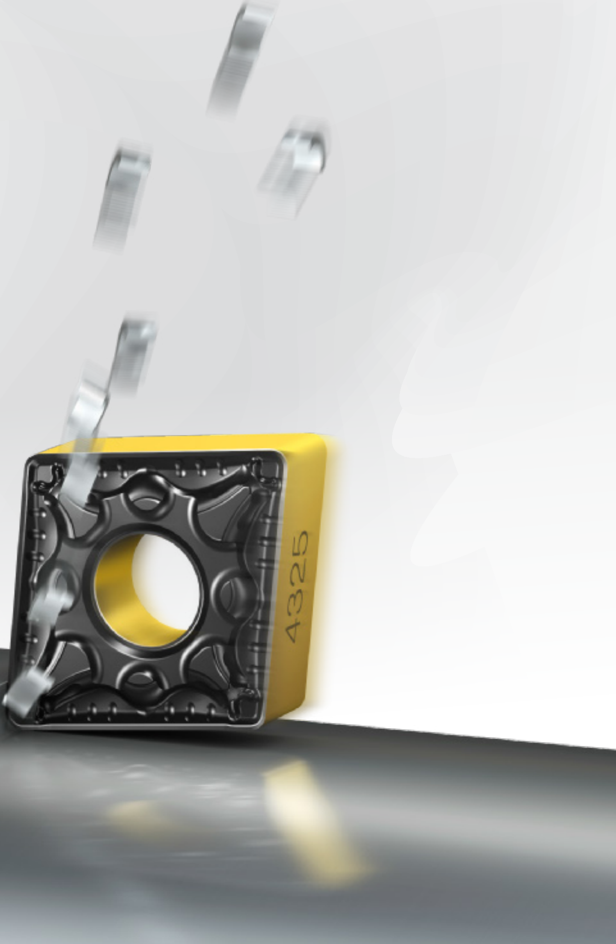
SOUTHERN CUTTING TECHNOLOGY

SANDVIK Coromant



Turning Handbook

General turning - Parting and grooving - Threading

Your conditions

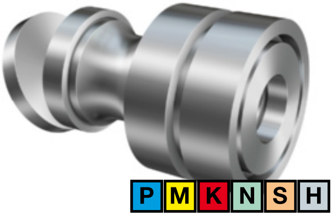
There are several things to consider before starting machining.

Component

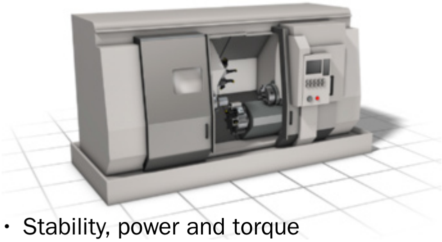
- Operation (e.g. external/internal, profiling, facing, longituding)
- Component design (e.g. large, slender)
- Thread profile
- Batch size
- Quality demands
- Roughing, finishing.

Material

- Machinability (e.g. easy or difficult to break chips)
- Surface structure (e.g. machined, forged)
- Hardness.



Machine



- Stability, power and torque
- Component clamping
- Normal or high coolant pressure
- Coolant supply or dry
- RPM limitations
- Sub-spindle or tailstock?

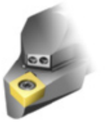
Content

General turning

First choice tool system

	External	Internal

Longitudinal and facing

	External	Internal
Finishing	 T-Max® P with HP*	 CoroTurn® 107 with HP*
Roughing	 T-Max® P with RC*	 T-Max® P with HP*

Profiling

	External	Internal
Finishing	 CoroTurn® TR	 CoroTurn® 107 with HP*
Roughing	 T-Max® P with RC*	 T-Max® P with HP*

Slender/Thin wall components

	External	Internal
Finishing	 CoroTurn® 107 with HP*	
Roughing	 T-Max® P with RC*	

*HP = High precision coolant
*RC = Rigid clamping solution

Geometry and grade

First choice for T-Max P® and CoroTurn® 107

ISO P (Steel)

Machining

Machining / Conditions	Good	Average	Difficult
Roughing	-PR GC4315	-PR GC4325	-PR GC4235
	-PM GC4315	-PM GC4325	-PM GC4235
Finishing	-PF GC4315	-PF GC4315	-PF GC4325

Conditions

ISO M (Stainless steel)

Machining

Machining / Conditions	Good	Average	Difficult
Roughing	-MR GC2025	-MR GC2025	-MR GC2025
	-MM GC2015	-MM GC2025	-MM GC2035
Finishing	-MF GC2015	-MF GC2015	-MF GC2025

Conditions

ISO K (Cast iron)

Machining

(**G**) = Grey, (**N**) = Nodular

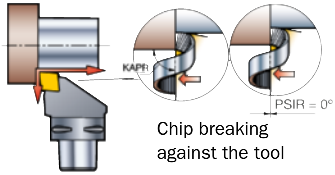
Machining / Conditions	Good	Average	Difficult
Roughing	-KR GC3205 (G) GC3210 (N)	-KR GC3215	-KR GC3215
	-KM GC3205 (G) GC3210 (N)	-KM GC3215	-KM GC3215
Finishing	-KF GC3205 (G) GC3210 (N)	-KF GC3215	-KF GC3215

Conditions

Entering KAPR (lead PSIR) angle

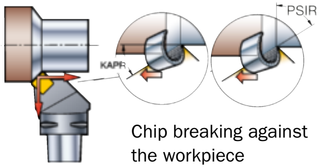
The entering angle KAPR is the angle between the cutting edge and the feed direction.

Large angle:



- Entering angle (KAPR) close to 90° (PSIR 0°) will direct the forces towards the chuck
- Less tendency for vibration
- Higher cutting forces, especially at entrance and exit of the cut.

