing operations of hole tolerances of IT9 or larger, where metal removal rate is the 1st priority. In multi edge boring all slides are set to the same diameter and height. The feed rate is given by multiplying the feed for each insert by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

Step boring



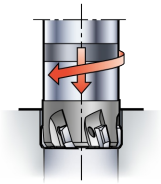
In Step boring the slides are set to different axial heights and diameters. Step boring is used where large radial depth of cuts are required or when boring in soft material (long chipping material). The width of the chip is divided into two small easy to handle chips by this method. The feed rate and 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

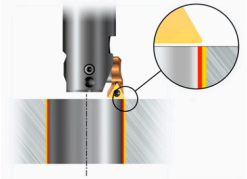


Reaming is a light finishing operation performed with a multi-edge reamer at high feeds.

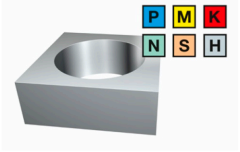
Tool selection procedure

Production planning process

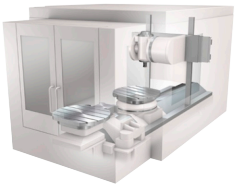
1 Component — Hole dimension and quality



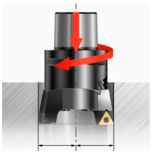
1 Component — Workpiece material, shape and quantity

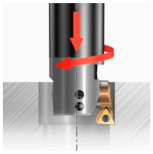


2 Machine — Machine parameters

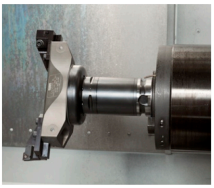


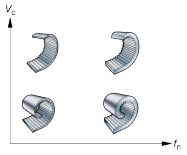
3 Choice of tool — Type of tool



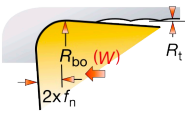


4 How to apply — Cutting data, coolant, etc.





5 Troubleshooting — Remedies and solutions



1. Component and the workpiece material

Parameters to be considered

P M K
N S H

Component

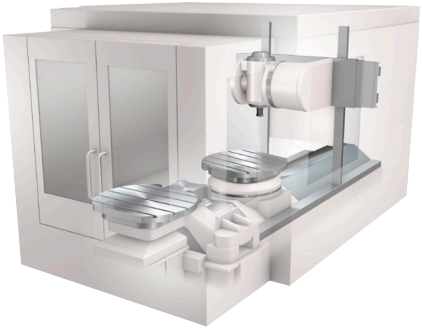
- Identify the type of operation and note characteristics regarding the hole to be machined, limitations, material and machine.
- Clamping, clamping forces and cutting forces. Is the component sensitive to vibrations?
- Select the tool that covers the boring diameter range and depth for the operation,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

Material

- Machinability
- Chip breaking
- Hardness
- Alloy elements.

2. Machine parameters

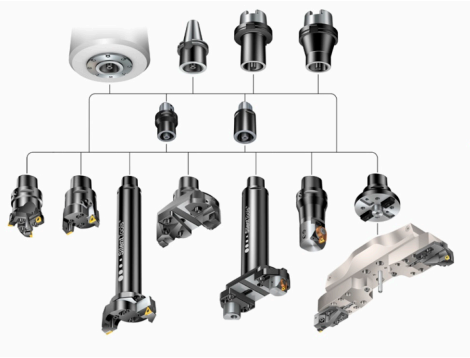
Condition of the machine

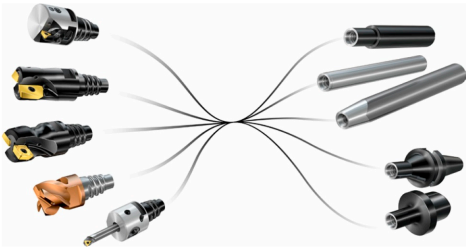


- Spindle interface
- Machine stability
- The spindle speed
- Coolant supply
- Coolant pressure
- Clamping of the workpiece
- Horizontal or vertical spindle
- Power and torque
- Tool magazine.

3. Choice of tools

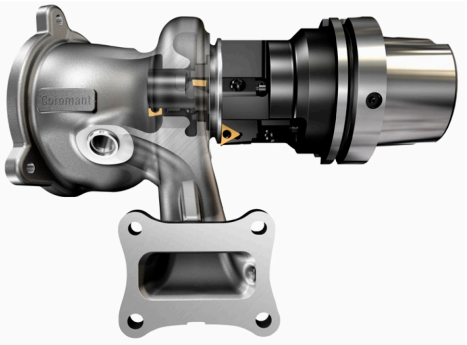
Bending stiffness and torque transmission are the foremost important factors when choosing a tool holder for boring operations. Choose the tool according to your specific needs:





- 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.

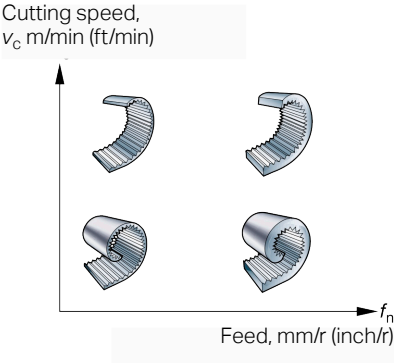
Engineered solutions

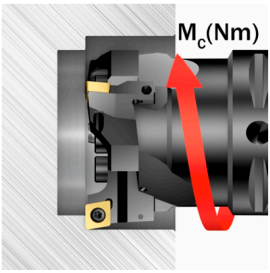


- Often a combination of multiple operations in one tool.
- The operations can be completed during one feed motion.

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

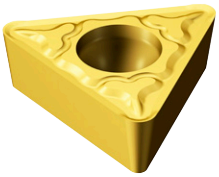
- Chip formation and evacuation are important factors in boring and affect hole quality and hole tolerance.

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.

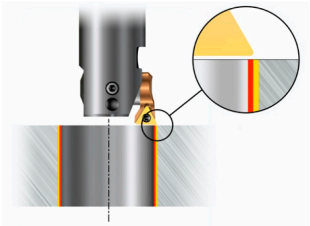
5. Troubleshooting

Important application considerations



Insert wear and tool life

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

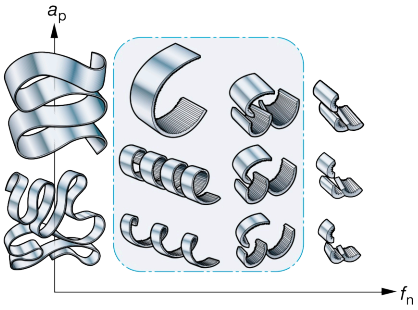


Chip evacuation

- Check the chip breaking and cutting fluid supply.

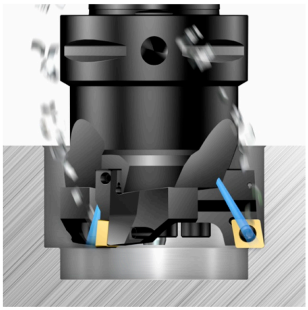
Hole quality and tolerances

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



Cutting data

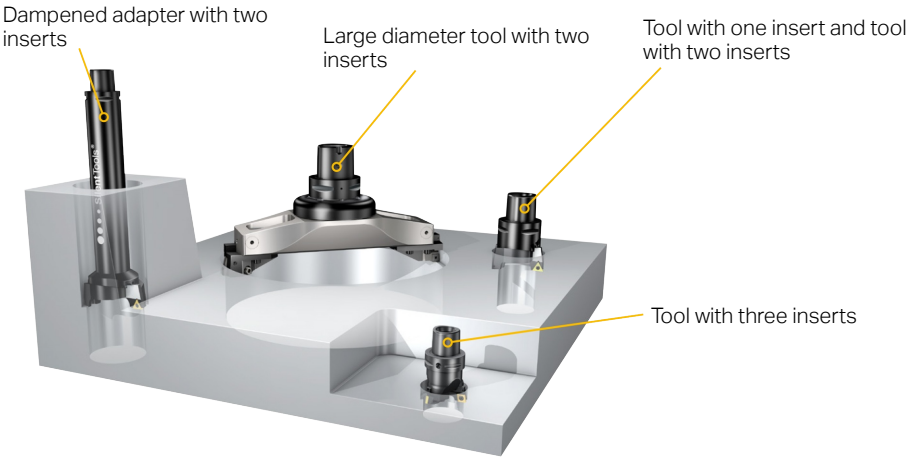
- Correct cutting speed, feed rate and cutting depth is essential for high productivity, tool life and to avoid vibrations.



System overview

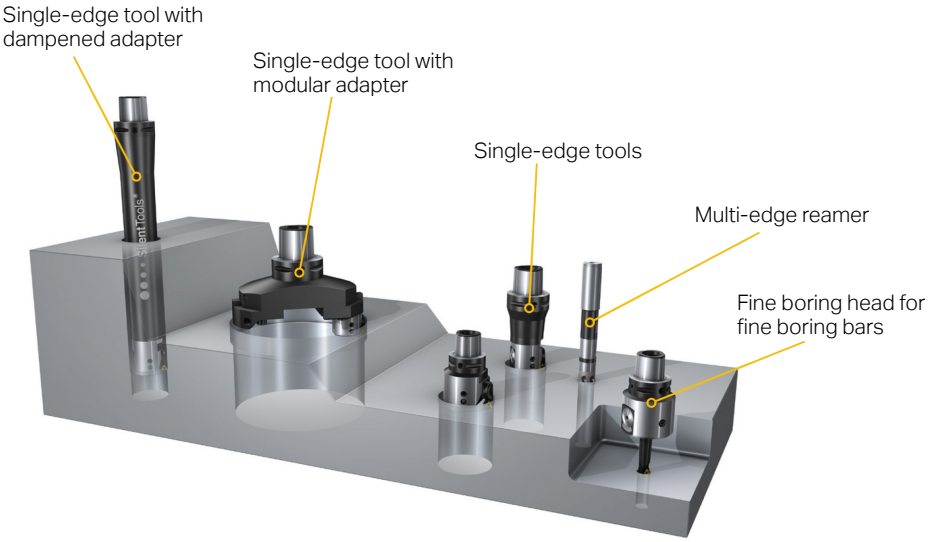
Rough boring tools

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

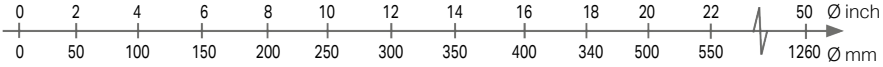


Fine boring tools

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



Rough boring



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

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

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

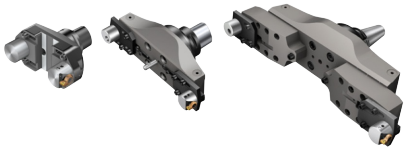
Fine boring with cylindrical shank Ø19-36 mm (0.75-1.42")

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")

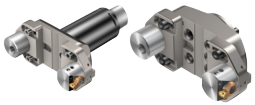
Fine boring with Coromant Capto (lightweight) Ø69-167 mm (2.716-6.575")

Fine boring – large diameters



Fine boring Ø150-1275mm (5.9-50")

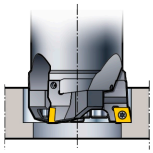
Fine boring (dampened) Ø150-315mm (5.9-12.4")



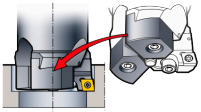
Fine boring with Coromant Capto or arbor mount (lightweight) Ø150-315mm (5.9-12.4")

Choice of tools

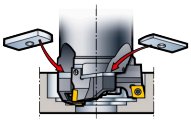
Roughing



Productive boring



Single-edge boring



Step boring

Productive boring

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

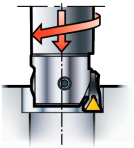
Single-edge boring

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

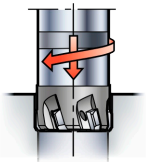
Step-boring

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

Finishing



Single-edge boring



Reaming

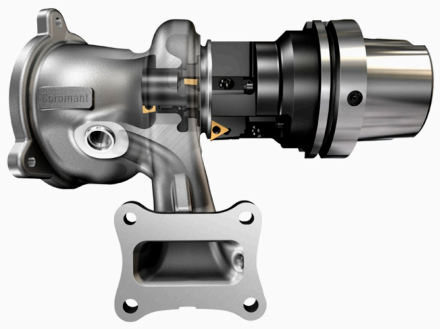
Single-edge boring

- 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.

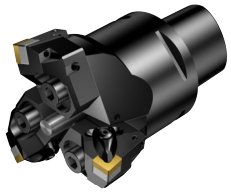
Engineered solutions



- 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

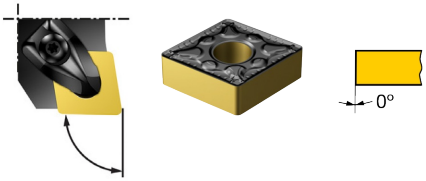
Reduces tool assembly weight, for decreased momentum, easier tool exchange and tool handling. For boring large diameters with increased stability without increased tool weight.

Dampened rough boring tool for long overhangs

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

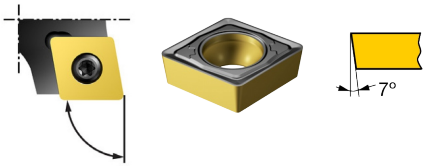
Slides for rough boring tools

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.

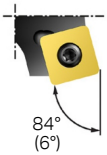
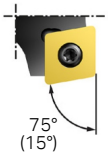
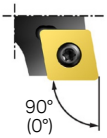
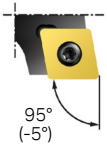
Slides with positive inserts



- 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.

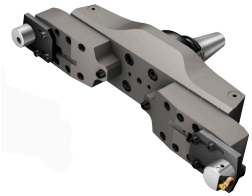
Positive inserts			Negative inserts
		For interrupted cuts, sand inclusions, stack boring etc. Through holes only.	
		First choice for general operations, step boring and for shoulder operations.	
		For high feeds or improved surface finish with Wiper inserts in stable conditions.	

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

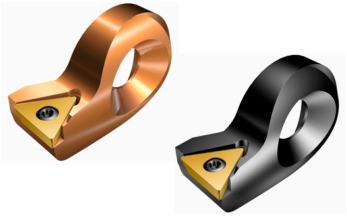
Silent Tools (dampened) are the first choice for overhangs longer than 4 times the coupling diameter.

Multi-edge reamer

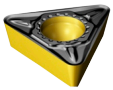
Multi-edge reamers are suitable for high feeds in mass production.

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.

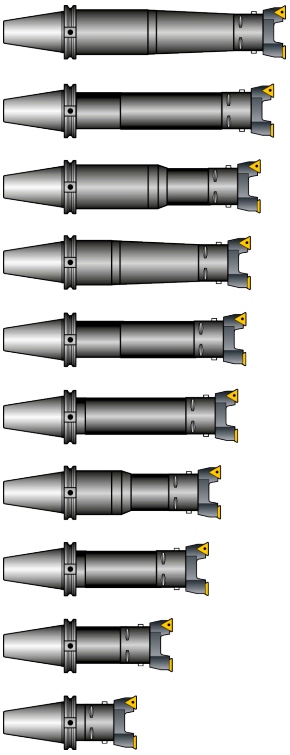
Insert shape

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.

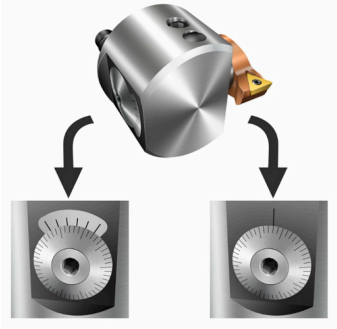
Always ensure a final adjustment is made after measurement of the hole diameter while the tool is still in the machine spindle. This will compensate for any misalignment that can happen between the machine-tool spindle and tool setting, radial deflections and insert wear.

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			

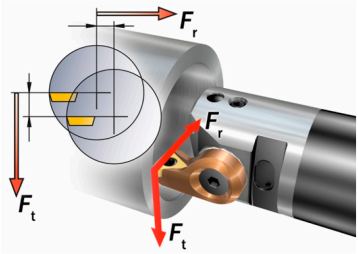
Fine boring tools

Adjustable fine boring mechanism



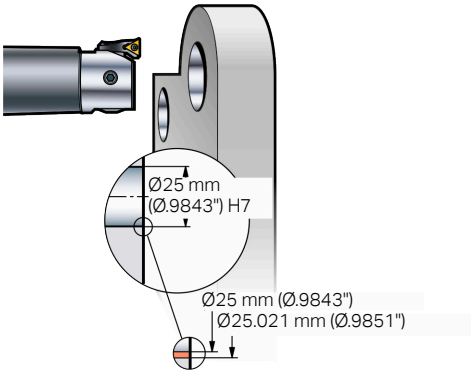
Single-edge fine boring tools have adjustment possibilities to accurately pre-set the cutting edge within microns.

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.

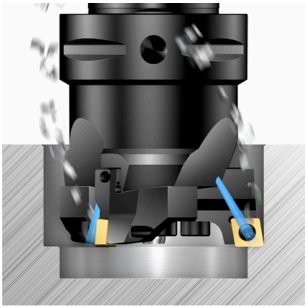
Hole tolerance



Boring tools – general

Cutting fluid supply

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

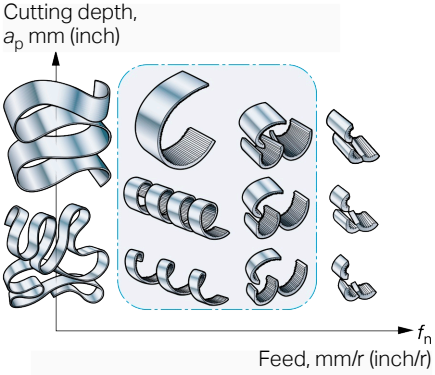


- Apply cutting fluid for optimized chip evacuation, cooling and lubrication.
- Affects hole quality and tool life.
- Internal cutting fluid is recommended in order to direct the fluid to the cutting zone.

Chip control and chip evacuation

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

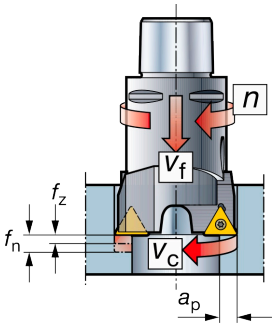
Ideally, chips should be in the form of defined commas or spirals.



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.

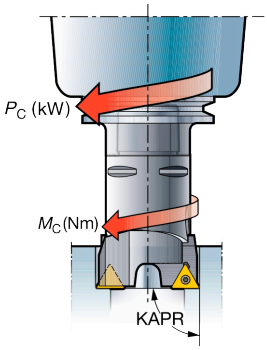
Cutting data recommendations



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.



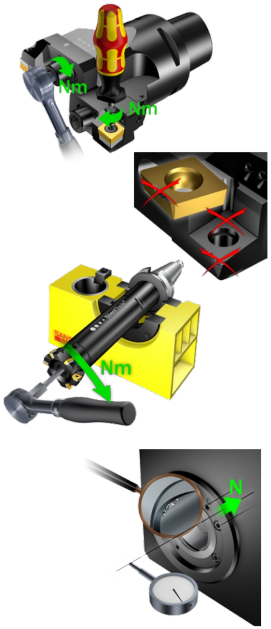
Feed and cutting depth

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

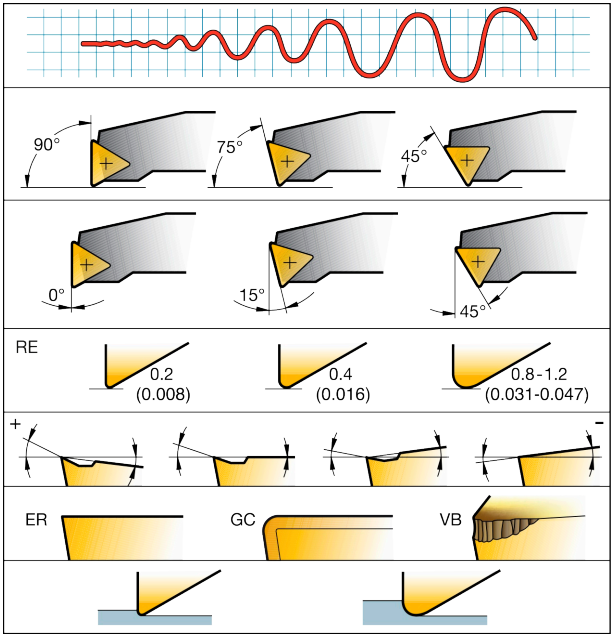
Lead angle

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.

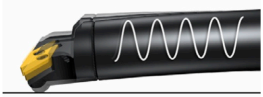
Insert wear

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

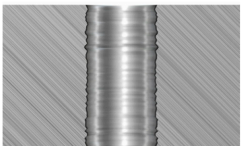
Chip breaking

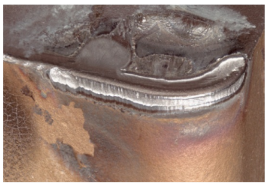
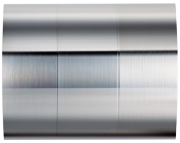
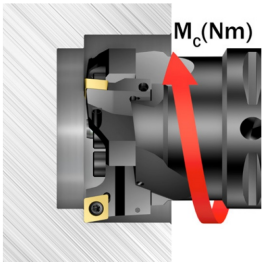
	Cause	Solution
	Too short, hard.	• Increase cutting speed. • Decrease feed. • Change geometry to a more open chip breaker.
	Too long.	• Increase feed. • Decrease cutting speed. • Change geometry to a more closed chip breaker.