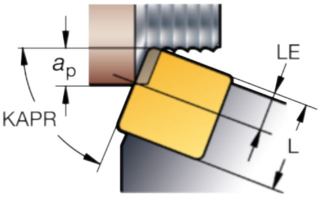
Small angle:

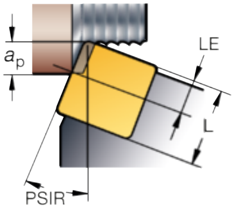


- Forces are directed both axially and radially
- More tendency for vibration
- Reduces insert notch wear
- Reduced load on the cutting edge at entrance/exit

Insert size

- Determine the largest depth of cut, a_p
- Determine the necessary cutting length, LE, while also considering the entering KAPR (lead PSIR) angle of the tool holder, and the depth of cut, a_p.





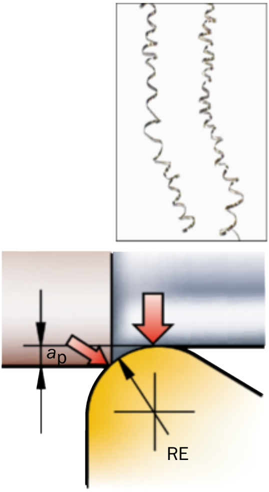
Example to reach a_p 5.0 mm (0.197 inch):

KAPR (PSIR)	LE mm (inch)	Insert:
75° (15°)	5.2 (0.205)	SNMG 1204 / SNMG 43
45° (45°)	7.1 (0.280)	SNMG 1506 / SNMG 54 (less sensitive for insert breakage)

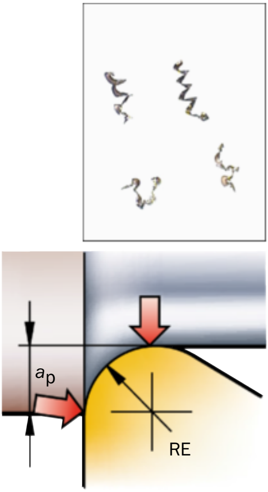
Nose radius

- Select the largest possible nose radius, RE, to obtain a strong cutting edge
- A large nose radius, RE, permits larger feeds and edge security
- Select a smaller radius, RE, if there is a tendency of vibration.

	Nose radius, RE, mm (inch):				
	0.4 (1/64)	0.8 (1/32)	1.2 (3/64)	1.6 (1/16)	2.4 (3/32)
Max feed, f_n mm/r inch/r	 0.25–0.35 .009–.014	 0.4–0.7 .016–.028	 0.5–1.0 .020–.039	 0.7–1.3 .028–.051	 1.0–1.8 .039–.071



a_p < RE



a_p = 2/3 x RE

The depth of cut, a_p, should be no less than 2/3 of the nose radius, RE, to avoid vibration and bad chips.

Note: For more information, see headline Productivity booster.

Wiper inserts

Wiper inserts are capable of turning at high feed rates without losing the capability to generate good surface finish or chip breaking ability.

Use wiper insert as first choice where possible:

- Longitudinal and facing applications
- Stable component set-ups
- Uniform component shapes.

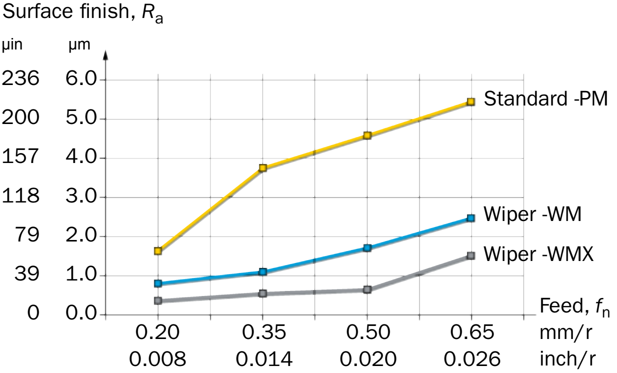
Note: Wiper insert is not recommended for internal machining with long overhang due to vibration.

-WMX

-WMX insert is first choice within the negative wiper family.

-WF

-WF insert is first choice within the positive wiper family.



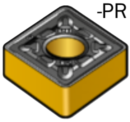
Two times feed with a wiper will generate as good or better surface as conventional geometries with normal feed.
The same feed with a wiper will generate twice as good surface compared with conventional geometries.

Geometry

Every insert has a working area with optimized chip control:

Roughing

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



Medium

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

Finishing

Operations at small depths of cut and low feed rates. Operations requiring low cutting forces.

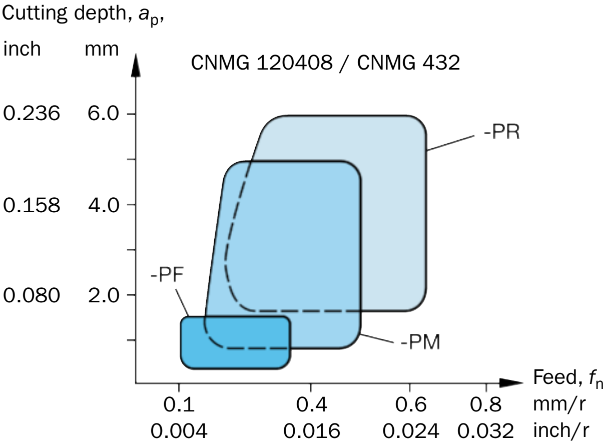
The diagram below shows the working area for a CNMG 120408 insert, based on acceptable chip breaking in relation to feed and depth of cut.

The chip illustration is an example from the diagram and cutting data:

Geometry: -PM

a_p: 3.0 mm (0.118 inch)

f_n: 0.3 mm/r (0.012 inch/r)

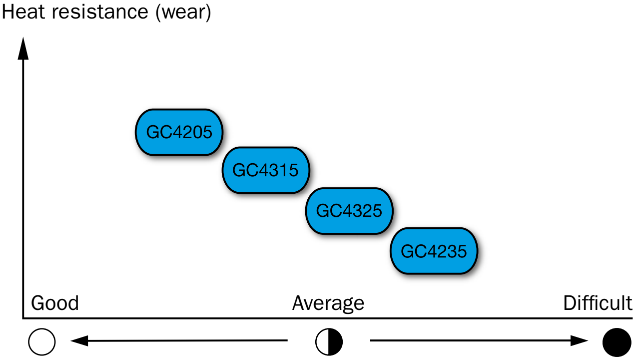


- First choice is -PM geometry
- Use -PR geometry for high f_n/a_p or interrupted cuts
- Use -PF geometry for low f_n/a_p.

Grade

The insert grade is primarily selected according to:

- Component (material and design, e.g. long or short time in cut)
- Application (e.g. roughing, medium or finishing)
- Machine (stability, e.g. good, average or difficult).



Example

- Steel component, MC P2.3.Z.AN (CMC 02.12)
- Medium machining, f_n 0.2–0.4 mm/r (0.008–0.016 inch/r), cutting depth, a_p, 2 mm (0.079 inch)
- Good stability (clamping, component size).

First choice: Use GC4325 grade for secure machining.

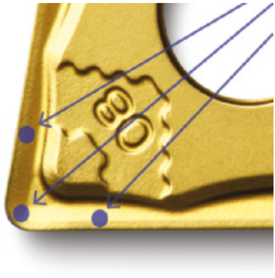
Use a GC4315 grade if higher need of heat resistance because of longer engagement time or higher cutting speed.

Productivity booster

Effects of HP (high pressure/precision coolant)

Chip control and tool life:

- Positive effects seen at 10 bar (145 psi)
- Even more obvious at 70 bar (1015 psi)
- At higher pressure, insert geometries dedicated for HP adds tool life.



Process security

Using a tool holder with high precision coolant (HP) improves chip control and supports predictable tool life. This can be seen when changing from a conventional holder to a CoroTurn® HP holder without changing any cutting parameters.

HP also gives room for increased cutting speed.

Consider the following for predictable and productive machining in stainless steel with bad chip breaking:

- Apply high coolant pressure 70 bar (1015 psi). Improvements are seen already at 35 bar (507 psi)
- Use CoroTurn® HP in combination with -MMC geometry.

Increase tool life

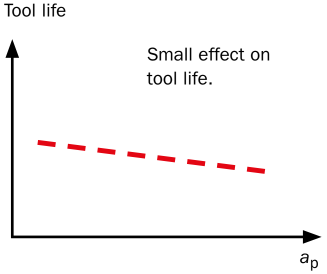
For best tool life:

1. Maximize a_p (to reduce number of cuts)
2. Maximize f_n (for shorter cutting time)
3. Reduce v_c (to reduce heat)

Cutting depth a_p

Too small:
- Loss of chip control
- Vibration
- Excessive heat
- Uneconomical.

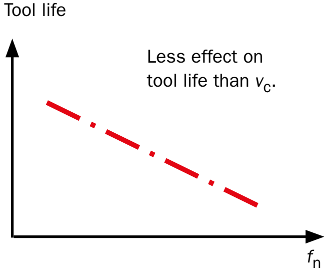
Too deep:
- High power consumption
- Insert breakage
- Increased cutting forces.



Feed rate f_n

Too light:
- Stringers
- Rapid flank wear
- Built-up edge
- Uneconomical.

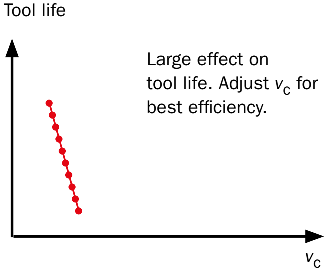
Too heavy:
- Loss of chip control
- Poor surface finish
- Crater wear/plastic deformation
- High power consumption
- Chip welding
- Chip hammering.



Cutting speed v_c

Too low:
- Built-up edge
- Dulling of edge
- Uneconomical
- Poor surface.

Too high:
- Rapid flank wear
- Poor finishing
- Rapid crater wear
- Plastic deformation.



Application tips

Vibration prone components

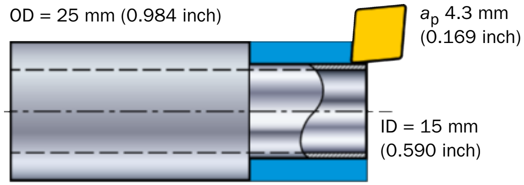
Cut in one pass (for exampel a tube)

Recommendation is to machine the whole cut in one pass to direct the force into the chuck/spindle.

Example:

- Outer diameter (OD) of 25 mm (0.984 inch)
- Inner diameter (ID) of 15 mm (0.590 inch)
- Depth of cut, a_p, is 4.3 mm (0.169 inch).

Resulting thickness of the tube = 0.7 mm (0.028 inch).

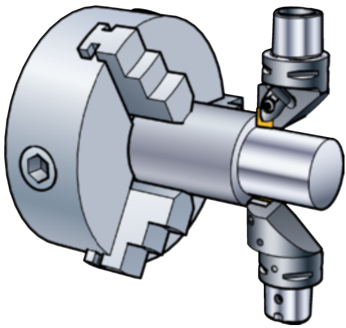


An entering angle close to 90° (lead angle 0°) can be used for directing the cutting forces in axial direction. This leads to minimal bending force on the component.