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		
	Wrong cutting data.	• Change cutting edge and investigate reason for wear pattern – cutting data, insert geometry and insert grade.
Chips scratching surface		
	Bad chip breaking.	• Change cutting data. • Change insert geometry.
Surface finish		
	Bad surface finish.	• Increase speed. • Use coolant. • Use a cermet grade.
Machine power limitation		
	Limited machine power.	• Decrease cutting data. • Apply step boring. • Decrease number of inserts in cut. • Reduce depth of cut.
Power and torque consumption		
	When rough boring, make sure the machine can provide sufficient power and torque.	Important parameters are: • Feed. • Number of inserts. • Diameter. • Depth of cut.

BIG-PLUS SYSTEM licensed by
930-B40-S-20
8 Nm
BIG-PLUS SYSTEM licensed by
392.55514-40
392.55523-40
O1
O5

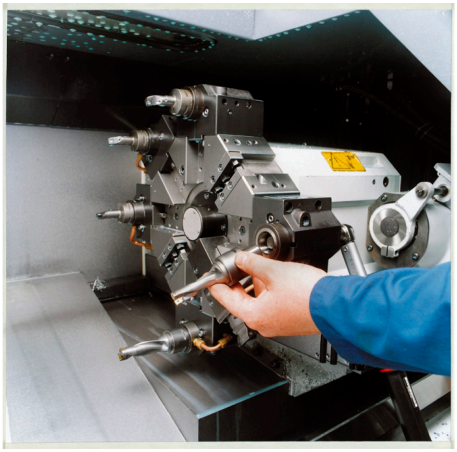
Tool holding

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.

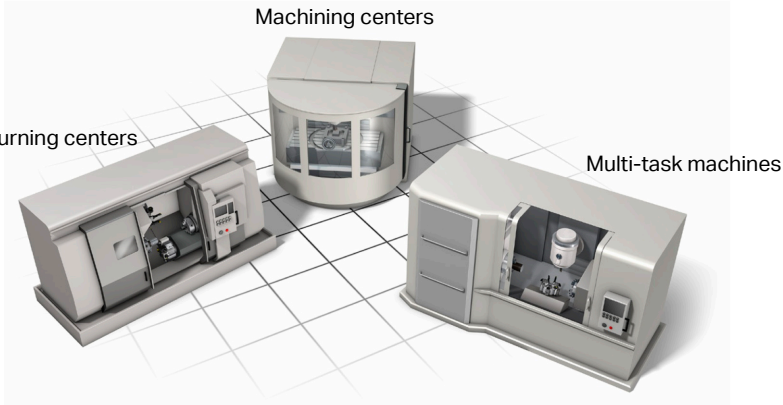
- History and background G 4
- Why modular tooling G 8
- Turning centers G 16
- Machining centers G 25
- Multi-task machines G 30
- Chucks G 35

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.

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

Modularity

- Turning center / Turning tools

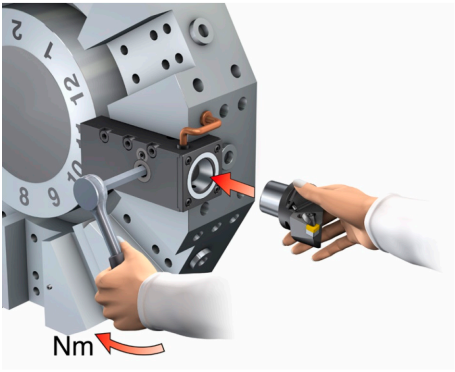
Shank holders

Block Tool System

Coromant Capto®/ Clamping units

The history of the Coromant Capto® system

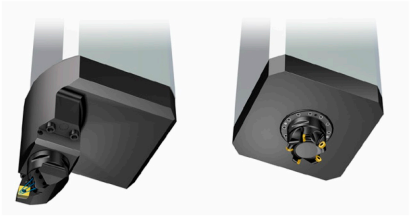
Quick change



- Turning Centers
- Vertical Lathes

Increased machine utilization

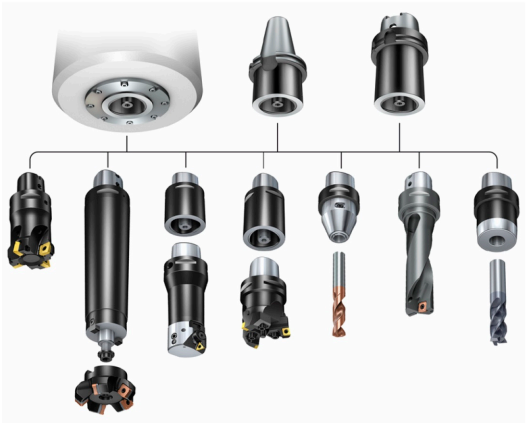
Integrated spindle



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

Increased stability and versatility

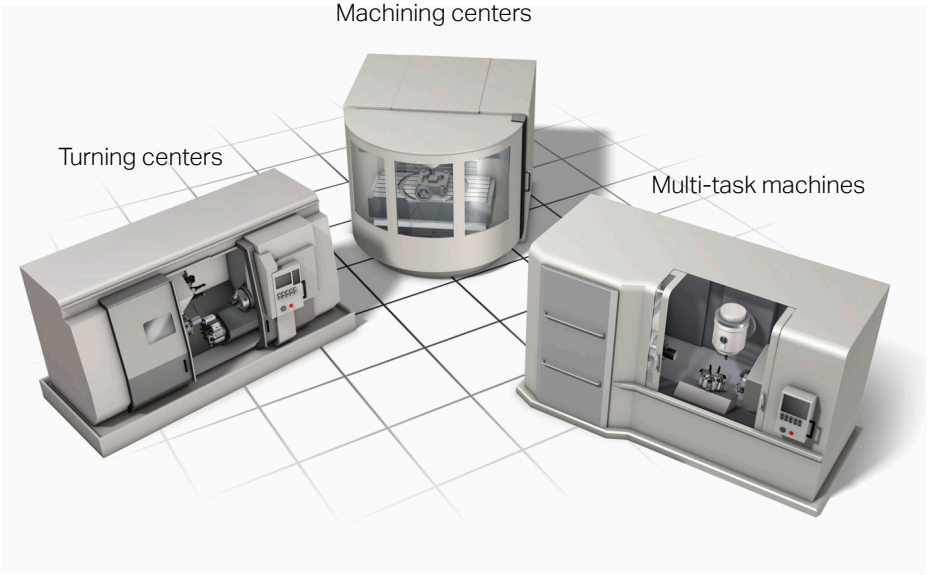
Modular systems



- Machining Centers
- Multi-Task Machines
- Vertical Lathes

Increased flexibility

A dramatic development of the machines

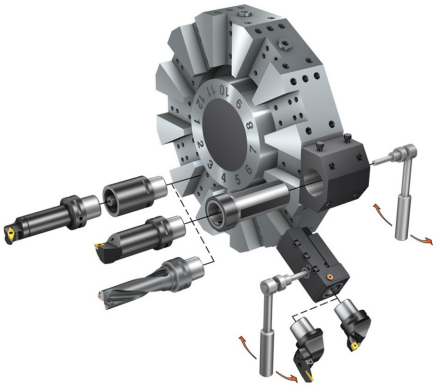


Trends

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.

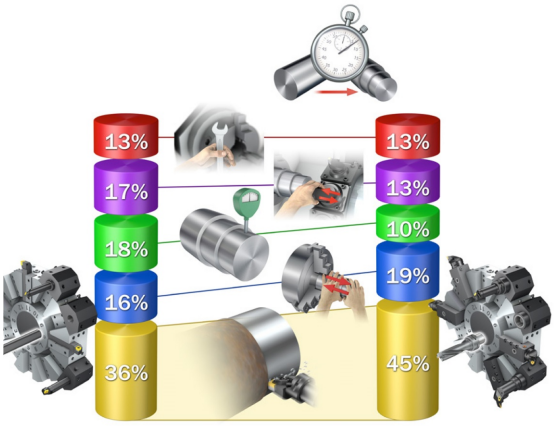
When to use quick change tooling



- 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

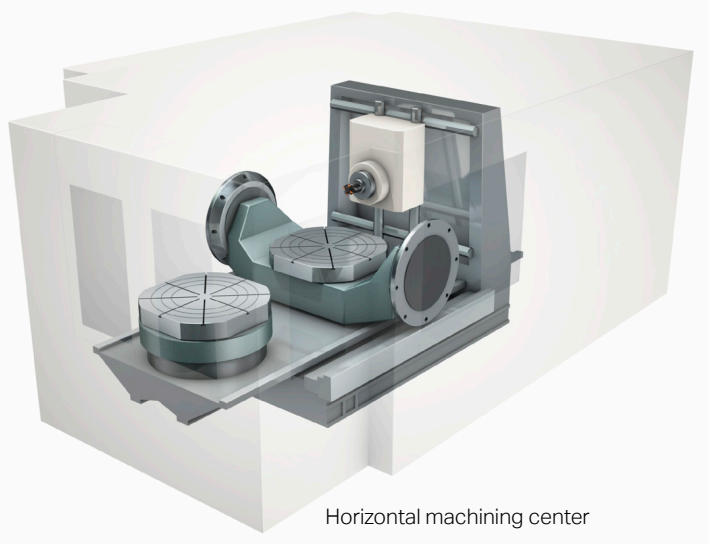
Coromant Capto® quick- change system

- 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*.

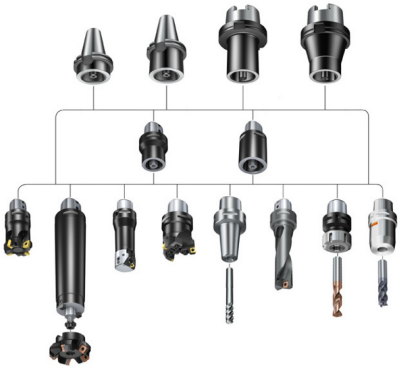
*SL is a universal modular system of adaptors with exchangeable cutting heads.

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

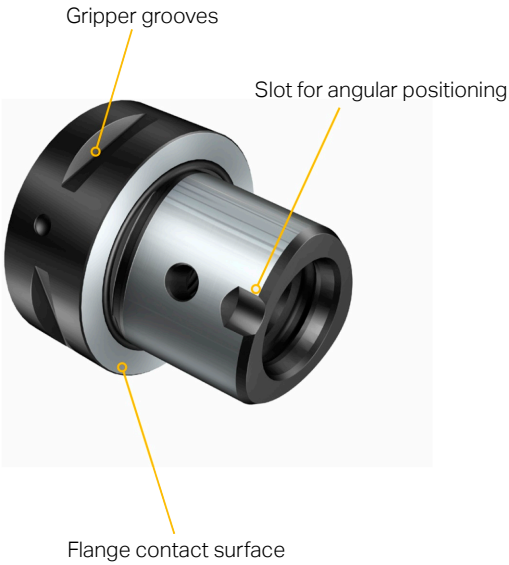
Total 64 items

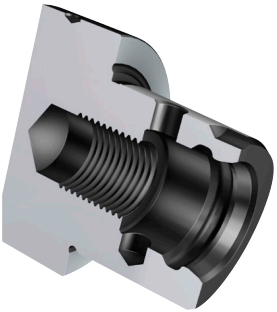
Modular tools give access to very large number of tooling solutions, with very few items.

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.

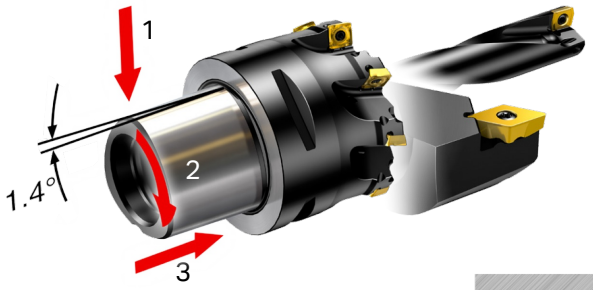
Coupling features and benefits

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

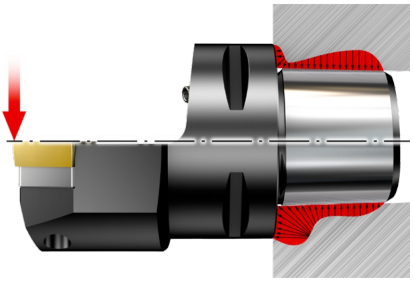
1. The radial centering is taken care of by the conical part of the polygon.
2. The low taper angle makes it possible to transmit the full force into the flange contact. The strength of the polygon coupling makes it possible to clamp with higher force than other systems. This is very important for the 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).

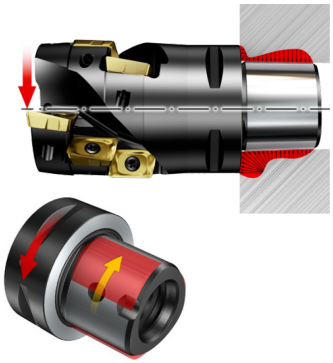
The gripper grooves are designed to give maximum bending stiffness and a higher clamping force, due to the fact that the Capto polygon has a greater surface area.



3-way locking



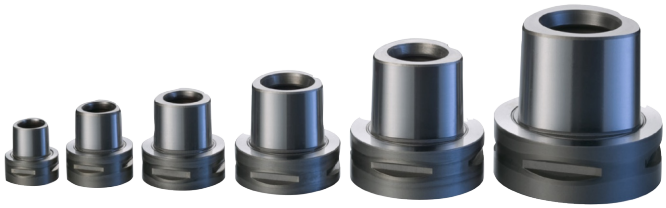
Transmission of torque



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

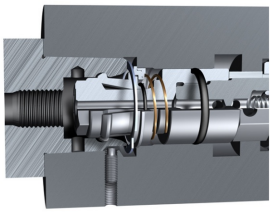


C3 = D 32 mm (1.260 inch)
C4 = D 40 mm (1.575 inch)
C5 = D 50 mm (1.969 inch)
C6 = D 63 mm (2.480 inch)
C8 = D 80 mm (3.150 inch)
C10 = D 100 mm (3.937 inch)

Different methods of clamping

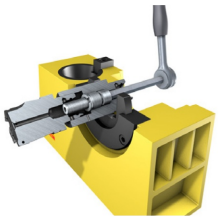
One coupling offers two methods of clamping.

Segment clamping



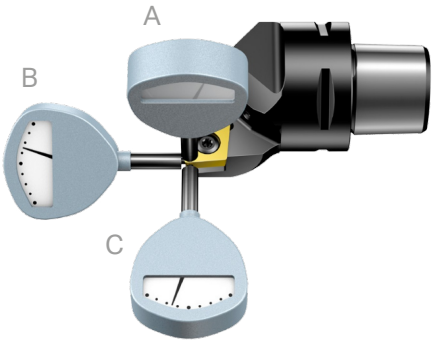
Clamping method for quick-change and automatic tool changing.

Center bolt clamping



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

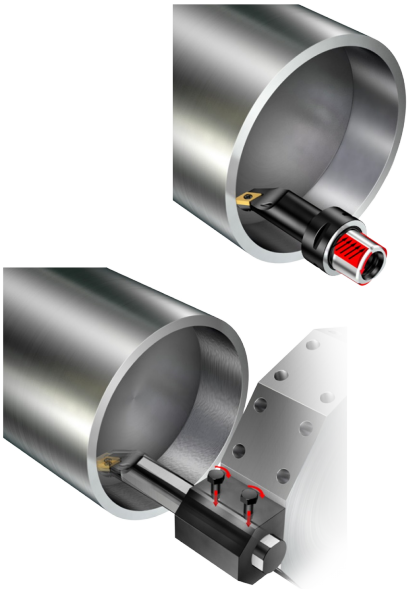
Excellent repetitive accuracy and guaranteed center height



- 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).

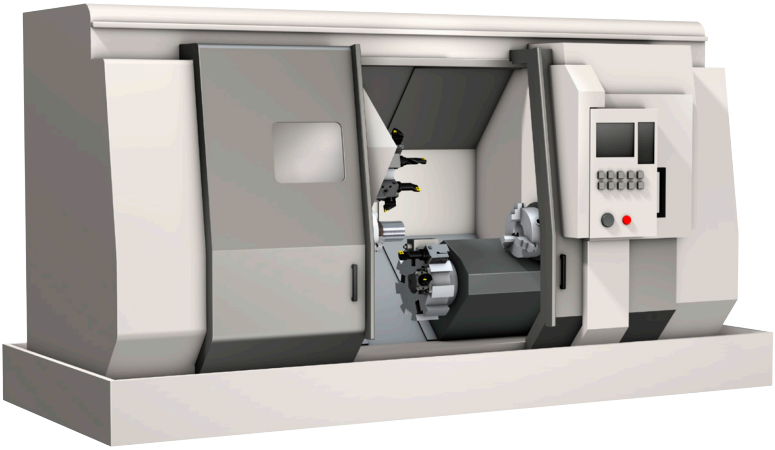
Less vibration with stable coupling

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?

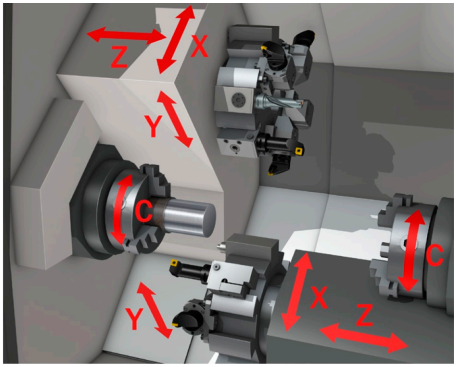
- The principle of lathes and turning centers is to cut a rotating component with a stationary cutting tool.
- The cutting tool moves parallel and perpendicular to the workpiece axis to provide the desired finished shape.
- When a cutting tool is applied to the workpiece, it can be shaped to produce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

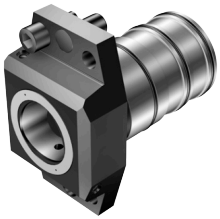
Less than 180° for clamp and unclamp

Typical clamping units for turning centers

VDI angled
Camshaft activated

Square shank
Camshaft activated

Automatic unit
Hydraulically operated



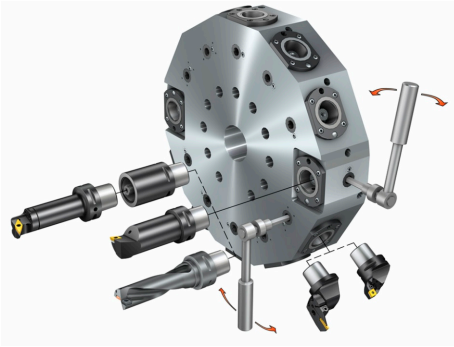
VDI straight
Camshaft activated

Round shank
Segment clamping

Special applications
Camshaft activated

Different methods how to install quick change

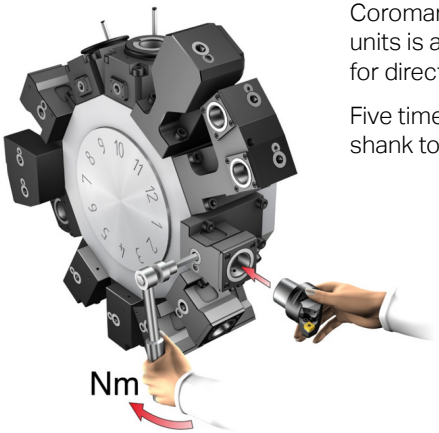
Directly integrated into the turret



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

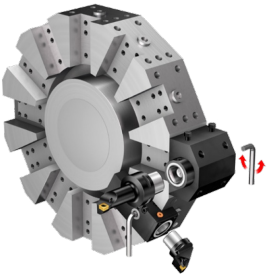


Coromant Capto® as a machine interface via clamping units is a good alternative when it's not possible to go for direct integration, (existing machines etc).

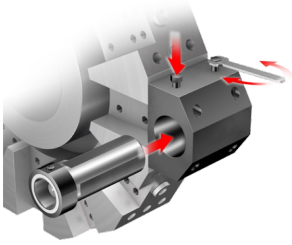
Five times faster tool change than with conventional shank tools.

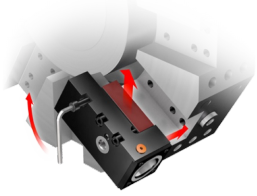
Turning lathes can easily be converted to Coromant Capto® quick change tools using standard clamping units. No modifications to the turret, and no special adaptors required.

Internal tools



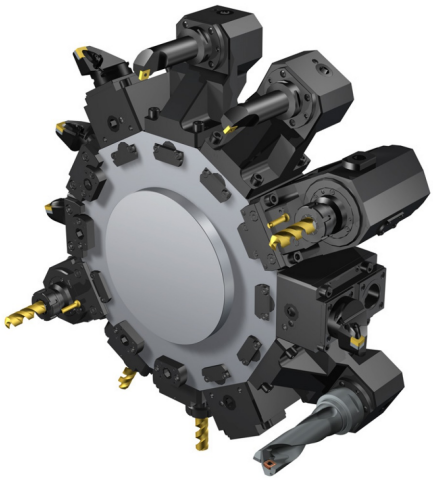
External tools





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

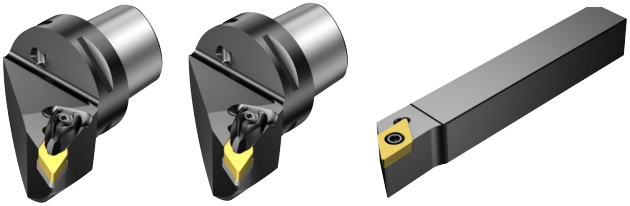
Clamping unit for external turning

Clamping unit for internal turning

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

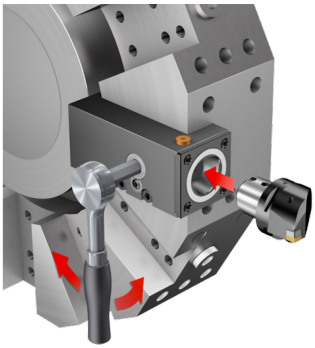
A quick change system

Insert change by using sister tools

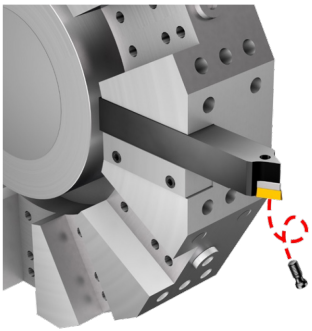


- Less downtime
- Few or no measuring cuts. Improved profitability
- No risk of losing insert screws in the chip conveyer
- Ergonomic
- Easy to clean the tip seat outside the machine.