Cut in two pass

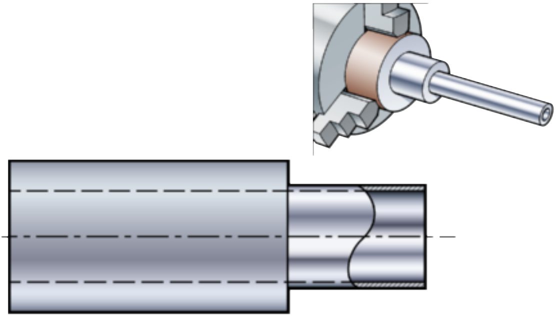
Syncronized upper and lower turret machining will level out radial cutting forces:

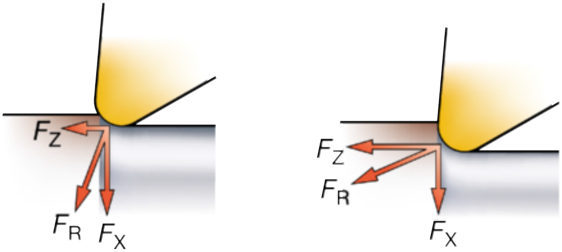
- Avoid vibration and bending of the component.



Slender/thin wall components

- Entering angle close to 90° (lead angle 0°)
- Depth of cut, a_p, bigger than nose radius, RE
- Sharp edge and small nose radius, RE
- Consider Cermet or PVD grade, e.g. CT5015 or GC1125.





Entering angle (lead angle):

- Even a small change (from a 91/-1 to a 95/-5 degree angle) will impact the cutting force direction during machining.

Depth of cut, a_p, bigger than nose radius, RE:

- Large a_p increases the axial force, F_z, and decreases the radial cutting force, F_x, which causes vibration.

Sharp edge and small nose radius, RE:

- Generates low cutting forces.

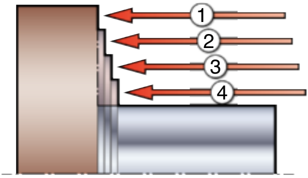
Cermet or PVD grade:

- To provide wear resistance and a sharp insert edge which is preferable in this type of operation.

Shouldering/turning shoulder

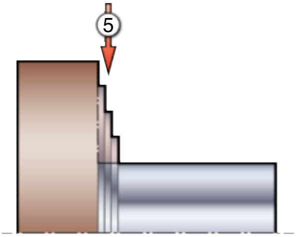
Step 1-4:

- The distance of each step (1-4) shall be the same as the feed rate to avoid chip jamming.



Step 5:

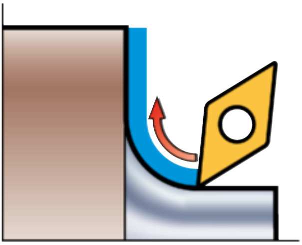
- The final cut shall be done in one vertical cut starting from outer diameter towards inner diameter.



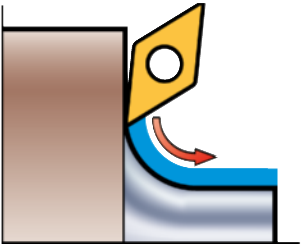
This:

- Avoids damage of the insert edge
- Is very favourable for CVD coated inserts and may reduce fractures considerably!

Problems can also occur with wrap around chips on the radii if going from inner diameter to outer diameter when facing up on the shoulder.



Changing the tool path can reverse the chip direction and solve the problem.



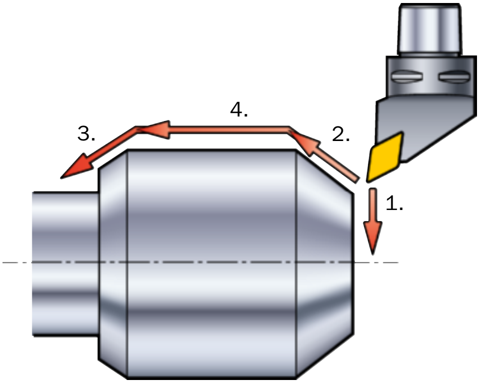
Facing

Process considerations:

- Start with the facing (1) and the chamfer (2) if possible.

Geometrical conditions on workpiece:

- Start with the chamfer (3).



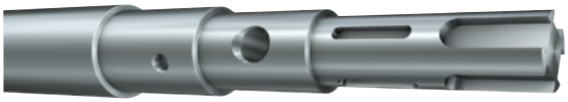
Facing shall be the first operation to set the reference point on the component for next pass.

Burr formation is often a problem at the end of the cut (when leaving the workpiece). Leaving a chamfer or a radius (rolling over a corner) could minimize or avoid burr formation.

A chamfer on the component will lead to a smoother entry of the insert edge (both in facing and longitudinal turning).

Interrupted cuts

- Use a PVD grade to provide edge-line toughness, e.g. GC1125
- Use a thin CVD grade if workpiece material is very abrasive, e.g. GC1515
- Consider a strong chip breaker, e.g. -QM or -PR to add sufficient chipping resistance
- A recommendation is to turn off coolant to avoid thermal cracks.
- Consider tough grade like GC4235 for heavier interruptions.

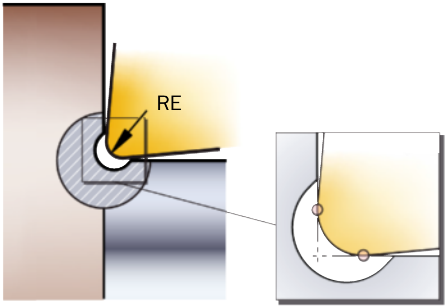


Finishing component with grinding relief

Use biggest possible nose radius, RE, for longitudinal and face turning. Do not exceed the width of the grinding.

- Strong edge
- Good surface quality
- Possibility to use high feed.