0.5 min



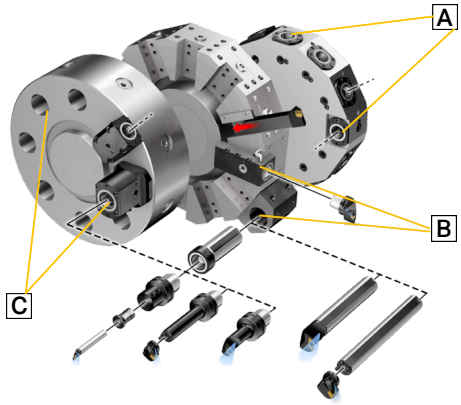
1.5 min



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

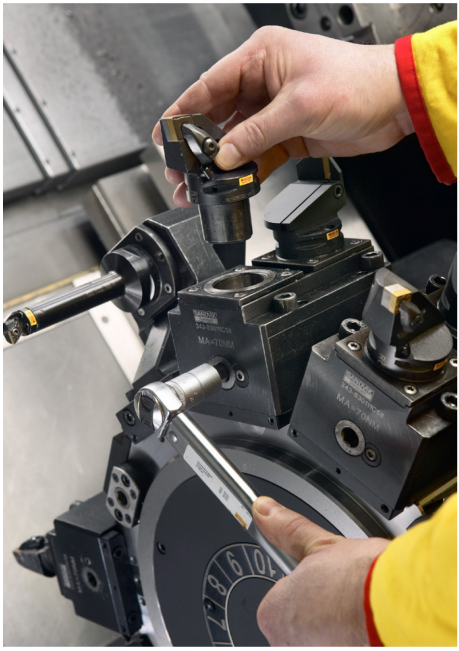
- Manual push-button tool changing
- Fully automatic tool changing possibilities.

B Shank type clamping units

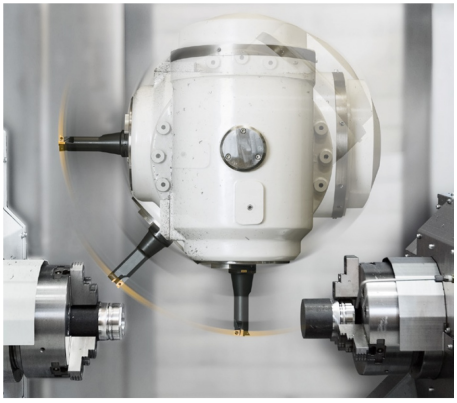
- Square and round shank tools as well as cutting units for external and internal operations.

C Clamping units for VDI turrets

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



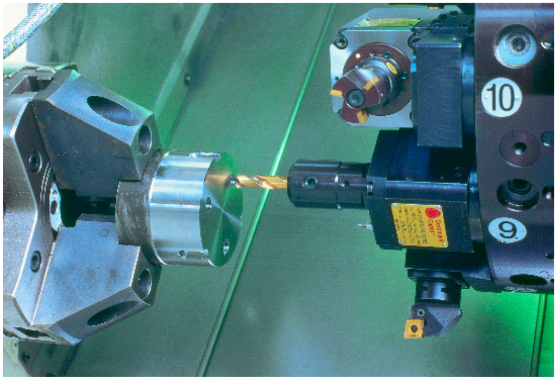
Example of installations.

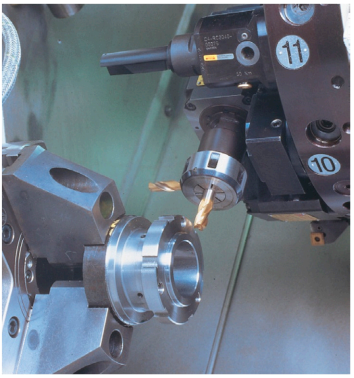


Coromant Capto® driven tool holders

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.

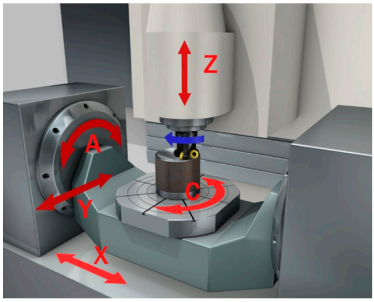
Modular tooling for machining centers

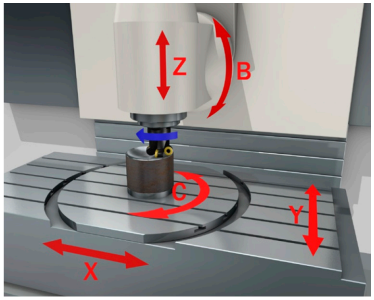
What is a machining center?

- 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).

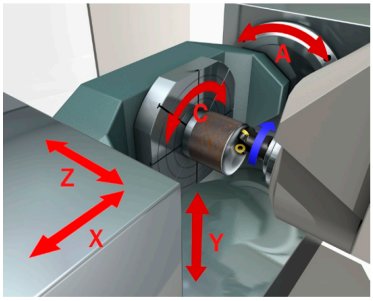
Spindle rotation and definitions of axis

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.

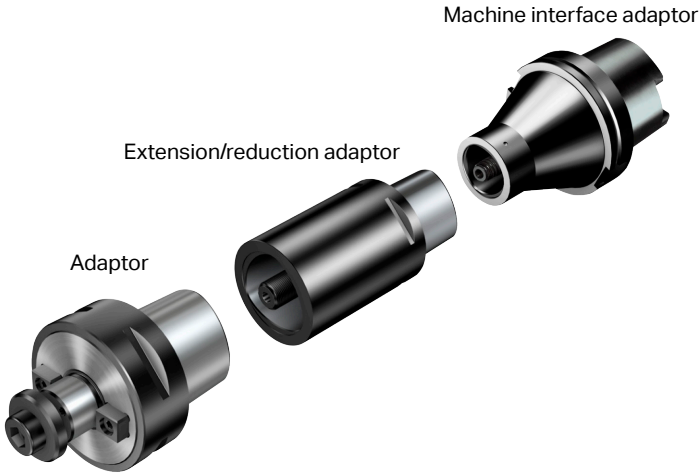
Modular tooling for machining centers

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

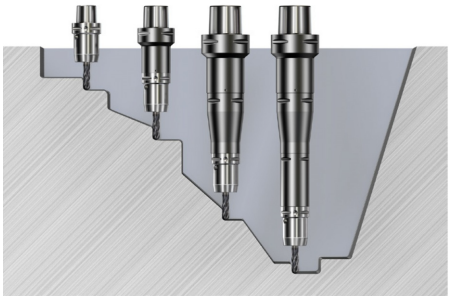
Number of items solid tools:
4 x 3 x (30 + 10) = 480 items.

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

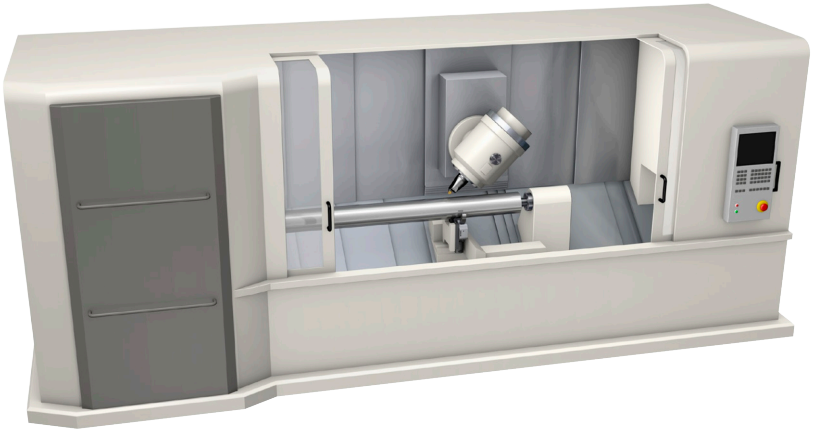
ISO BIG PLUS® 40
ISO BIG PLUS® 50

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

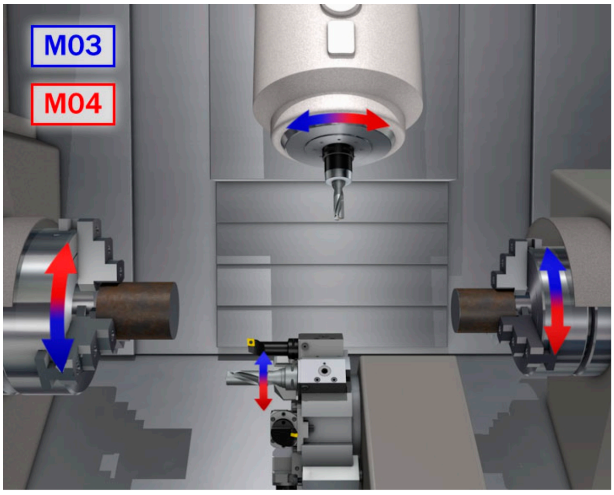


What is a multi-task machine?

- 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

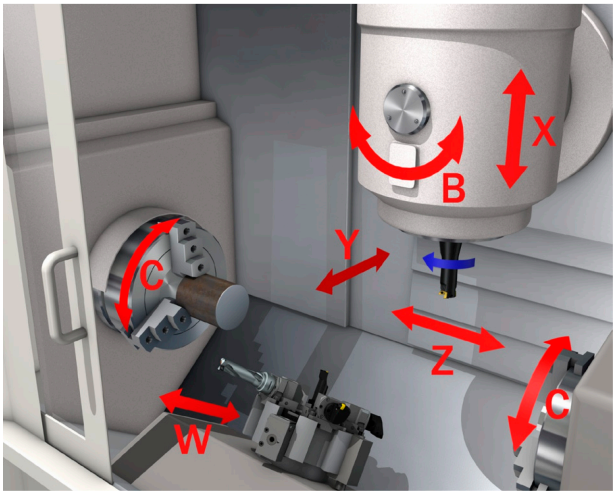


M03 = Clockwise spindle direction

M04 = Counterclockwise spindle direction

Configuration of a multi-task machine

Spindle rotation and definitions of axis

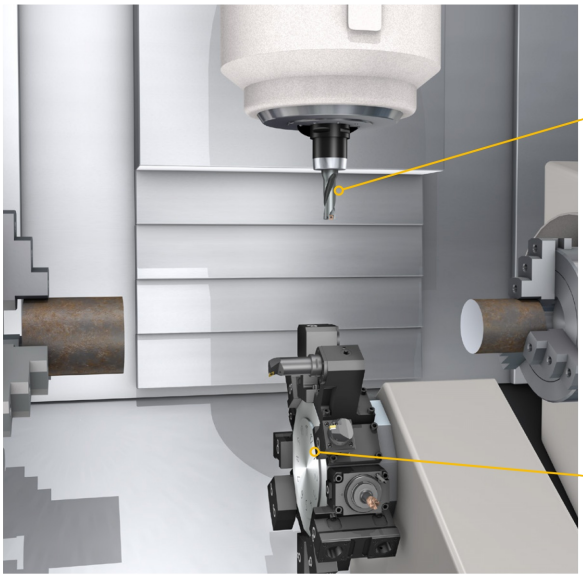


How to use modular tooling in a multi-task machine

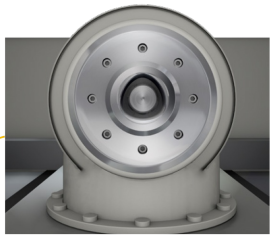
The milling spindle in a multi-task machine tool should be able to carry both rotating and non-rotating tools. Coromant Capto® is the only tooling system that can fulfill this demand without compromise.

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

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

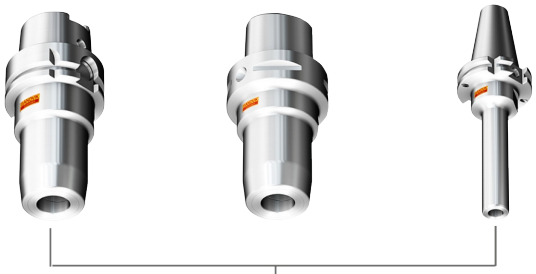
Chucks

Benefits of using hydraulic chucks

Hydraulic chuck
Heavy duty design

Hydraulic chuck
Slender design

Hydraulic chuck
Pencil design



Open sleeves

Sealed sleeves

Direct clamping

Shrink fit

Direct clamping

ER collet chuck

Open sleeves

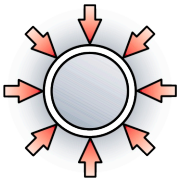
Sealed sleeves

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

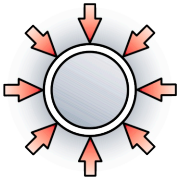
Very good Good Acceptable

Hydraulic chucks



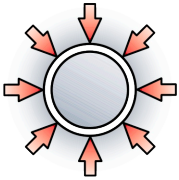
- Best pull out security on the market - clamping force repeats time after time.
- Precision run out < 4 µm (.00016") at 2.5 x DC - high precision repetition.
- Easy handling - torque wrench used for secure clamping.

Shrink fit chuck



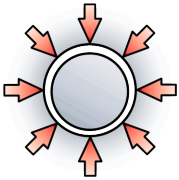
- High pull out security and high precision.
- Small nose diameter possible – good accessibility.
- Symmetrical design.

Mechanical chucks



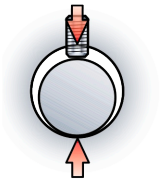
- Cylindrical sleeves can be used – good flexibility.
- Accessibility not so good because of its design (often Heavy Duty).

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

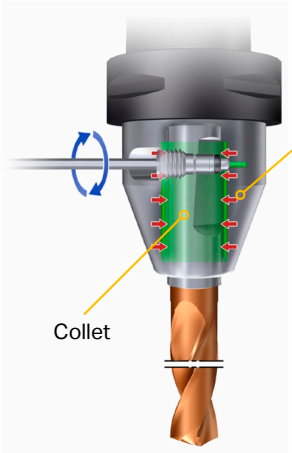


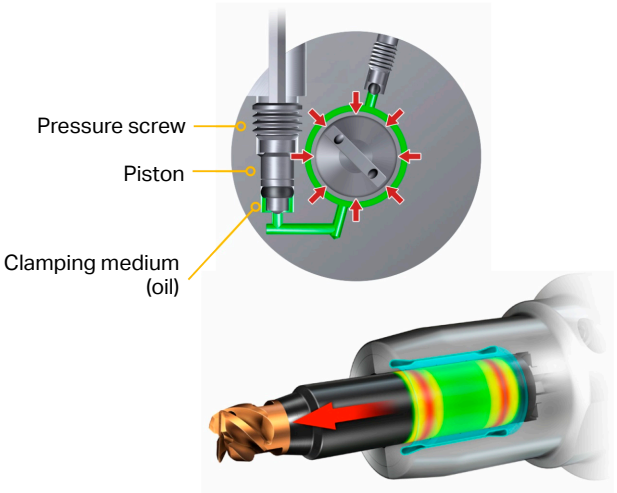
- High torque transmission.
- Low precision – low tool life and low surface finish.

Hydraulic chucks

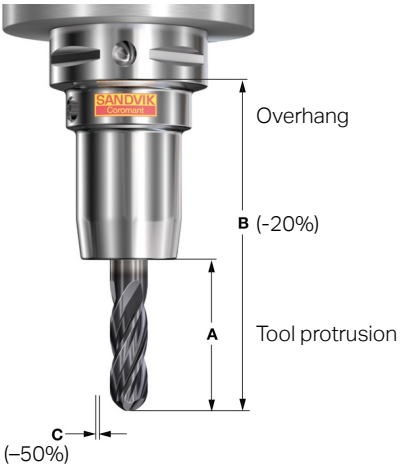
The secret behind the high precision and pull-out security

- 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

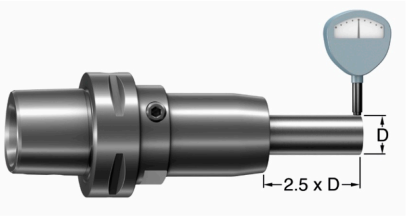
- Runout should be < 0.006 mm (< .001 inch).
- For every 0.01 mm (.0004 inch) runout - up to 50% decrease in tool life.
- More critical as tool diameter gets smaller.

Tool holding requirements

Application - Roughing and semi-finishing

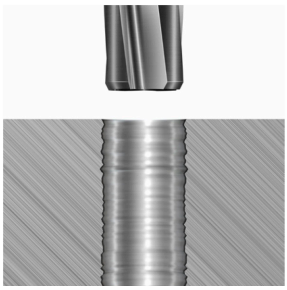
- Main criteria = clamping force
- High torque capability
- For best performance use cylindrical shanks
- Versatility of collets.

Application - Finishing



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

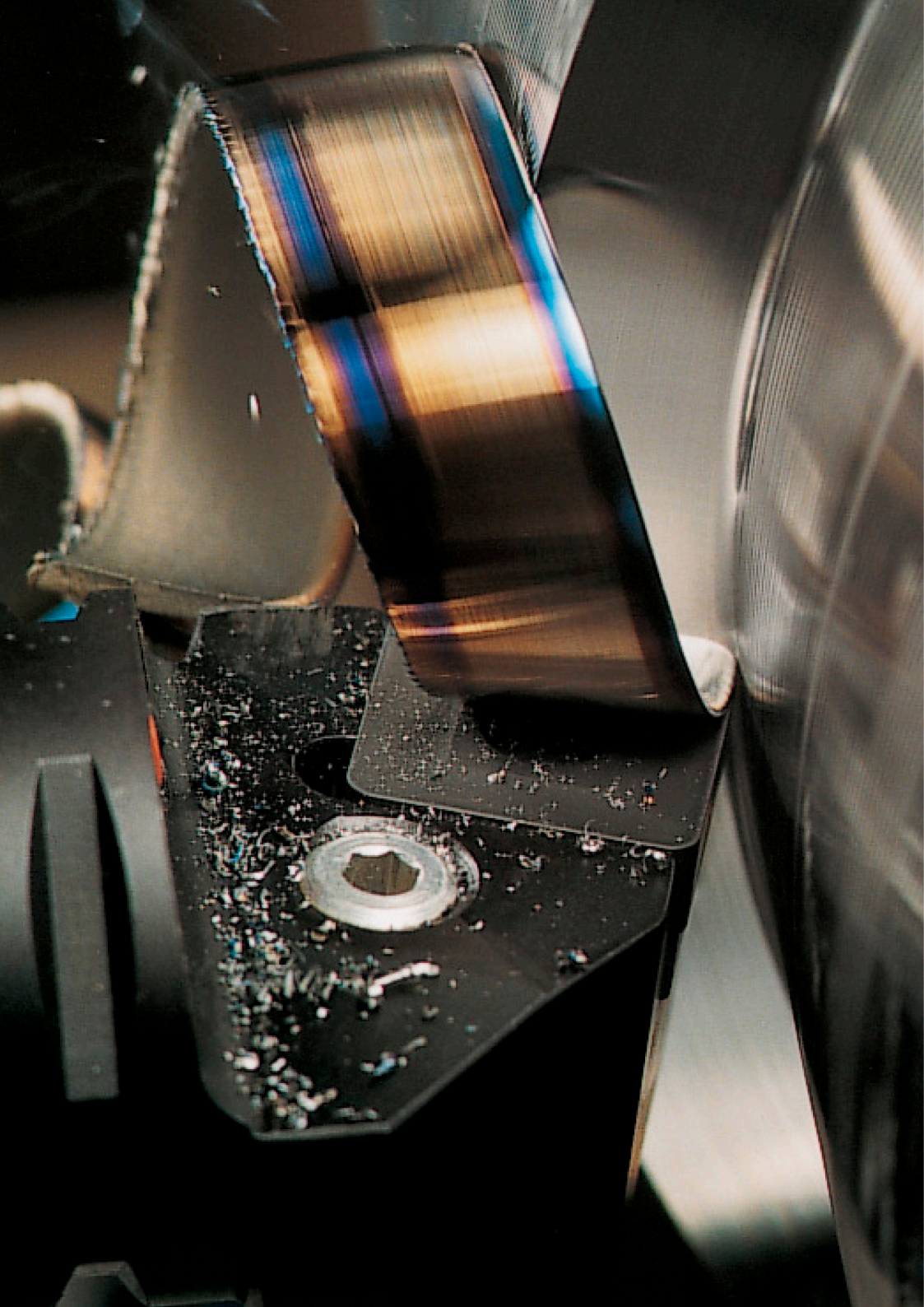
Unbalance in tool holders



Unbalance in tool holders causes:

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

05
01
BIG-PLUS SYSTEM
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.

- Workpiece materials H 4
- Manufacture of cemented carbide H 18
- The cutting edge H 29
- Cutting tool materials H 40
- Tool wear & maintenance H 52

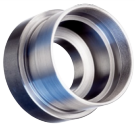
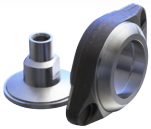
Other information

- Machining economy H 63
- ISO 13399 - The industry standard H 78
- Formulas and definitions H 81
- E-learning H 92

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.