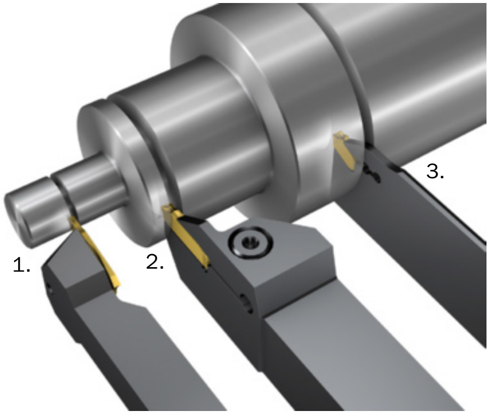
The relief shall be performed as the last operation to take away burr.



Parting and grooving

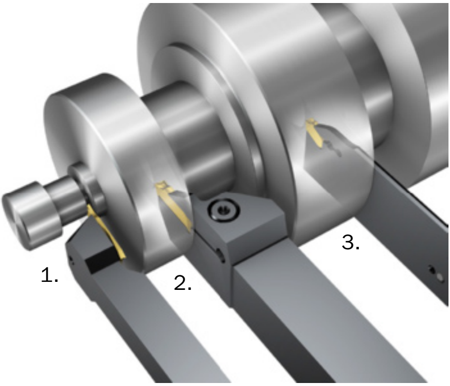
First choice system

Parting off



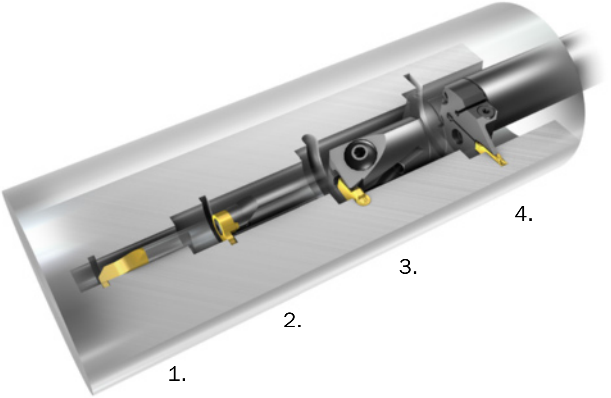
1. CoroCut® 3 DCX Ø ≤12 mm (0.5 inch)
2. CoroCut® 2 DCX Ø12-38 mm (0.5–1.5 inch)
3. CoroCut® QD DCX Ø38-160 mm (1.5–6.3 inch)

External grooving



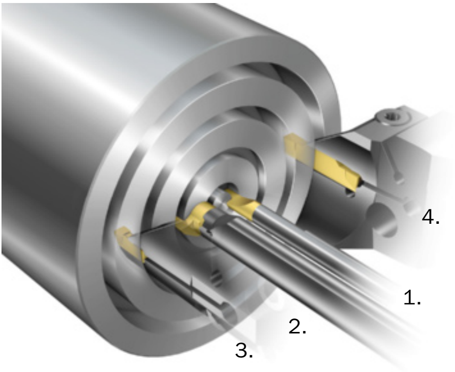
1. CoroCut® 3 CDX 1.5–6 mm (0.06–0.24 inch)
2. CoroCut® 2 CDX 13–28 mm (0.5–1.1 inch)
3. CoroCut® QD CDX 15–80 mm (0.6–3.15 inch)

Internal grooving



1.	CoroTurn® XS	DMIN Ø4.2 mm (0.165 inch)
2.	CoroCut® MB	DMIN Ø10 mm (0.394 inch)
3.	T-Max Q-Cut®	DMIN Ø12 mm (0.472 inch)
4.	CoroCut® 2	DMIN Ø26 mm (1.024 inch)

Face grooving



1.	CoroTurn® XS	DAXIN Ø1-8 mm (0.04–0.315 inch)
2.	CoroCut® MB	DAXIN Ø8 mm (0.31 inch)
3.	T-Max Q-Cut®	DAXIN Ø16 mm (0.63 inch)
4.	CoroCut® 2	DAXIN Ø34 mm (1.34 inch)

Application tips for parting off

Minimize overhang, OH

At long OH:

- Use a light cutting geometry, e.g. -CM.

OH less than 1.5 x H:

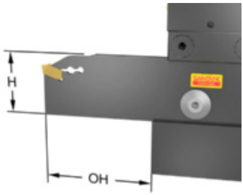
- Use recommended feed for the geometry.

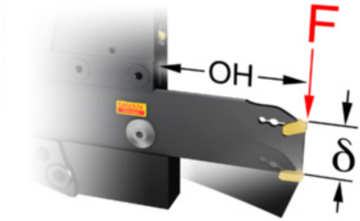
OH exceeds 1.5 x H:

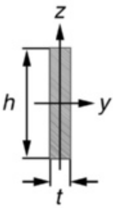
- Reduce feed rate to the lower end of recommended feed for the geometry.

Shorter overhang decreases bending down in cubic:

$$\delta = \frac{4 \times F \times OH^3}{t \times h^3}$$



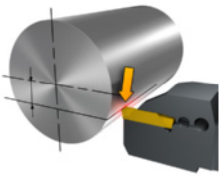




Centre height

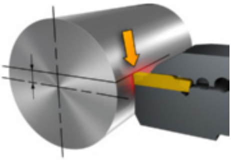
- Centre height ±0.1 mm (±0.004 inch)
- At long overhangs, set cutting edge 0.1 mm (0.004 inch) above centre to compensate for bending down.

Below centre causes:



- Increased pip
- Breakage (unfavourable cutting forces).

Over centre causes:



- Breakage (pushing through centre)
- Rapid flank wear (small clearance).

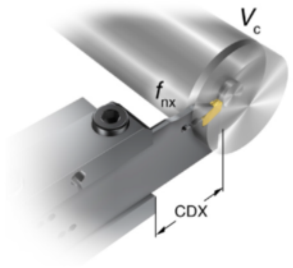
Always reduce feed before centre

Breakages in parting off bars generally occur at centre. Always reduce feed by -75% from 2 mm (0.08 inch) before centre:

- Lower feed at centre reduces the forces and increases tool life
- Higher feed in periphery improves productivity and tool life
- Feed reduction at centre drastically increases tool life.