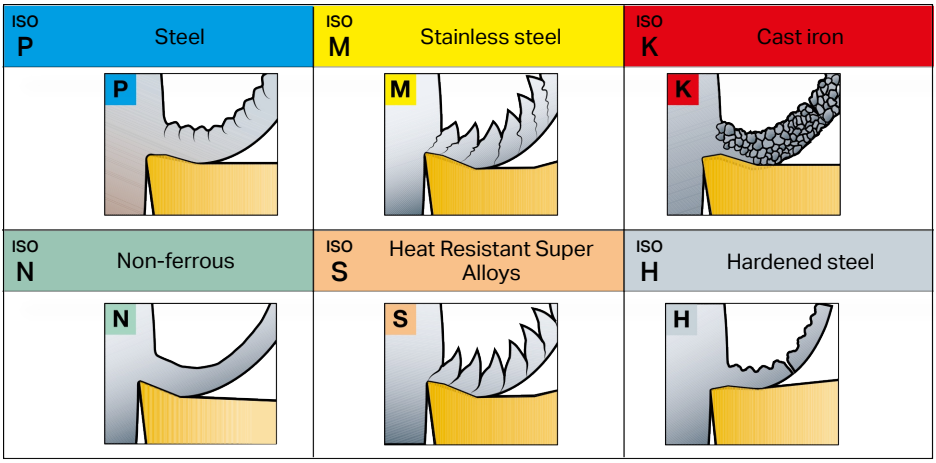
K The K-area is dominated by automotive components, the machine builders and the iron works production.

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.



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.

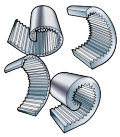
N Low power needed per mm^3 ($inch^3$), but due to the high metal removal rate, it is still a good idea to calculate the maximum power required.

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

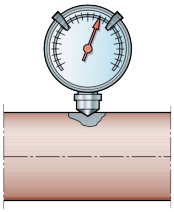
N Non-ferrous

S Heat resistant alloys

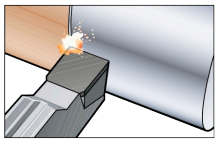
H Hardened steel

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

Hardness

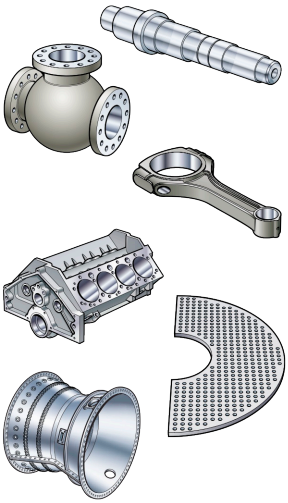


Hardness Brinell

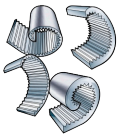


Usually there is a relation between material hardness and tool life, as well as machining data and type of geometry and grade. The higher the hardness, the shorter the tool life, with more rapid wear on the cutting edge.

Component

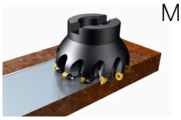


Depending on the type of material, set-up and way of machining, different choice of tooling is required to perform different applications turning, milling, drilling etc.

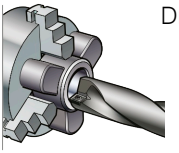


Application

Turning



Milling



Drilling

H R Roughing/ Heavy

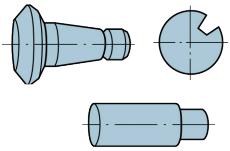
M M Medium

L F Finishing/ Light

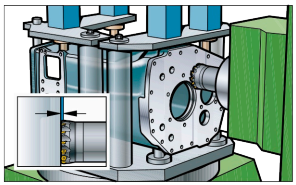
There are three major types of application, all requiring different tools, inserts and grades. These also depend on the load on the cutting edge, from finishing to roughing.

Condition

Cutting conditions



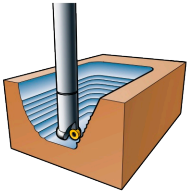
Clamping conditions



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

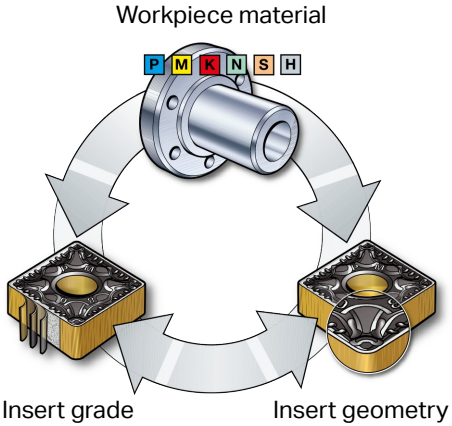
Coolant



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.

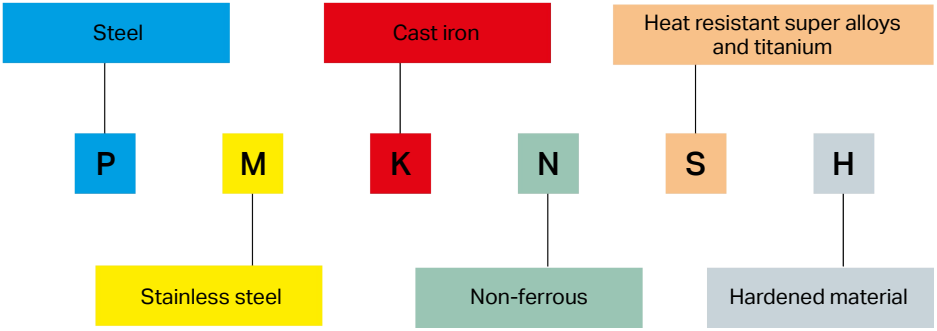
The interaction between workpiece material, geometry and grade



- 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

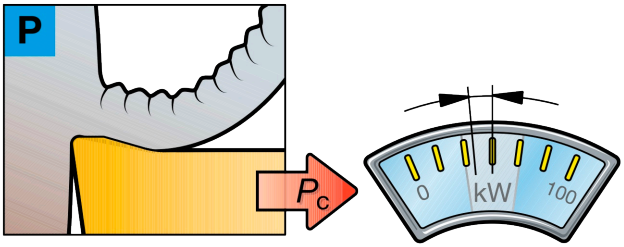
3 = material subgroup: non-ferrous with Si content 1-13%

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/mm² (217,500–449,500 lbs/inch²).
- Cutting force, and the power needed to machine ISO P materials, stays within a limited range.

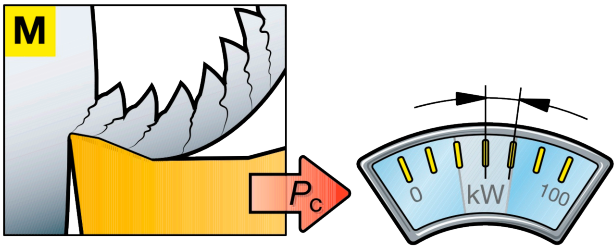
What is steel?

- 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.

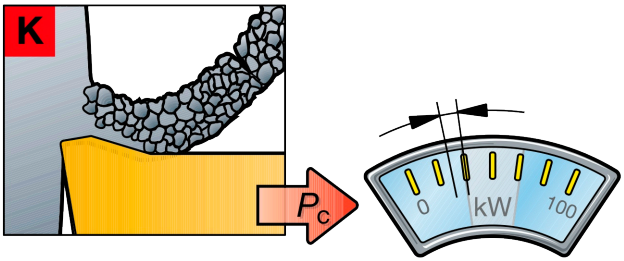
What is stainless steel?

- 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.

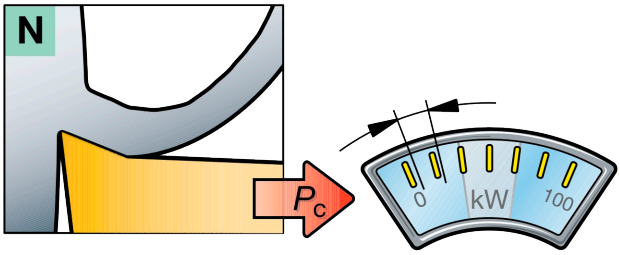
What is cast iron?

- 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.

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

See product catalogs for details on MC codes.

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/mm^2 (50,750–101,500 lbs/inch^2).
- Cutting force, and the power needed to machine ISO N materials, stays within a limited range.

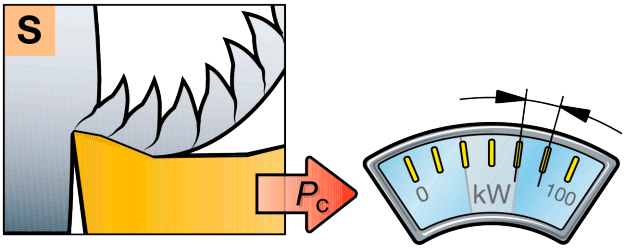
What is Non-ferrous material?

- 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).

See product catalogs for details on MC codes.

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

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

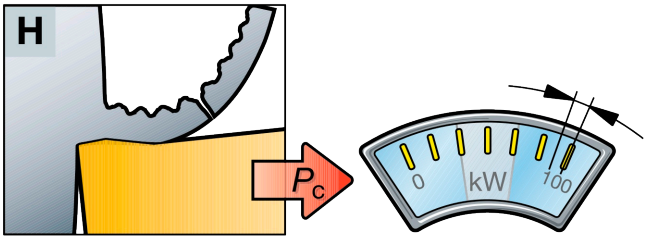
Condition: Annealed, Solution heat treated, Aged rolled, Forged, cast.

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

See product catalogs for details on MC codes.

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

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?

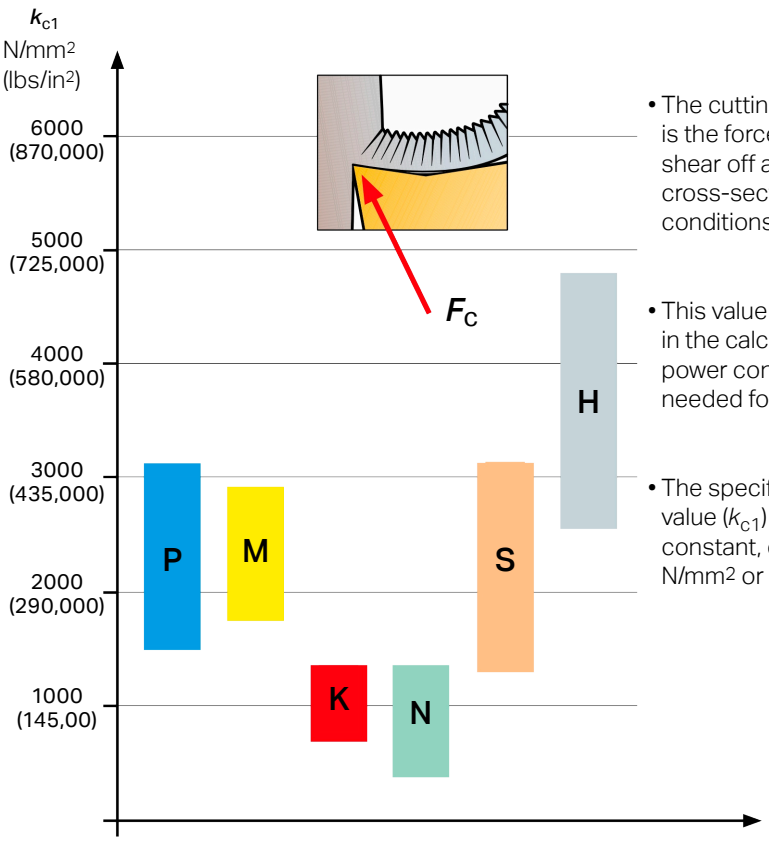
- Hardened steel is the smallest group from a machining point of view.
- This group contains hardened and tempered steels with hardness >45–65 HRC.
- Typically, however, hard part turned components can be found to be within the range of 55–68 HRC.

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

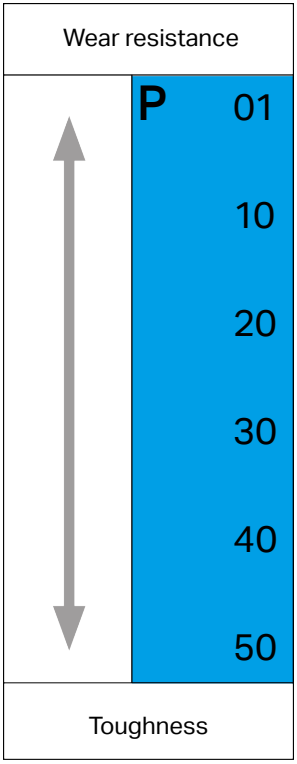
See formulas section on specific calculations.

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.

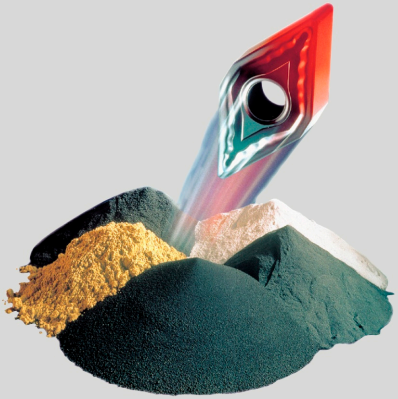
P10: Turning; copying; threading; milling; high cutting speed; small to medium chip area.

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.

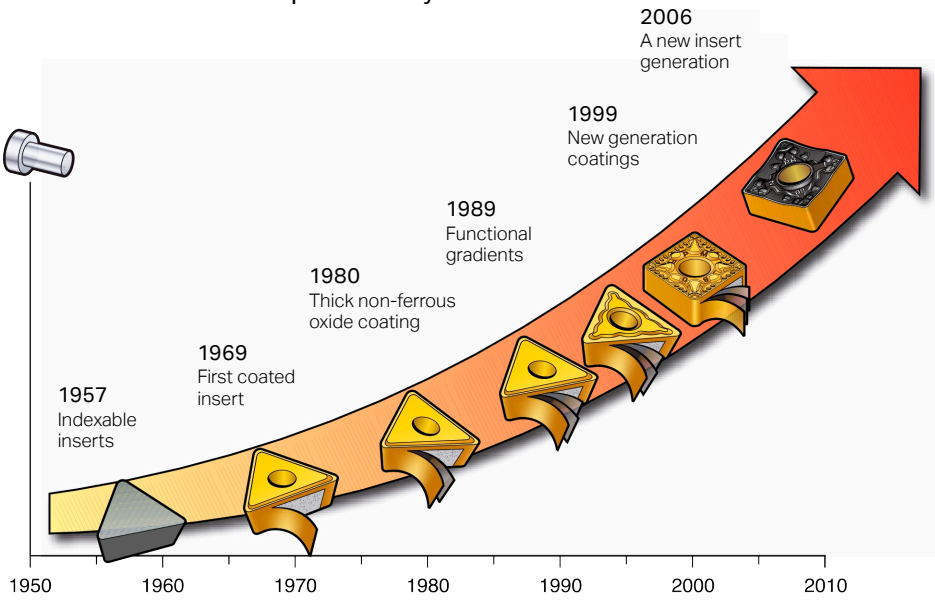
The development of cutting tool material

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



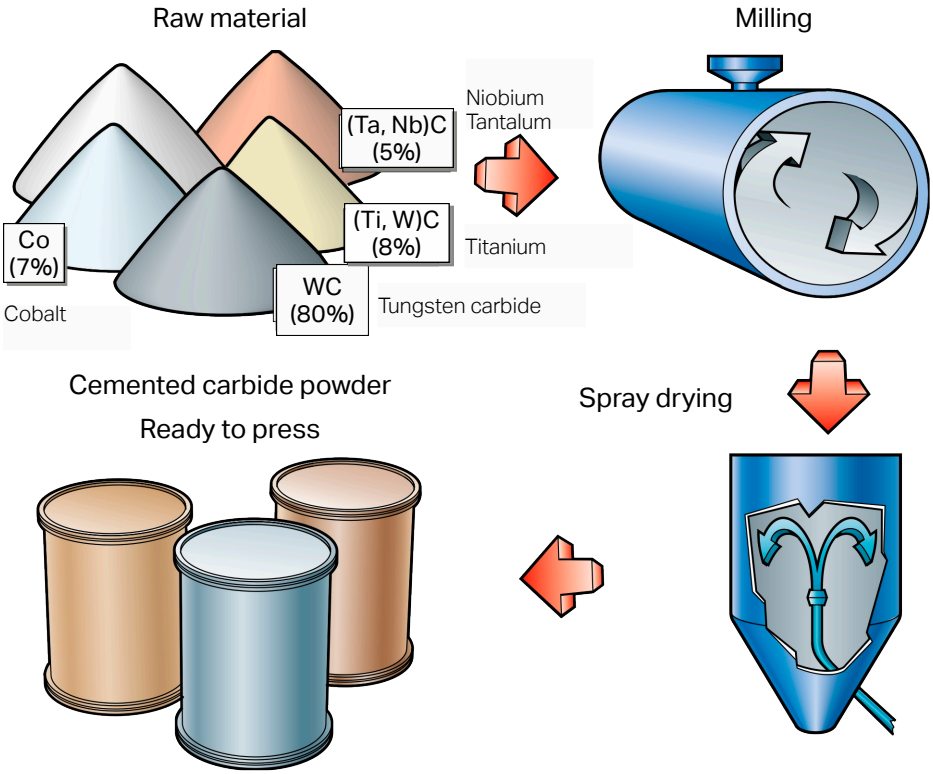
Powder production

There are two main elements of a cemented carbide insert:

- Tungsten Carbide (WC)
- Cobalt (Co)

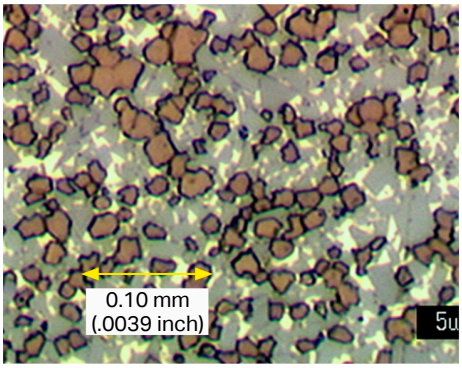
Other commonly used elements are Titanium, Tantalum and Niobium Carbides. Designing different types of powder and different percentages of the elements is what makes up the different grades.

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



Tungsten powder

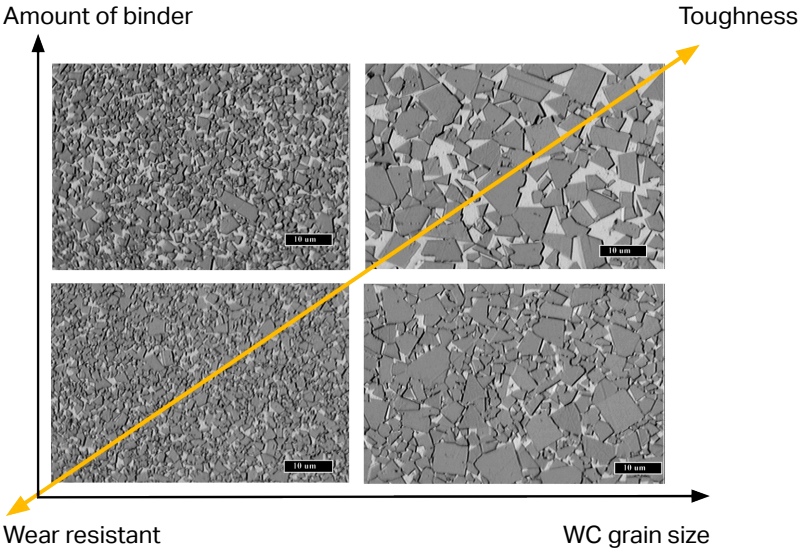
The size of the tungsten carbide grains



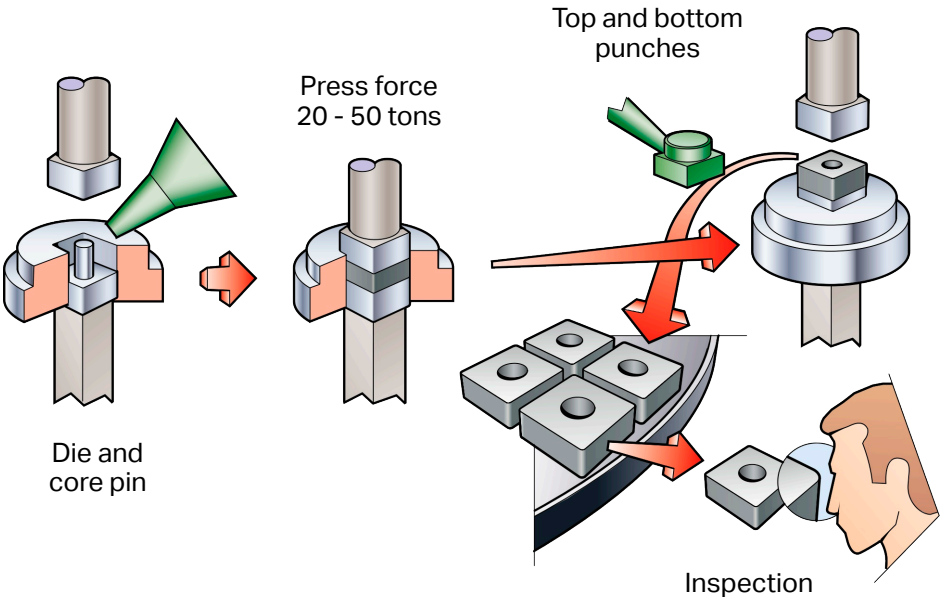
The main raw material for the manufacture of cemented carbide is tungsten-ore concentrate. Tungsten powder is produced from tungstic oxide derived chemically from the raw material. By varying the conditions of reduction, tungsten powder of various grain size can be manufactured. The carbide granules 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.