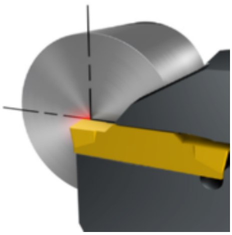
Calculating speed:

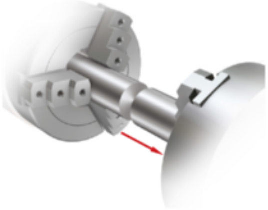
$$v_c = \frac{\pi \times D_m \times n}{1000}$$



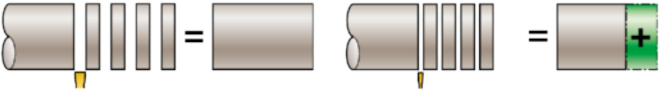
Always stop feed before reaching centre

- Stop feed 0.5 mm (0.02 inch) before centre
- The component will fall off by the centrifugal force.


Feeding through centre causes breakage.

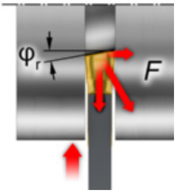

A sub chuck can be used to pull the component.
Leave a pip with ø 1 mm (0.04 inch) to be pulled off.

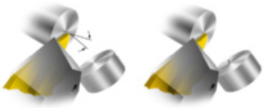
Reduce insert width to save material.



Pip free parting

- Front angle reduces pip and burr on one side
- Use front angle inserts only at small overhangs
- Front angle reduces tool life and increase bending
- For longer overhangs use neutral inserts.



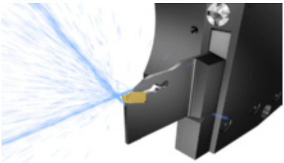


	Front angle	Neutral
Stability and tool life	bad	good
Radial cutting forces	low	high
Axial cutting forces	high	low
Pip/burr	small	large
Risk of vibration	high	low
Surface finish and flatness	bad	good
Chip flow	bad	good

High precision coolant (HP)

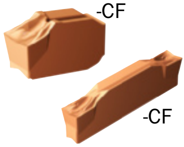
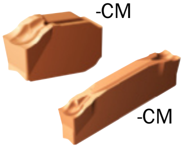
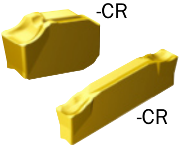
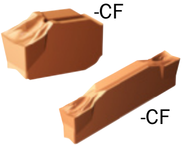
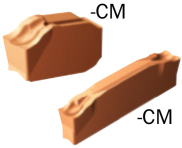
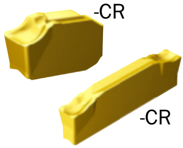
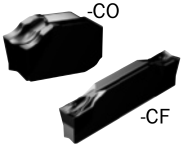
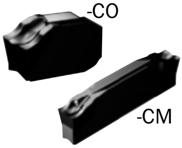
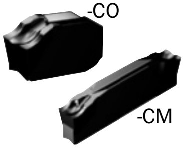
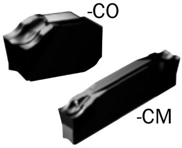
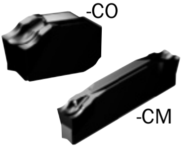
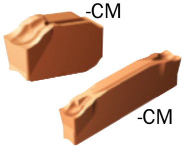
- Accesses cutting edge even in deep grooves
- Tools with HP is first choice for parting and grooving
- Improves chip control and surface finish
- Internal coolant decreases temperature
- Largest gains at long time in cut and materials with low conductivity (HRSA, Stainless steel)
- Effective coolant allows usage of tougher grades with maintained or increased tool life
- Increase cutting speed with 30-50% when HP is used
- Shut off coolant at the diameter where the machine reach its rpm limit to avoid build-up edge.

High precision coolant gives good effect also at lower pressures but best effect at 20 bar (290 PSI) and higher.

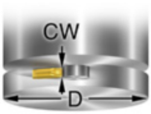


Geometry and grade

First choice for parting off

ISO	 Tubes - good conditions	 Bars - good conditions (sub-chuck)	 Bars - difficult conditions
P Steel	 -CF -CF **GC1125**	 -CM -CM **GC1125**	 -CR -CR **GC1135/2135**
M Stainless steel	 -CF -CF **GC1125**	 -CM -CM **GC1125**	 -CR -CR **GC1135/2135**
N Non-ferrous metals	 -CO -CF **GC1105**	 -CO -CM **GC1105**	 -CO -CM **GC1105**
S HRSA	 -CO -CM **GC1105**	 -CO -CM **GC1105**	 -CM -CM **GC1145**

Use the table to choose insert width, CW, depending on component diameter, D:



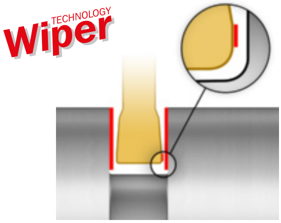
D mm (inch)	CW mm
–10 (–0.4)	1.0
10–25 (0.4–1.0)	1.5
25–40 (1.0–1.6)	2.0
40–50 (1.6–2.0)	2.5
50–65 (2.0–2.6)	3.0

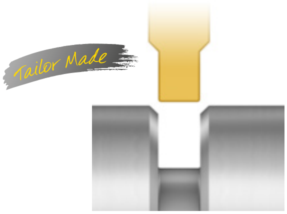
Save material by reducing insert width!

Application tips for external grooving

Single cut grooving

- Use Wiper inserts for surface finish, e.g. -TF
- Wide range of different corner radii and widths with tight tolerances offered with CoroCut 2 -GF
- Tailor Made with option of specific profile and chamfers in insert profile for mass production.

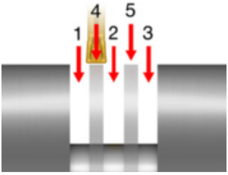




Roughing wide grooves

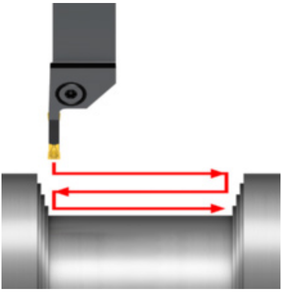
Multiple grooving

- For deep wide grooves (depth greater than width)
- Flanges left for final cuts (4 and 5) shall be thinner than insert width (CW -2 x corner radii)
- Increase feed 30-50% when machining flanges
- First choice geometry -GM.



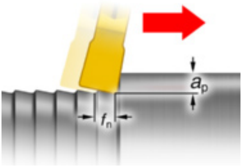
Plunge turning

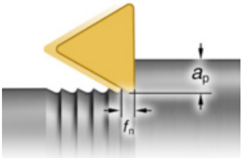
- For wider and more shallow grooves (width greater than depth)
- Stop turning before shoulder
- First choice geometries -TF and -TM.

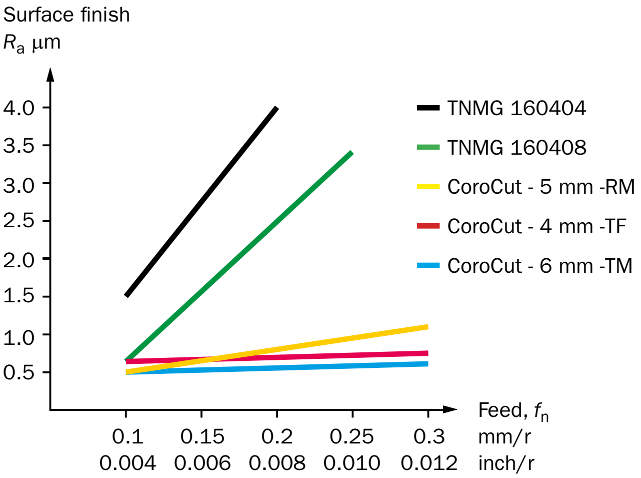


Turning with parting and grooving insert

- When side turning use a_p larger than insert corner radii
- Wiper effect – f_n/a_p must be high enough to generate a small insert and tool bending
- Too low f_n/a_p causes tool rubbing, vibration and poor surface finish
- Max a_p 75% of insert width.





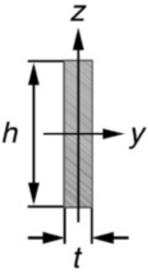


The diagram shows surface finish for CoroCut inserts in comparison to a TNMG insert with a 04 or 08 corner radius.

Turning a groove

When side turning tool and insert must bend. However, too much bending can cause vibration and breakages:

- Thicker blade decreases bending
- Shorter overhang decreases bending
- Avoid turning operations with long and/or thin tools.



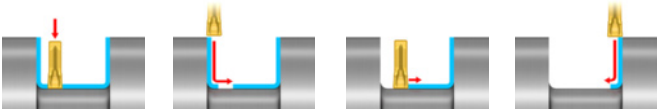
Shorter overhang decreases bending sideways:

$$\delta = \frac{4 \times F \times OH^3}{t^3 \times h}$$

Finishing turning a groove

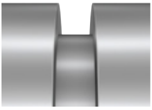
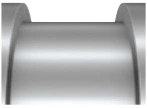
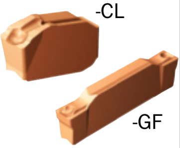
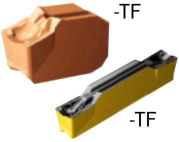
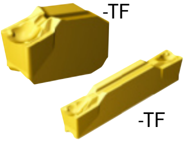
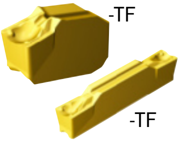
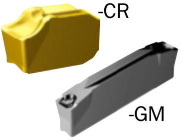
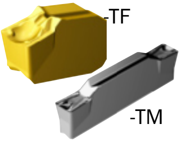
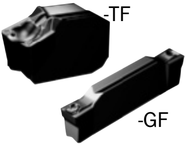
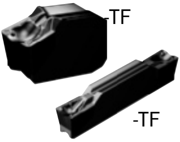
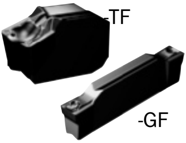
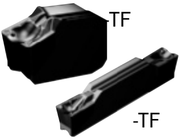
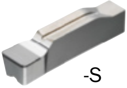
To avoid deflection use a cutting depth larger than the corner radius of the insert.

- Option 1: Use a turning geometry, e.g. -TF
- Option 2: Use a profiling geometry, e.g. -RM for grooves with large radii
- Recommended axial and radial cutting depth 0.5–1.0 mm (0.02–0.04 inch).



Geometry and grade

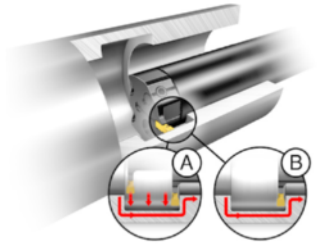
First choice for grooving

ISO	 Grooving	 Turning wider grooves
P Steel	 -CL -GF **GC1125**	 -TF -TF **GC1125/4225**
M Stainless steel	 -TF -TF **GC1135/2135**	 -TF -TF **GC1135/2135**
K Cast iron	 -CR -GM **GC1135/3115**	 -TF -TM **GC1135/3115**
N Non-ferrous metals	 -TF -GF **GC1105**	 -TF -TF **GC1105**
S HRSA	 -TF -GF **GC1105**	 -TF -TF **GC1105**
H Hardened steel	 -S **CB7015**	 -S **CB7015**

For external grooving, tools with high precision coolant is first choice.