Pressing powder compacts



The pressing operation consists of several pieces of tooling:

- Top and bottom punches
- Core pin
- Cavity.

The pressing procedure:

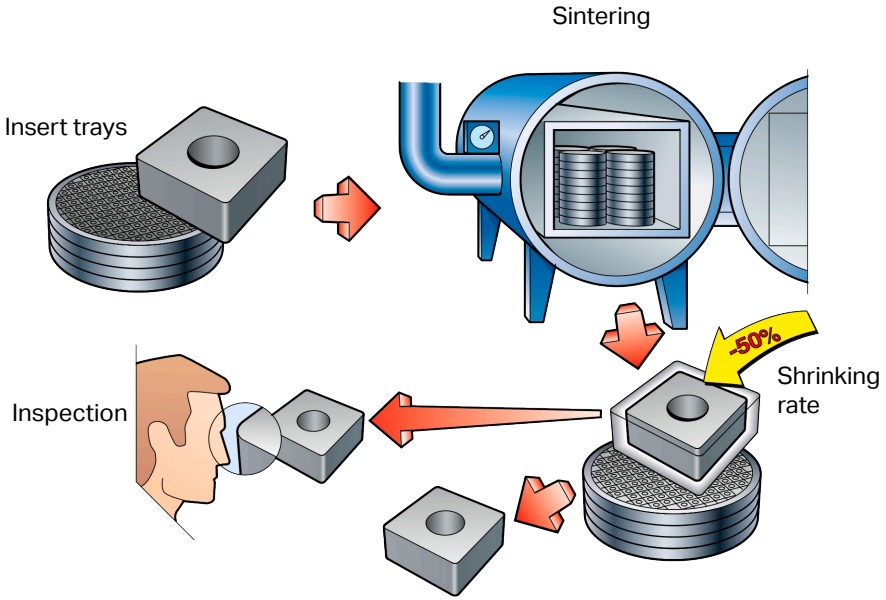
- Powder is poured into the cavity
- Top and bottom punches come together (20-50 tons)
- The insert is picked and placed via robot onto a graphite tray
- Random SPC is performed, to check for weight.

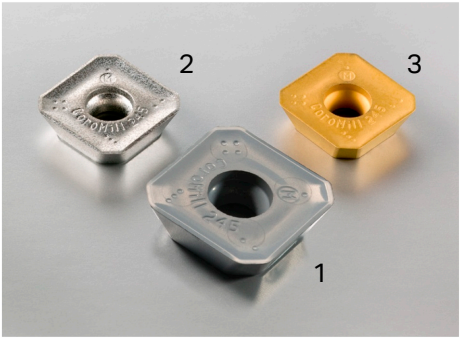
The insert is 50% porous at this stage.

Sintering the pressed inserts

Sintering consists of the following:

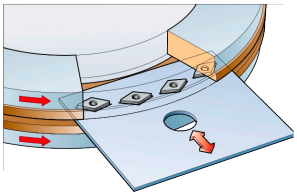
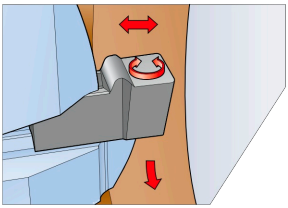
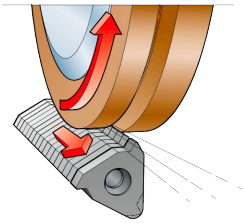
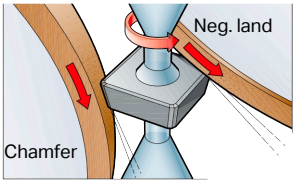
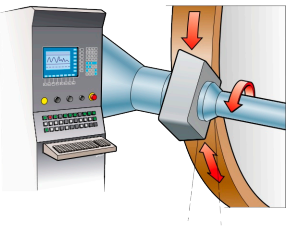
- Loading trays of inserts into a sintering furnace.
- The temperature is raised to ~1400° C (~2550° F) .
- This process melts the cobalt and the cobalt acts as a binder.
- The insert will shrink 18% in all directions during the sintering; this corresponds to about 50% in volume.





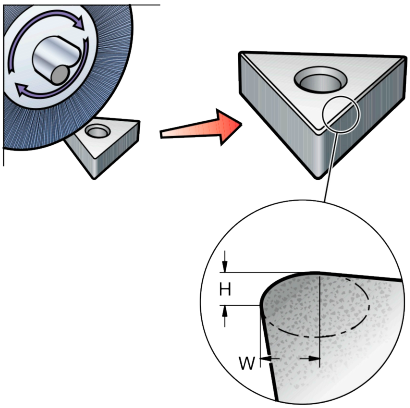
1. Unsintered insert
2. Sintered insert
3. Coated insert

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.

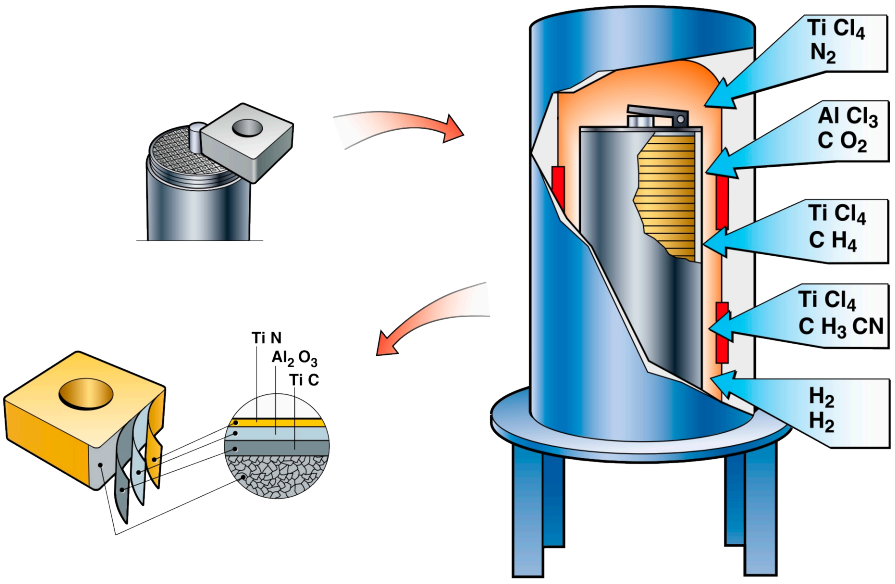


- ER-treatment (Edge Roundness) is done before coating.
- The relation between W/H depends on the application.

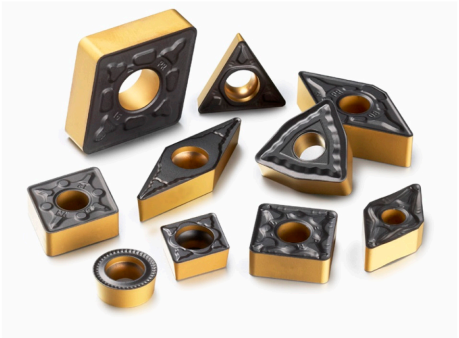
Generally the ER corresponds to the thickness of a hair, diameter: ~80 µm (~.0031 inch).

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

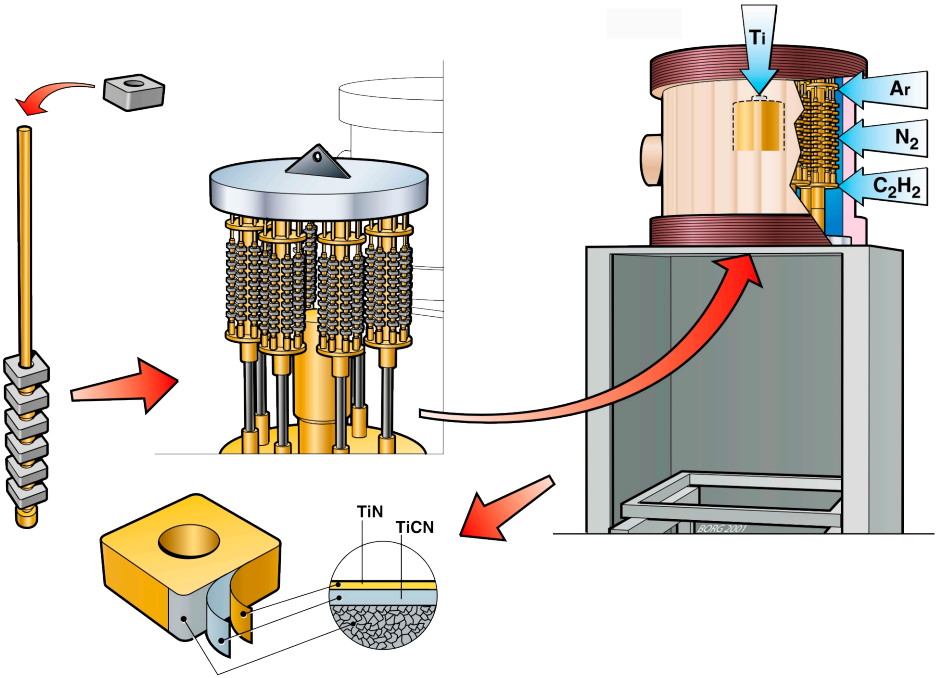


- The ability to making thick coatings.
- Ability to make even coating thickness.
- Very good adherence to the carbide substrate.
- Very good wear resistance.
- Possibility to make oxide coatings.

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.

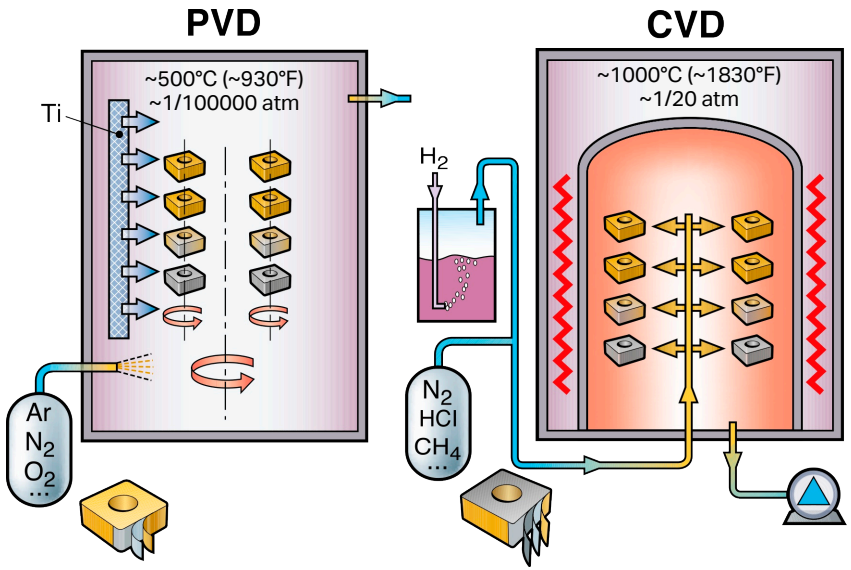
PVD vs. CVD coating process

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).

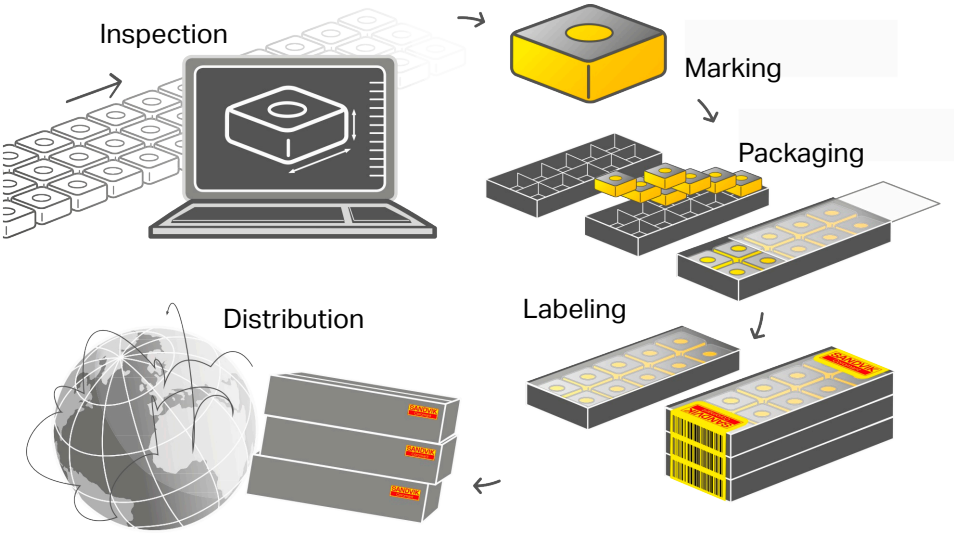


- Thinner coating
- Sharper edges
- Tougher

- 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.



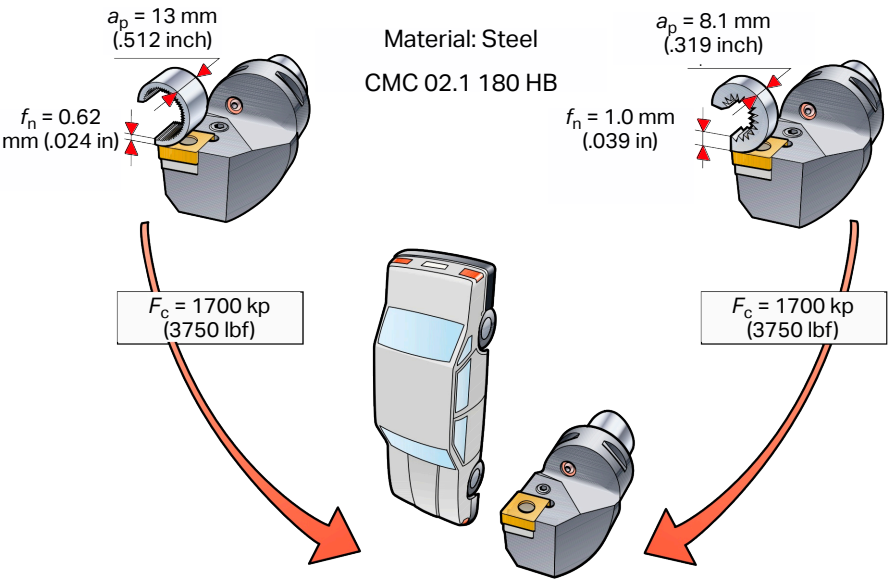
The cutting edge

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.



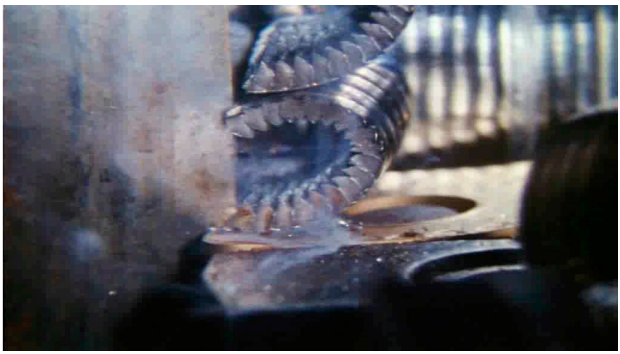
The cutting force in these two cases is equivalent to the weight of a passenger car.

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

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

The machining starts at the cutting edge

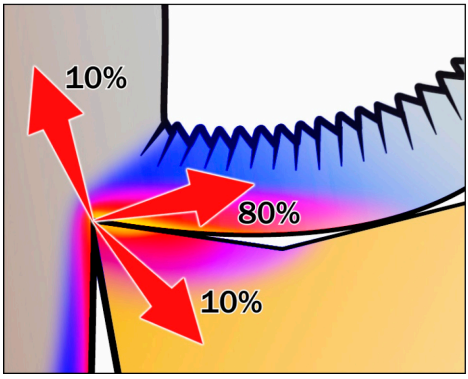


Typical chip breaking sequences with high speed imaging.

Cutting zone temperatures

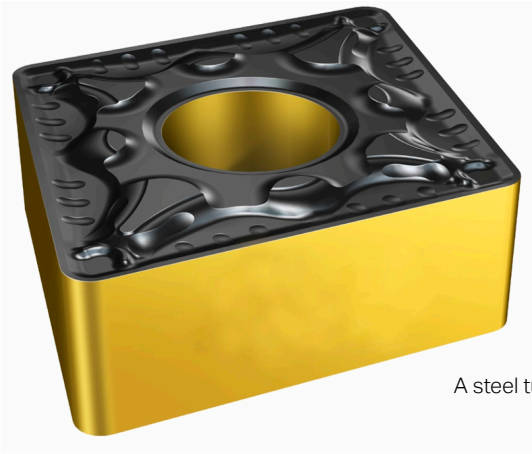
The maximum heat generated during cutting is on the top part of the insert, 1000° celsius (1832° fahrenheit), in the chip breaker, and close to the cutting edge.

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



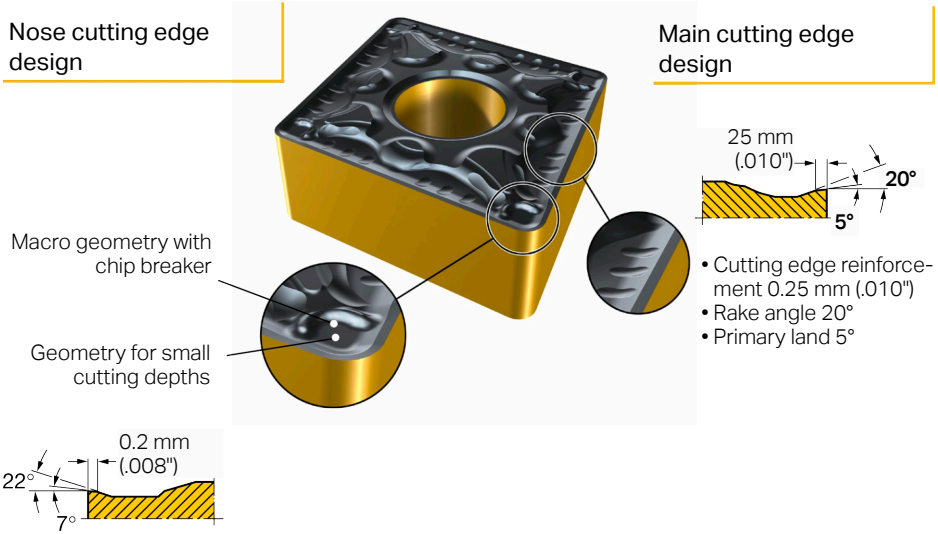
- The rake angle, geometry and feed play an important role in the chip formation process.
- Removing heat from the cutting zone through the chip (80%) is a key factor.
- The rest of the heat is usually evenly distributed between the workpiece and the tool.

The design of a modern insert



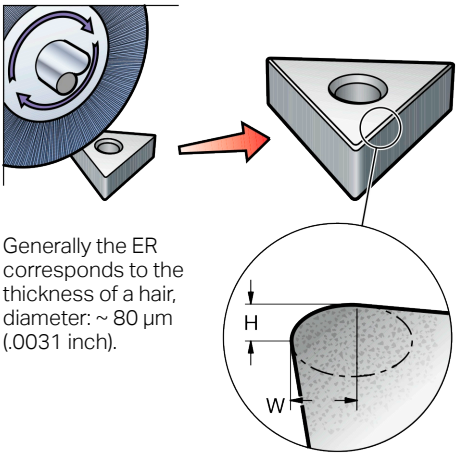
A steel turning insert for medium turning.

Definitions of terms and geometry design



The reinforcement of the cutting edge

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

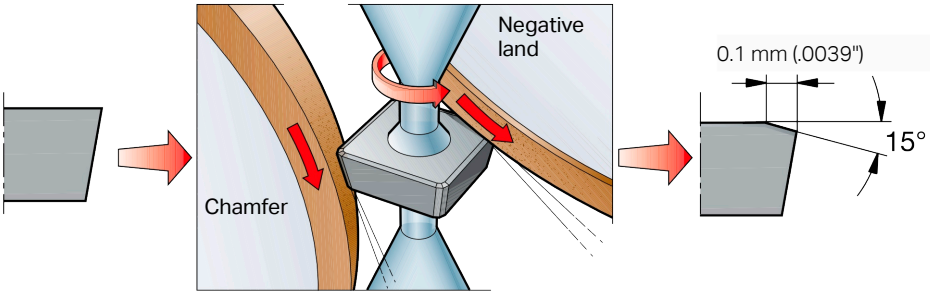


Generally the ER corresponds to the thickness of a hair, diameter: ~ 80 µm (.0031 inch).

- 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.

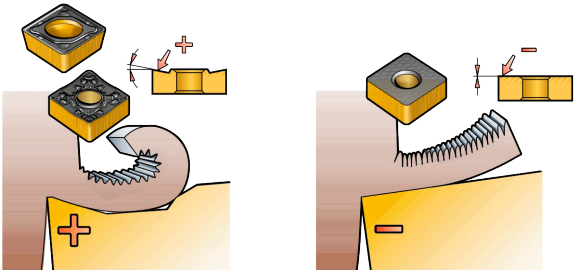


- A negative land increases the strength of the cutting edge, but also creates higher cutting forces.

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

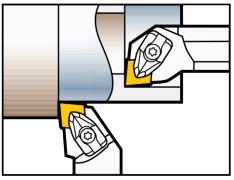
K

N

S

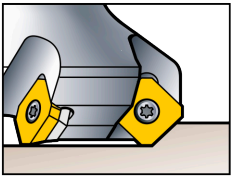
H

Turning



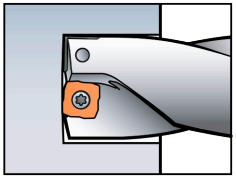
- Needs a durable edge that can perform for a long time, and often in continuous cuts at high temperature.
- Good chip breaking ability.
- Good resistance against different types of wear and against plastic deformation.

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

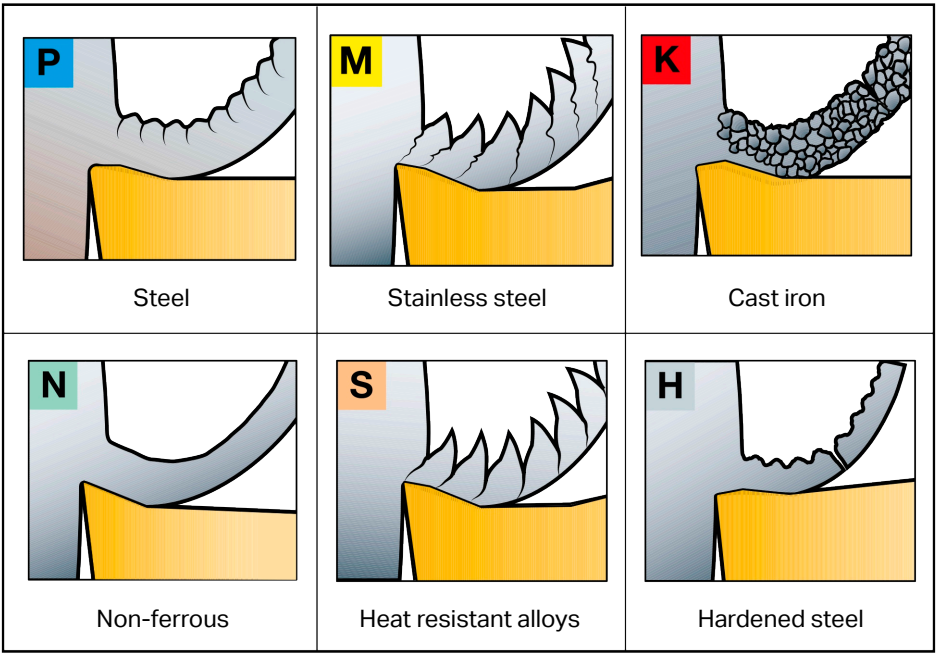


- The edge must be strong enough to last at very low cutting speeds; in fact, at zero speed in the center of the drill.
- Coolant is present, mainly for chip transportation reasons, which puts the edge under extra stress from temperature variations.
- To transport the chips from the narrow chip flutes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

Six main groups of workpiece materials

Different characteristics for removing chips

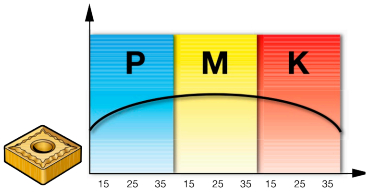
Good chip forming usually results in high cutting forces and excess heat, depending on the material. This can lead to low cutting speeds with adhesive stresses as a result. On the other hand, materials like non-ferrous, unalloyed steels and low-strength cast iron produce less cutting force.



From universal to optimized turning inserts

General inserts

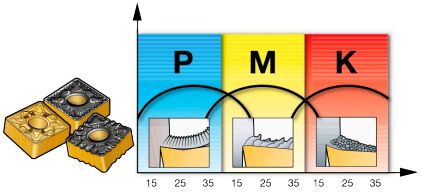
- General geometry
- Optimizing with grades
- Performance compromised



Application area

Dedicated inserts

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



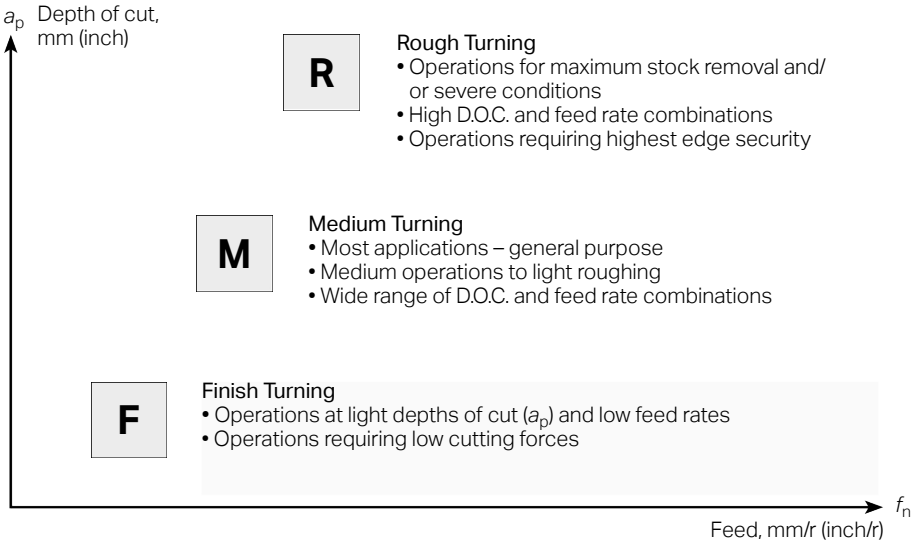
Application area

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

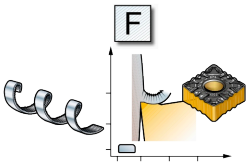
The different micro- and macro-geometries are adapted to the various requirements in the applications and materials.

Workpiece material	Finishing	Medium	Roughing
P	17° 0.7 mm (.028") 4°	22° 0.2 mm (.008") 7°	22° 0.32 mm (.013") 3°
M	15°	22° 0.29 mm (.011") 12°	22° 0.32 mm (.013") 8°
K	6° 0.1 mm (.004") 15°	2° 0.25 mm (.010") 15°	0°
S	15°	25° 15°	10°

Type of application - Turning

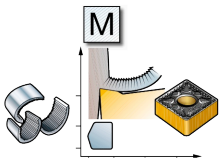


Selecting the insert geometry in turning



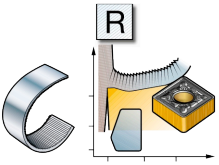
Finishing (F)

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



Medium (M)

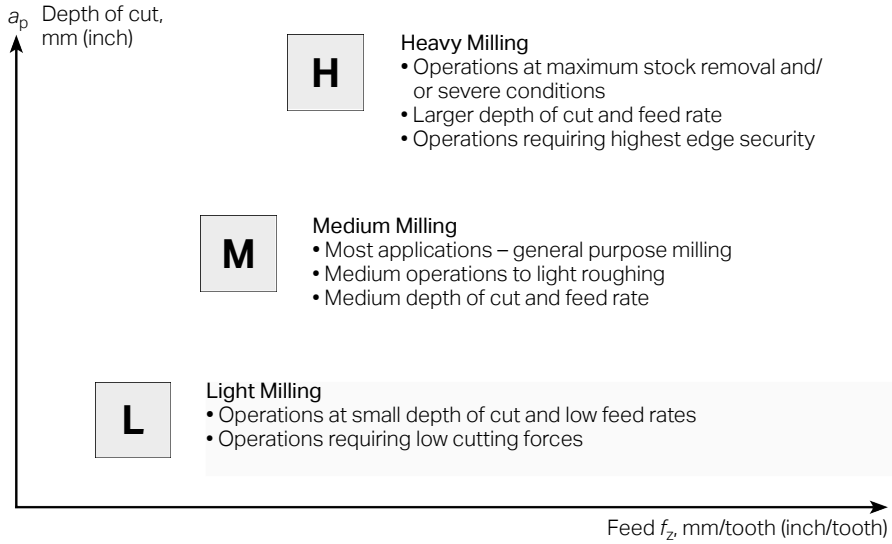
- General purpose geometry
- Medium feed rates
- Medium operations to light roughing.



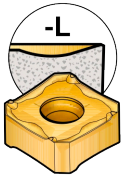
Roughing (R)

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

Type of application - Milling

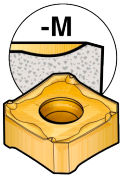


Selecting the insert geometry in milling



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.

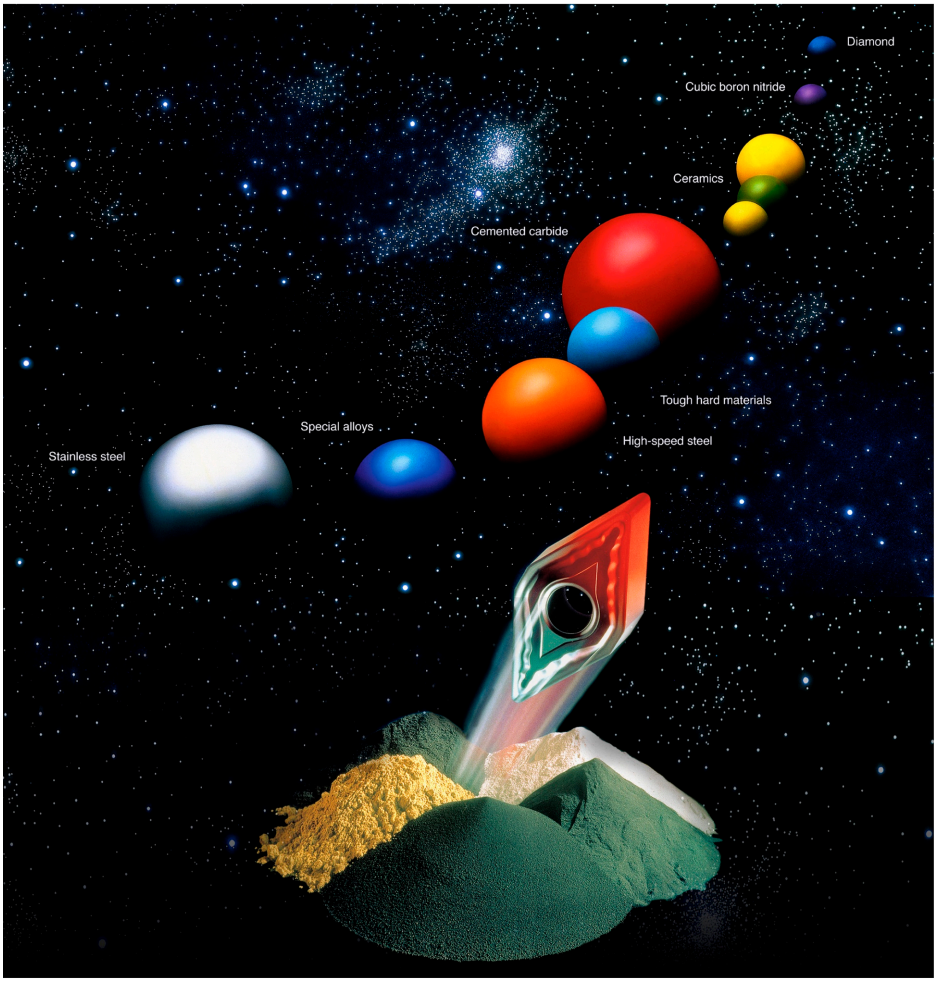
Heavy (-H)

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

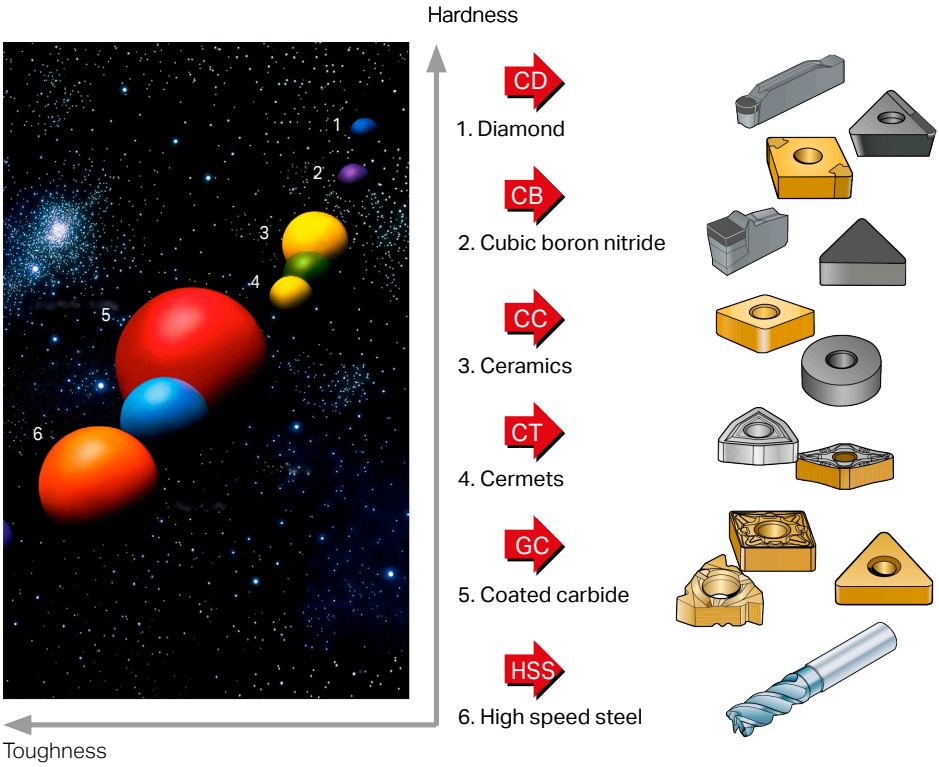
Cutting tool materials

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.



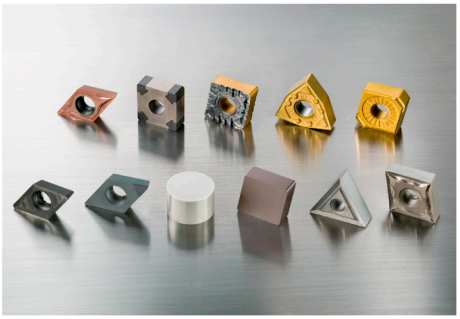
Different types of cutting tool materials



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



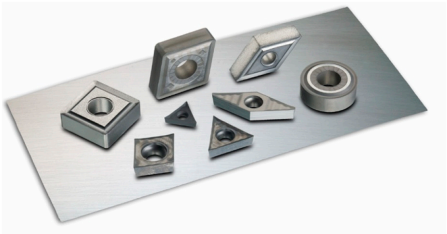
- Uncoated cemented carbide (HW)
- Coated cemented carbide (HC)
- Cermet (HT, HC)
- Ceramic (CA, CN, CC)
- Cubic boron nitride (BN)
- Polycrystalline diamond (DP, HC)

- (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

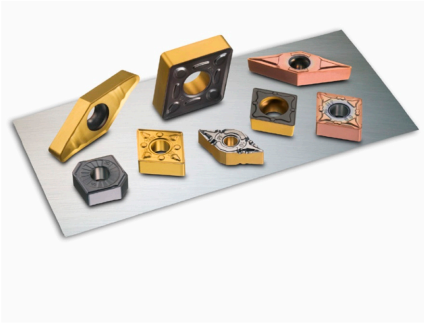
Characteristics, features and benefits



- 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

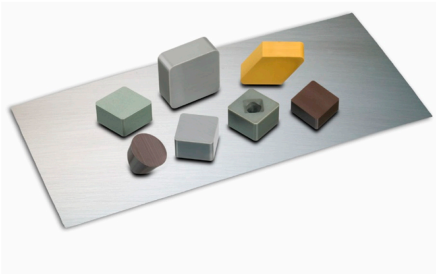


- General use in all kinds of components and materials for turning, milling and drilling applications.
- Extremely good combination of wear resistance and toughness in a variety of jobs.
- Consists of a large variety of grades with hard to tough substrates, usually with gradient sintering, and various 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



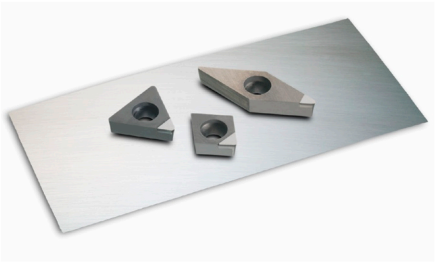
- Depending on type of ceramic, the grades are mainly used in cast iron and steel, hardened materials and HRSA.
- Ceramic grades are generally wear resistant and with good hot-hardness. Wide application area in different types of material and component.
- Ceramics are considered brittle and need stable 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.

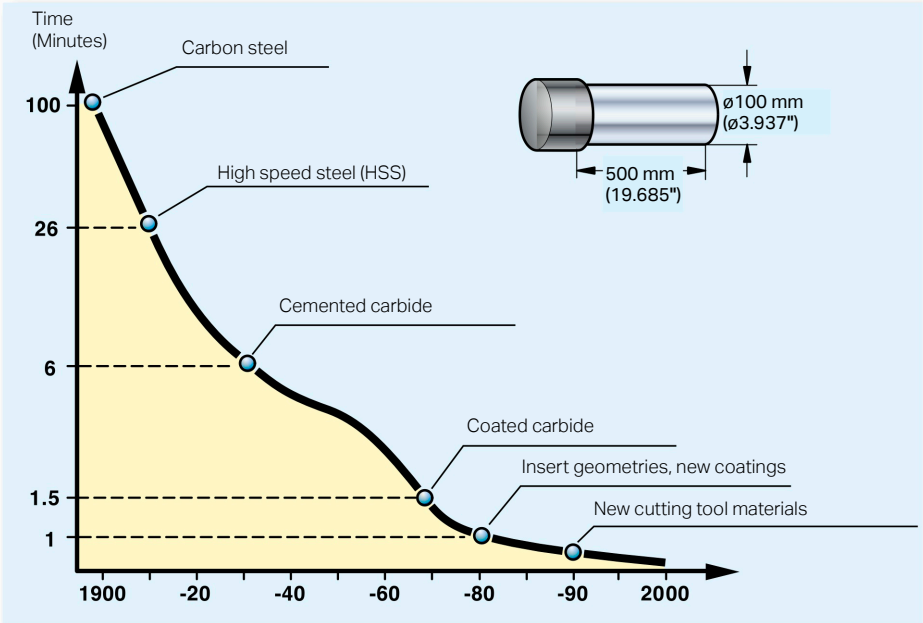
The introduction of HSS brought major improvements, which resulted in reduced cutting time.

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

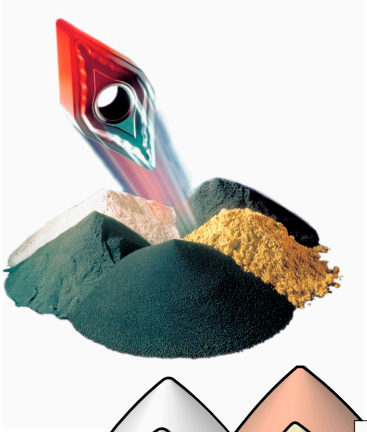
The introduction of coated carbide again lowered the cutting time to 1.5 minutes.

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.

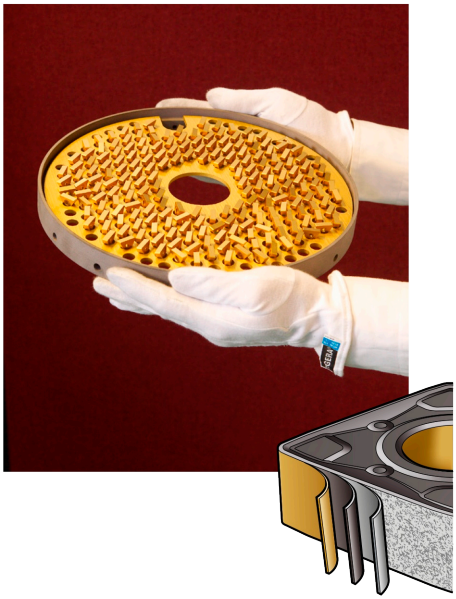


What is cemented carbide and a grade?



- 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



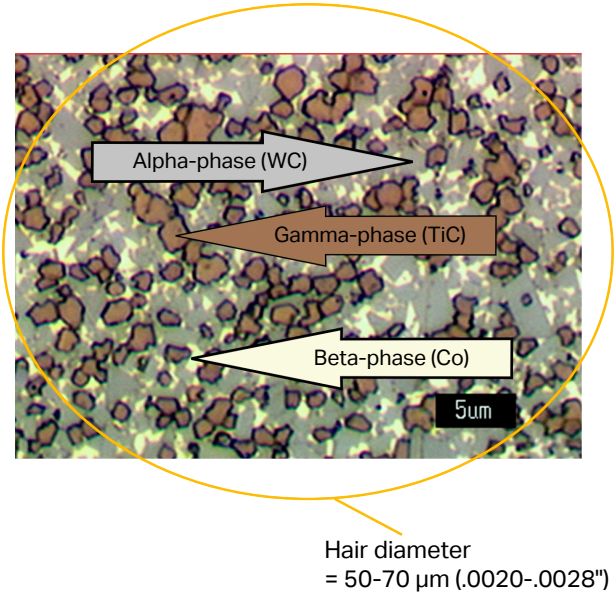
- Coating of cemented carbide was developed in the 1960s.
- A thin Titanium Nitride coating layer was added, only a few microns thick. This improved the performance of carbide overnight.
- Coatings offer improved wear resistance giving longer tool life and possibility to use higher cutting data.
- Today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

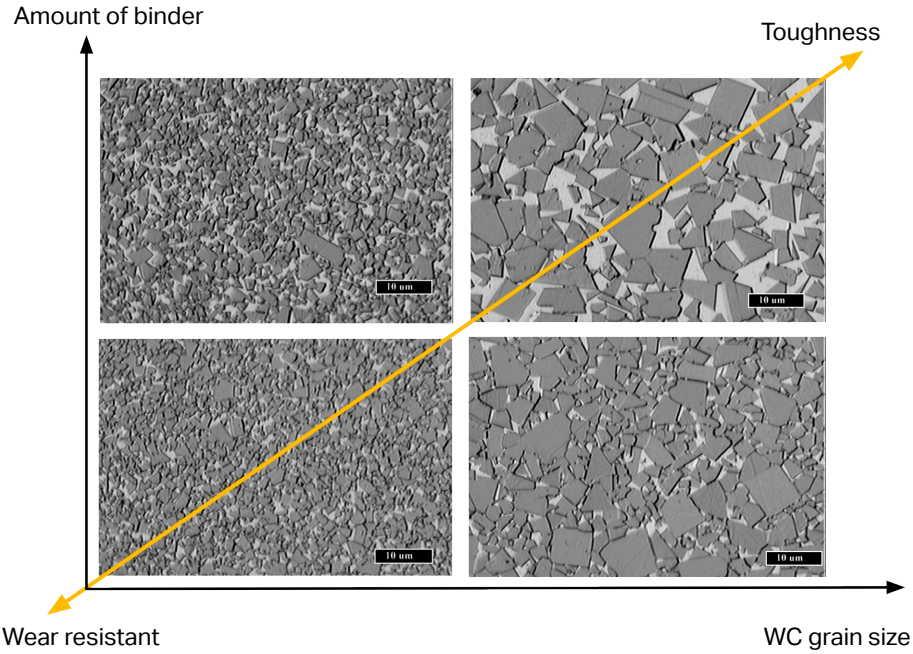
Gamma-phase
(Ti,Ta,Nb)C
(titanium, tantalum, niobium-carbides)

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.

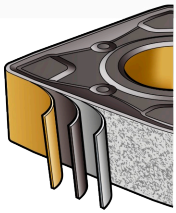
An increase in Co content and WC grain size contributes to an increase in bulk toughness, but also lowers the hardness. As a result, the substrate has less resistance to plastic deformation, which means less wear resistance/lower practical tool life.



Coating design

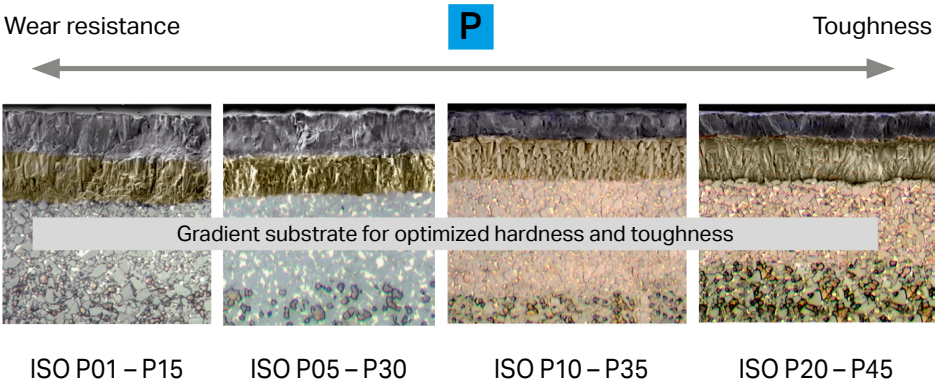
Many factors influence the behavior of the insert:

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



Example of modern steel turning grades

Structure and build-up of the coating layers

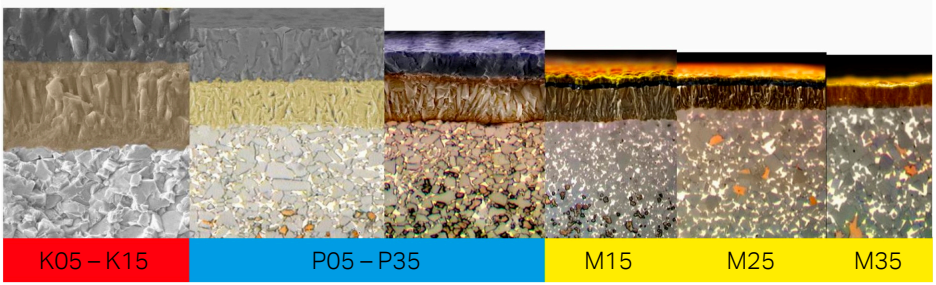


Thicker coatings mean more wear resistance.

Harder substrates mean more deformation resistance.

Grade design

Coatings and substrates vary with the type of application

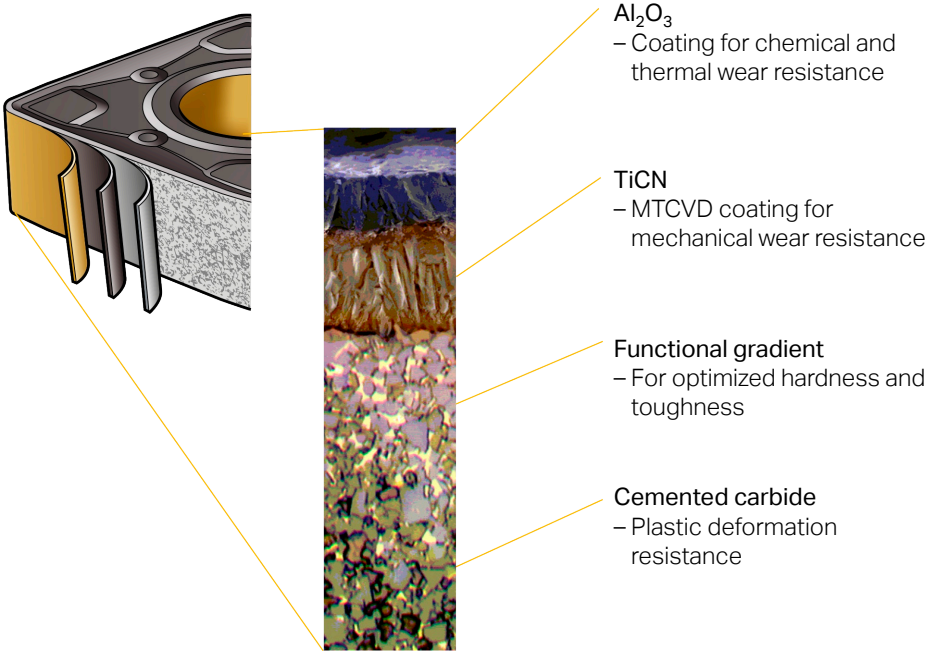


Thicker coatings mean more wear resistance.

Harder substrates mean more deformation resistance.

The coating of a modern turning grade

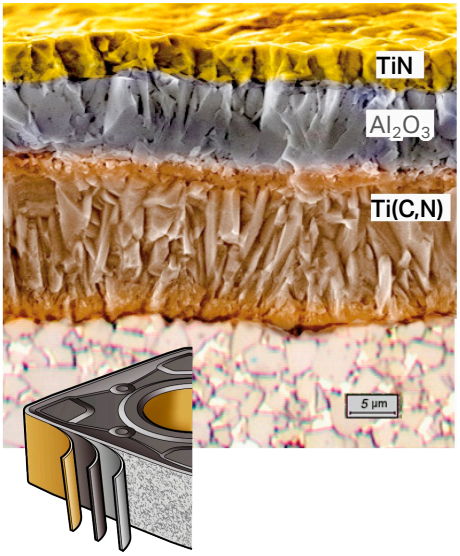
The grade plays a very important part of the performance



Properties of different coating materials

CVD coating of inserts

Chemical Vapor Deposition

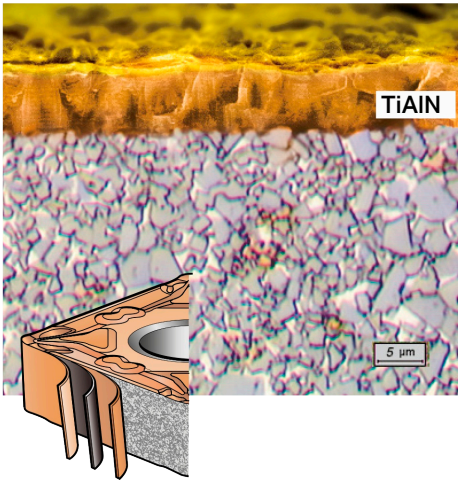


- The most common CVD layers today are TiN, Ti(C,N) and Al_2O_3.
- TiCN provides flank wear resistance.
- Al_2O_3 provides temperature protection (plastic deformation resistance).
- TiN provides easy wear detection.

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

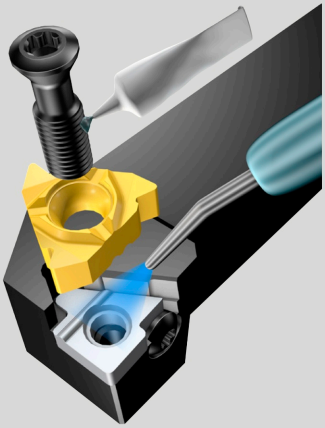
PVD coating of inserts

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



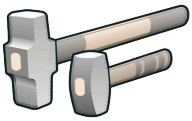
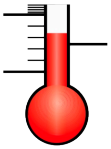
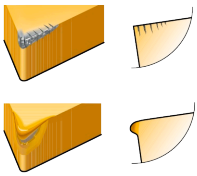
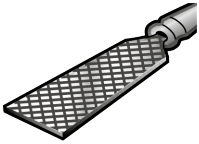
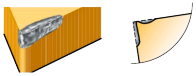
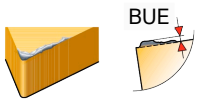
Tool wear & maintenance

• Tool wear H 53

• Maintenance H 61

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

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).

Edge impression

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)

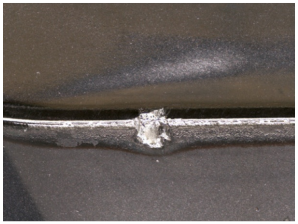
Cracks are narrow openings in which new boundary surfaces have been formed through rupture. Some cracks are confined to the coating, while others extend down into the substrate. Comb cracks are roughly perpendicular to the edge line and most often thermal cracks.

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.

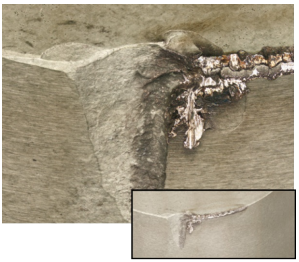


Cause
There are many combinations of wear mechanisms that can cause chipping. However, the most common are thermo-mechanical and adhesive.

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.

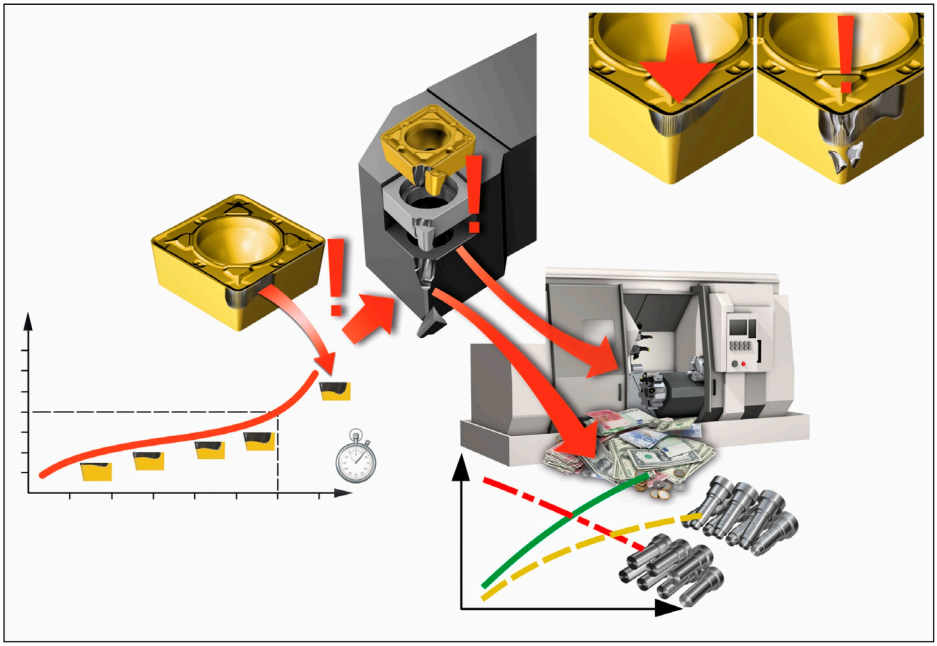
Built up edge (adhesive)

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.

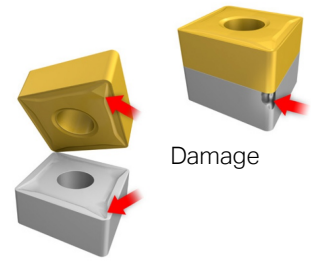
Consequences of poor tool maintenance



- Damaged inserts
- Damaged shims
- Damaged tool holders
- Damaged components
- Damaged machine

Result:
- Reduced production
- Higher production costs

Inspection of tool

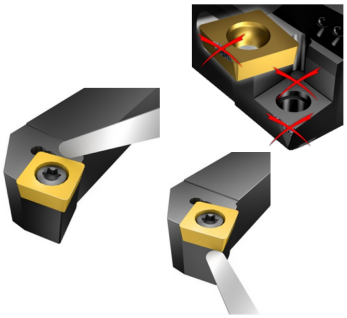


Damage

Chip breakage impression

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?

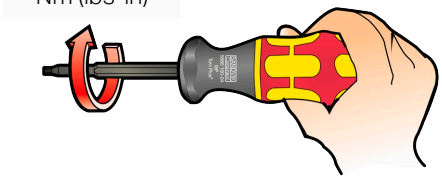
- Extends life of screw and wrench.
- Reduces risk of stripping screw.

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

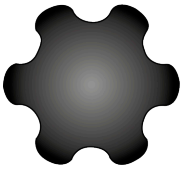
Torx Plus from Sandvik Coromant



Torx Plus® vs. Torx

Cross section

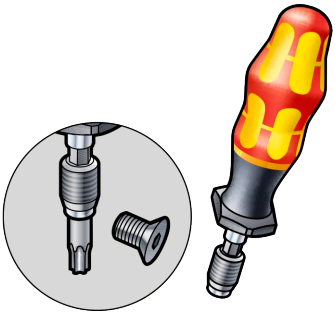
Torx Plus®



Torx

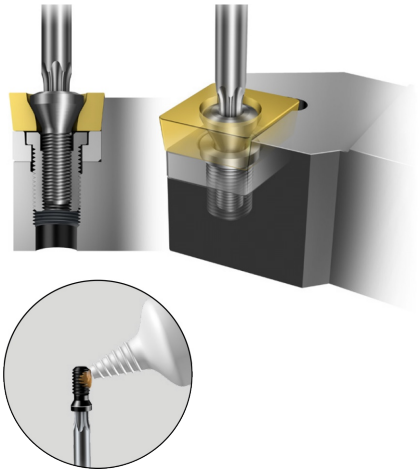
Torx Plus is a registered trademark of Camcar-Textron (USA)

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

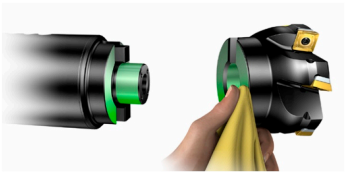


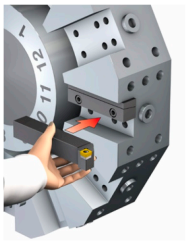
- Screw threads, heads and Torx sockets should be in good condition.
- Use correct keys.
- Ensure correct screw-tightening torque.
- Apply sufficient screw lubrication to prevent seizure. Lubricant should be applied to the screw thread as well as the screw-head face.
- Replace worn or exhausted screws.

Important!

Use Anti-seize for screw heads and threads

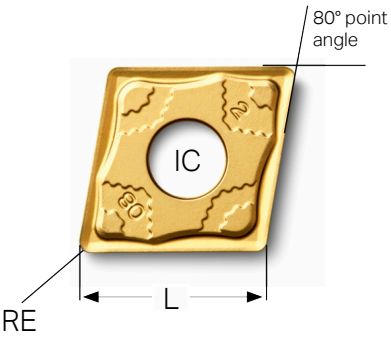
Tool maintenance





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.

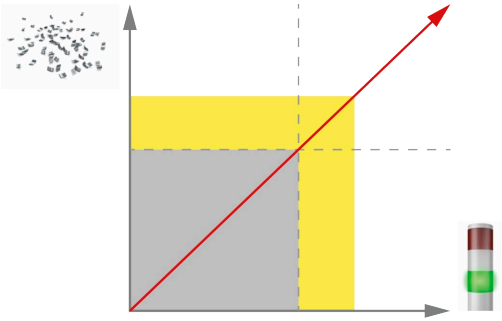
Summary of maintenance checklist

- ☐ 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

How to improve
machining economy H 64

Doing more machining in the same production time



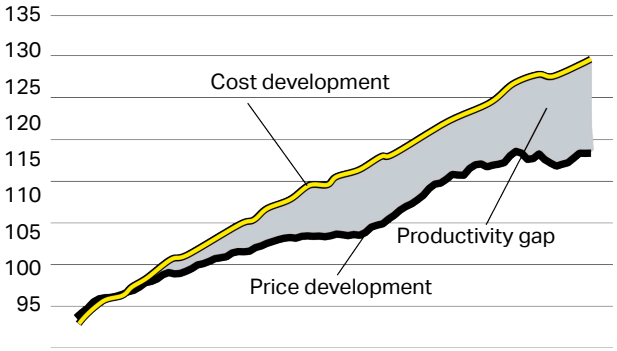
Productivity definition

The value of output produced divided by the value of input or resources.

= Output / Input

Attack the productivity gap

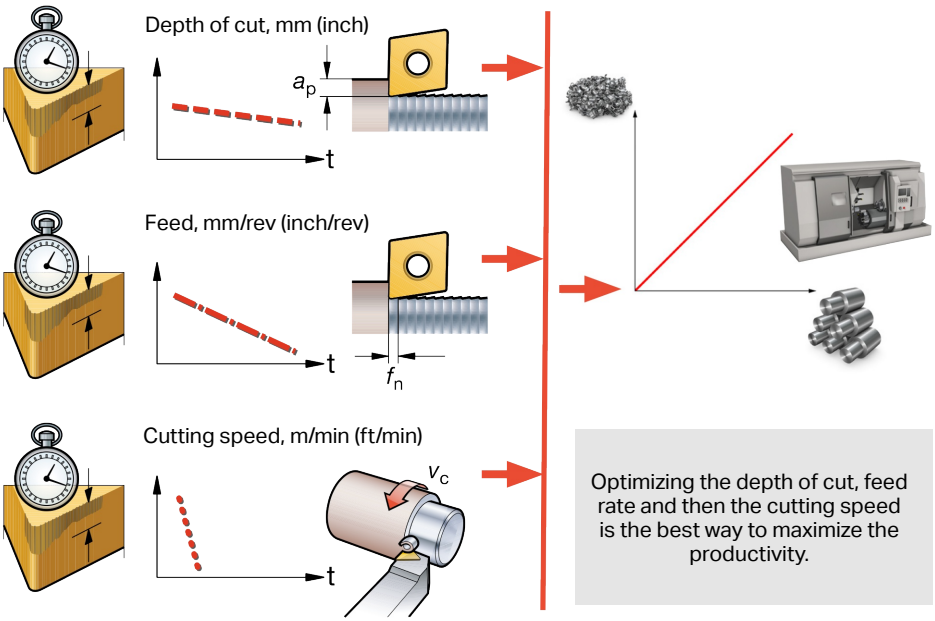
In all industrial operations, the cost of running the operation, e.g. for labor, raw material, equipment, etc., is increasing at a faster rate than the price of the goods that are sold. In order to bridge that gap, one needs to continuously increase efficiency, resulting in higher productivity. Bridging this gap is the only way to stay competitive and ultimately to stay in business.



Source: Mechanical Industry in OECD.

Maximizing productivity

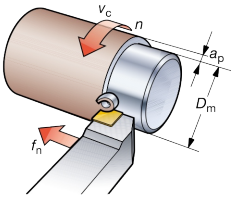
The three main machining parameters, cutting speed, feed, and depth of cut, have an effect on tool life. The depth of cut has the smallest effect followed by the feed rate. Cutting speed has the largest effect by far on insert tool life.



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



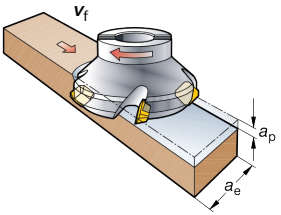
Metric

$$Q = v_c \times a_p \times f_n$$

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

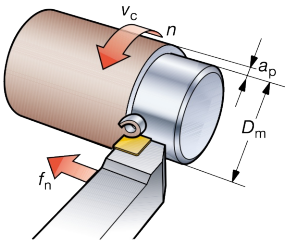
$$Q = a_p \times a_e \times v_f$$

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)

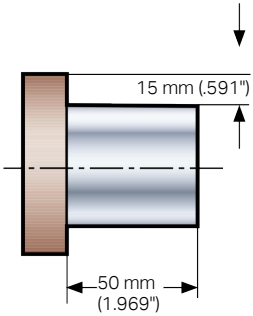
a_p, mm (inch)	3.0 (.118)	3.0 (.118)	3.0 (.118)
f_n, mm/r (inch/r)	0.15 (.006)	0.3 (.012)	**0.5 (.020)**
v_c, m/min (ft/min)	425 (1394)	345 (1132)	**275 (902)**
Q, cm³/min (inch³/min)	191 (12)	310 (19)	412* (25)*

* 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



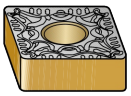
Trigon shape

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

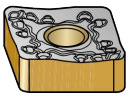
Rhombic shape

Insert: double sided for medium machining.



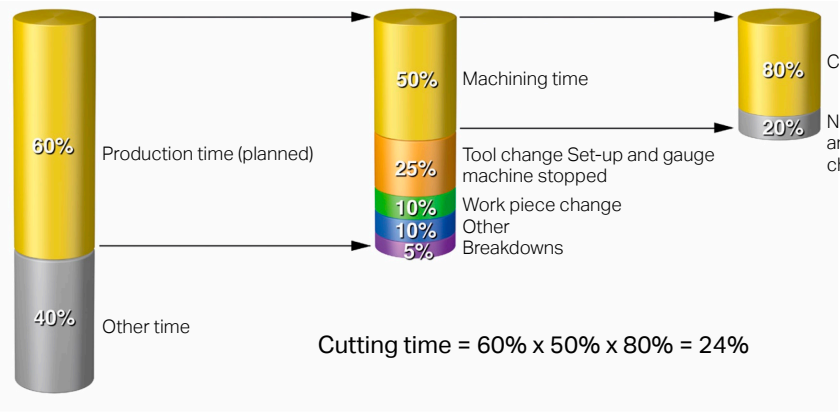
No of passes / cutting depth, a_p	3/5 mm (.118/.197 inch)
Machining time, T_c	16 seconds

Insert: Single sided for rough machining.

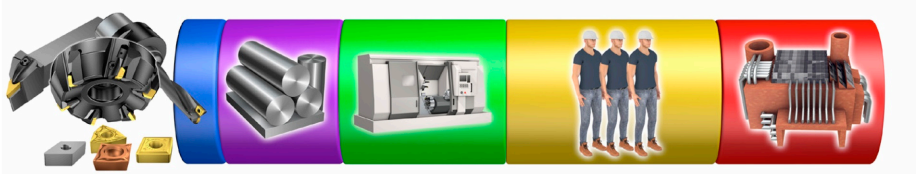


No of passes / cutting depth, a_p	2/7.5 mm (.079/.295 inch)
Machining time, T_c	8 seconds

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:**

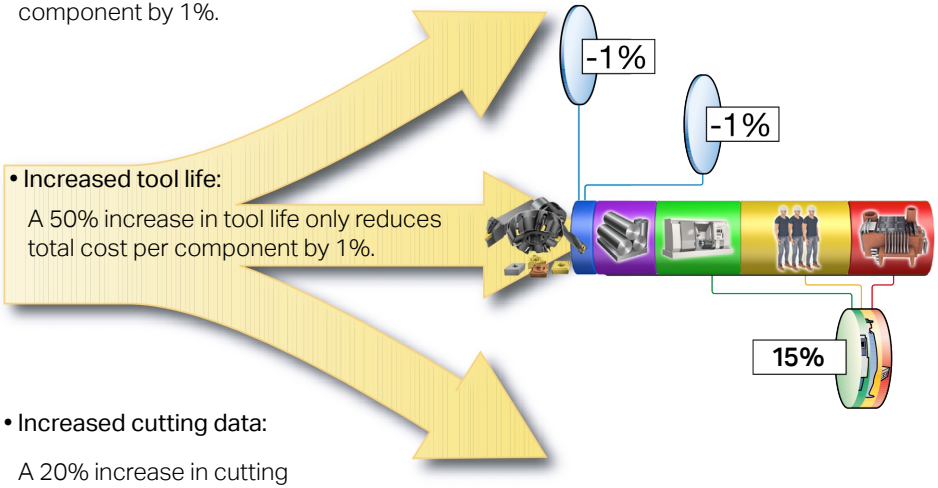
  A 30% decrease in price only reduces total cost per component by 1%.

- **Increased tool life:**

  A 50% increase in tool life 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%.

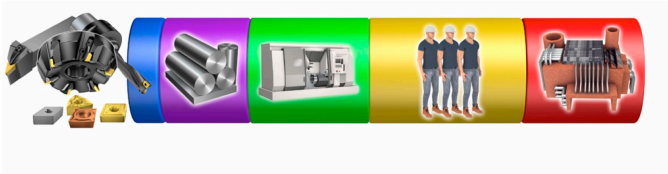


Machine tool utilization

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%

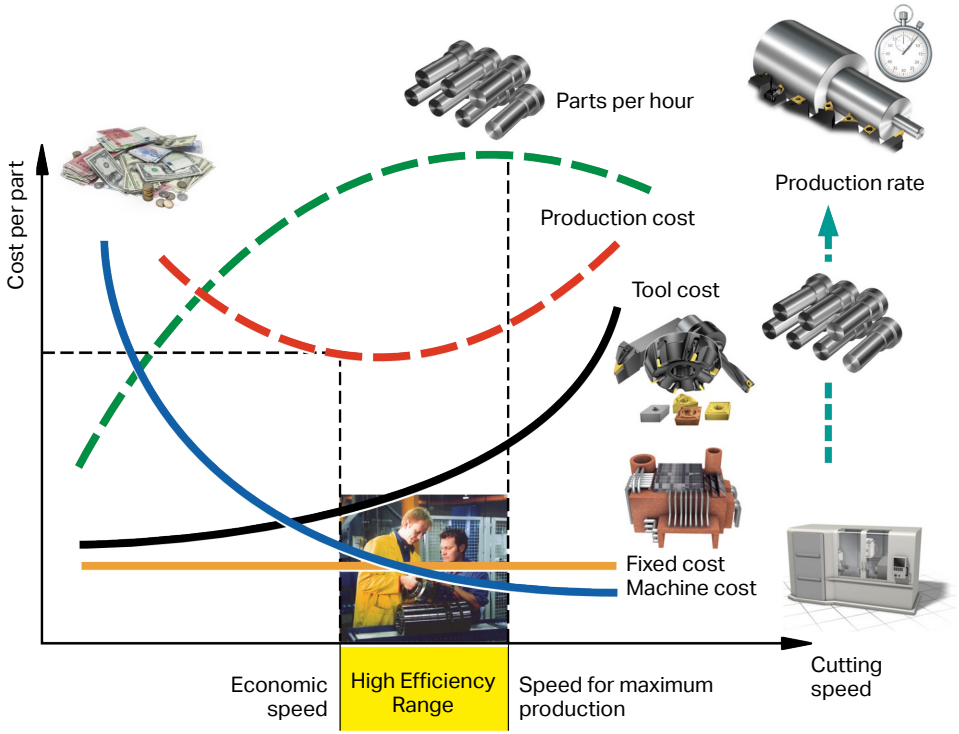
Machining ecomomy

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/mm^2	lbs/inch^2)	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	–

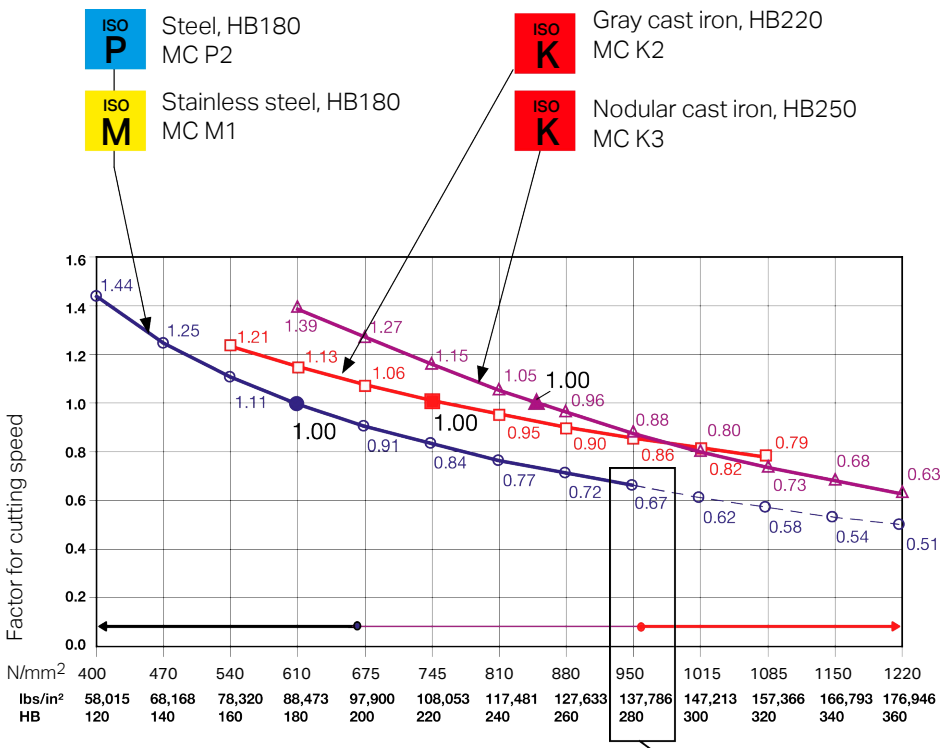
Customer workpiece material (match to information on chart)

Tensile strength = 950 N/mm^2 (137,750 lbs/inch^2)

HB = 280, HRC = 29.2

Example of conversion table, finding the factor for hardness

Diagram form for P, M and K



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).

			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				

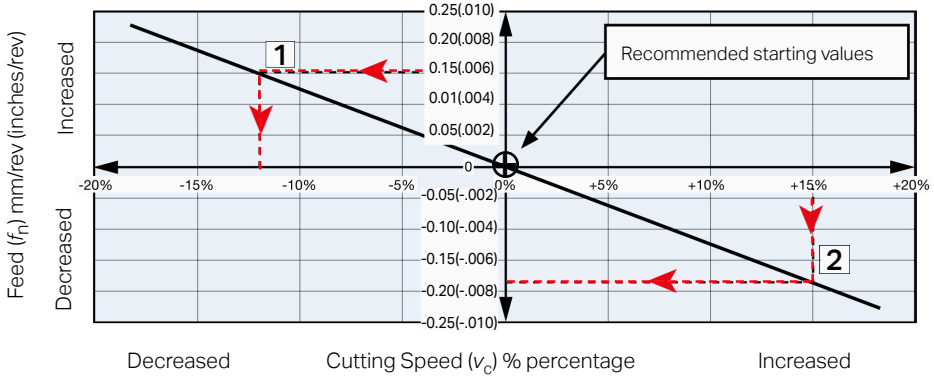
1) MC = material classification code
2) HB = Hardness Brinell
3) HRC = Hardness Rockwell

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%.

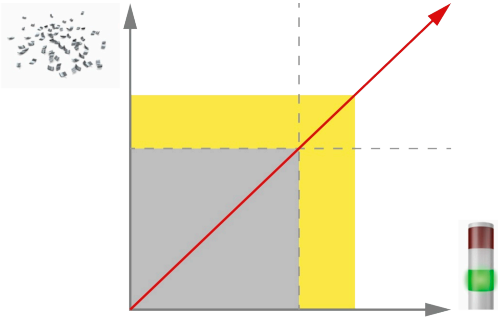
Example 2: Better Surface finish

- 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

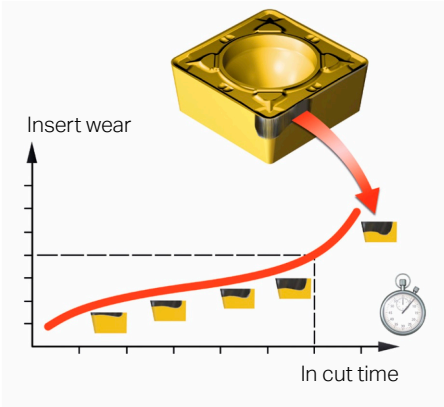
How can you improve productivity?



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 H 79

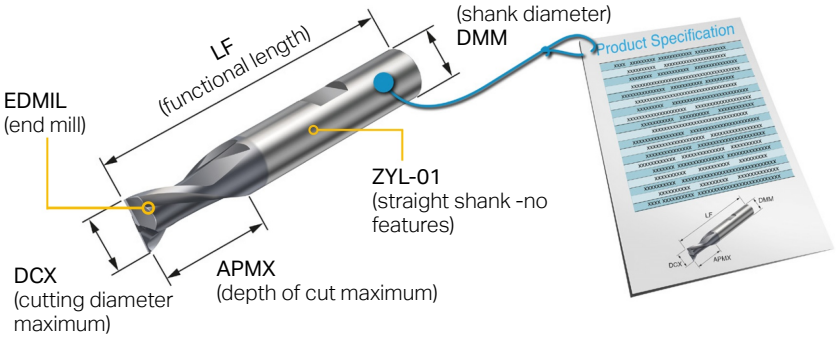
ISO 13399 - The industry standard

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.

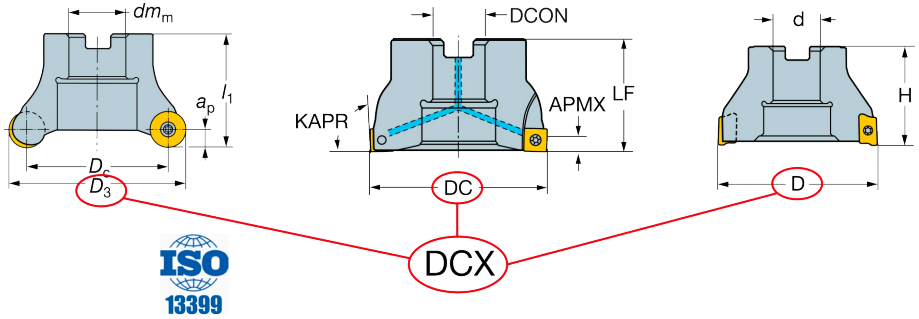
ISO 13399 - What it means for the industry

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.



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

Glossary of terms H 82

Turning H 84

Milling H 86

Drilling H 88

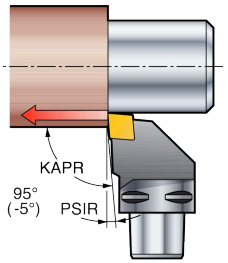
Boring H 90

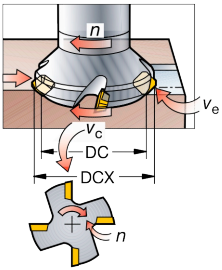
E-learning

E-learning and app information H 92

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.

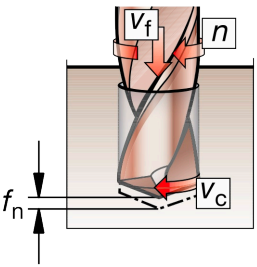
h_{ex} (Maximum chip thickness) is the maximum thickness of the non-deformed chip at the right angles of the cutting edge, and it is influenced by the radial engagement, edge preparation of the insert and feed per tooth. Keep in mind, however, that different radial widths of cut and different entering (lead) angles require feed rate adjustments to maintain proper chip thickness. Measured in millimeters (mm) or inches.

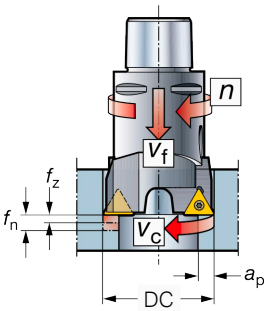
h_m (Average chip thickness) is the average thickness of the non-deformed chip at the right angles of the cutting edge, and it is influenced by the radial engagement, edge preparation of the insert and feed per tooth. Keep in mind, however, that different radial widths of cut and different entering (lead) 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

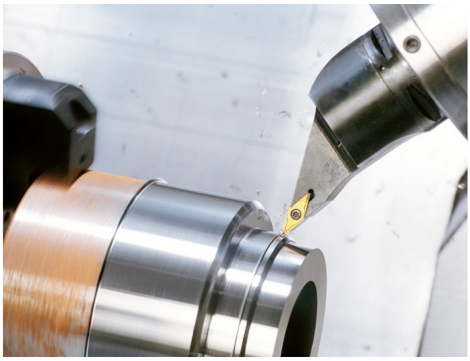
$$k_c = k_{c1} \times \left(\frac{1}{h_m}\right)^{m_c} \times \left(1 - \frac{\gamma 0}{100}\right)$$

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

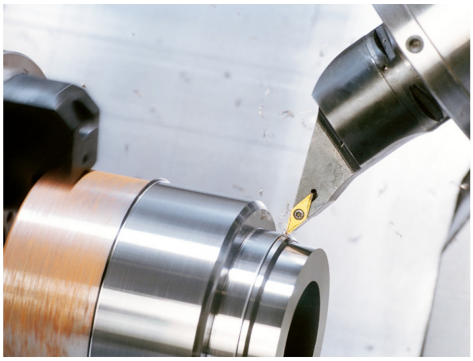
$$k_c = k_{c1} \times \left(\frac{0.0394}{h_m}\right)^{m_c} \times \left(1 - \frac{\gamma 0}{100}\right)$$

Average chip thickness

$$h_m = f_n \times \sin(90 \text{ PSIR})$$

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
$\gamma 0$	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

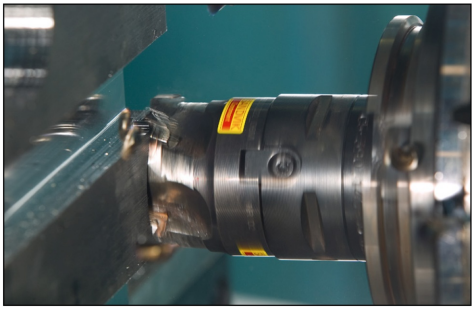
$$Q = \frac{a_p \times a_e \times v_f}{1000}$$

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/mm²
P_c	Net power	kW
M_c	Torque	Nm
Q	Metal removal rate	cm³/min
KAPR	Entering angle	degree

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

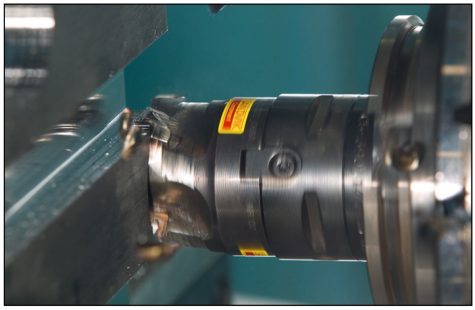
$$Q = a_p \times a_e \times v_f$$

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

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

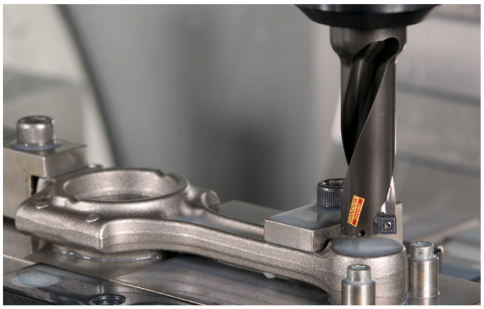
$$Q = \frac{v_c \times DC \times f_n}{4}$$

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

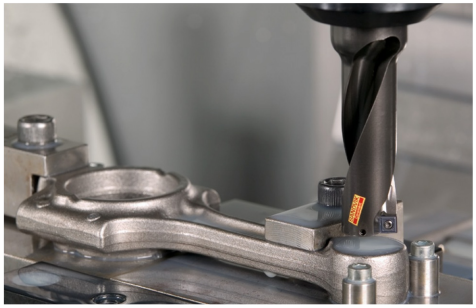
$$Q = v_c \times DC \times f_n \times 3$$

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

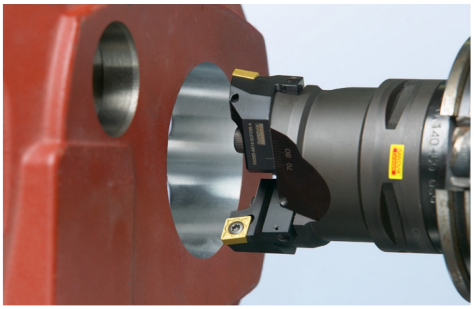
$$Q = \frac{v_c \times DC \times f_n}{4}$$

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/mm²
M_c	Torque	Nm
P_c	Net power	kW
Q	Metal removal rate	cm³/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

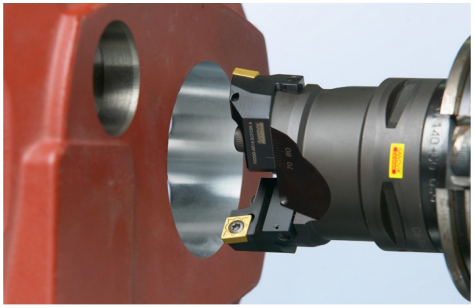
$$Q = v_c \times DC \times f_n \times 3$$

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

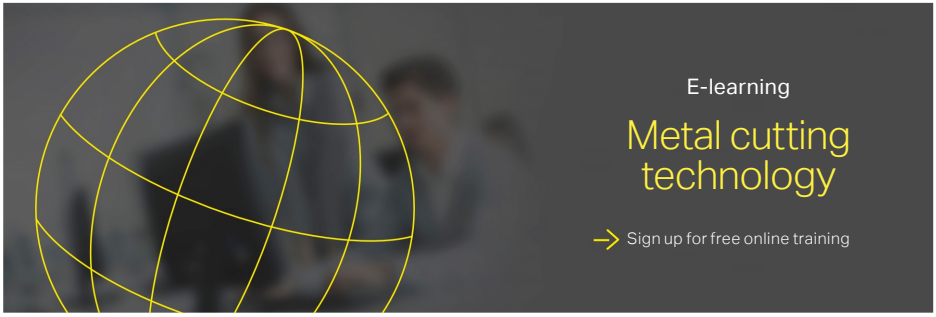
Feed force, N

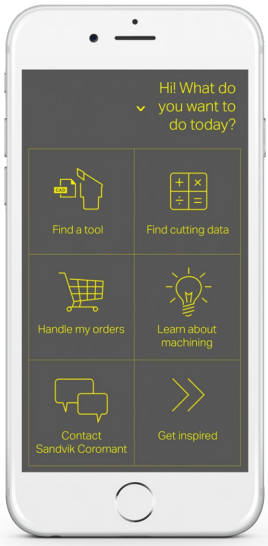
$$F_f \approx 0.5 \times k_c \times a_p \times f_n \times \sin KAPR$$

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

C-2920:40 en-GB © AB Sandvik Coromant 2017.11

SANDVIK
Coromant