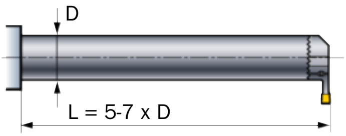
Application tips for internal grooving

Chip evacuation

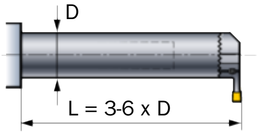


- Start at bottom of hole, machine outwards to steer chip out from hole
- Coolant with high flow improves chip control and evacuation
- A smaller bar improves chip evacuation but reduces stability
- Use plunge turning (B) for best chip control and stability
- Use light cutting geometries like -GF or -TF
- Use smaller insert width and corner radii for lower cutting force.

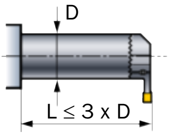
For overhang 5-7 x D use carbide reinforced dampened bars.



For overhang 3-6 x D use dampened or carbide bars.

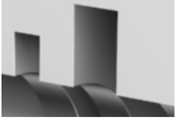
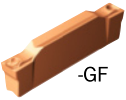
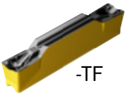
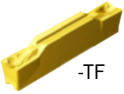
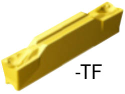
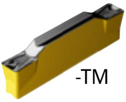
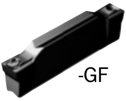
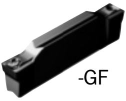
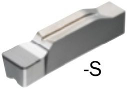
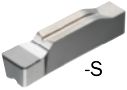


For overhang below 3 x D use steel bars.



Geometry and grade

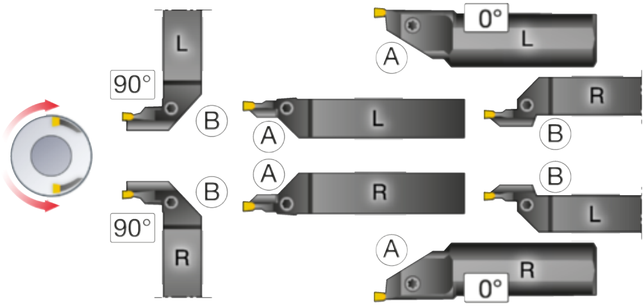
First choice for internal grooving

ISO	 Grooving	 Turning wider grooves
P Steel	 -GF **GC1125**	 -TF **GC4225**
M Stainless steel	 -TF **GC2135**	 -TF **GC2135**
K Cast iron	 -GM **GC4225**	 -TM **GC4225**
N Non-ferrous metals	 -GF **GC1105**	 -TF **GC1105**
S HRSA	 -GF **GC1105**	 -TF **GC1105**
H Hardened steel	 -S **CB7015**	 -S **CB7015**

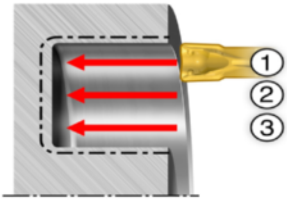
Application tips for face grooving

Choice of tool

Tools curved to fit a range of grooves.



Start on outside, work inwards.

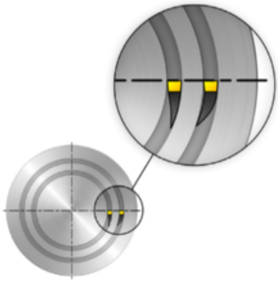


The groove can always be widened by overlapping cuts (or side turning) as long as the first cut is within the diameter range of the tool.

Use the tool for the biggest diameter that fits your groove.

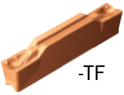
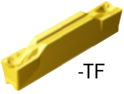
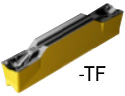
A tool for a bigger diameter is less curved and hence more stable.

- Larger diameter gives improved chip control and better stability. For wider grooves – use side turning for improved chip control
- Always use a tool with shortest possible cutting depth.

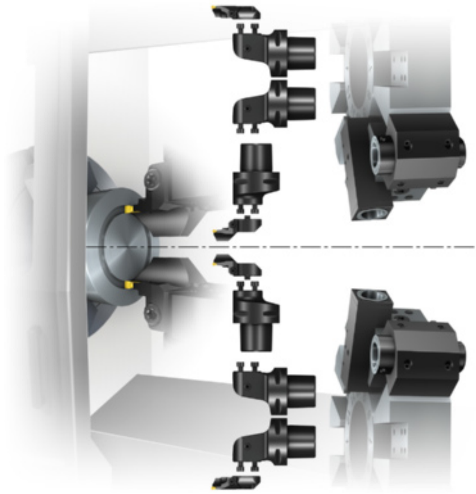


Geometry and grade

First choice for face grooving

ISO	 Face grooving
P Steel	 -TF **GC1125**
M Stainless steel	 -TF **GC2135**
K Cast iron	 -TF **GC4225**

ISO	 Face grooving
N Non-ferrous metals	 -TF **H13A**
S HRSA	 -TF **GC1105**
H Hardened steel	 -S **CB7015**

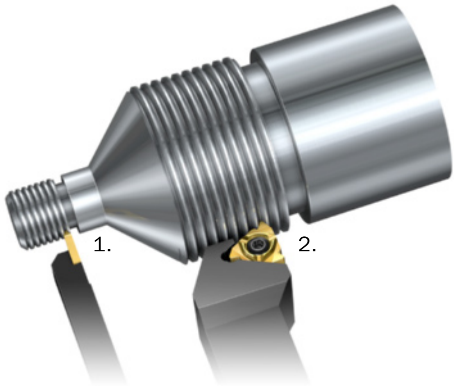


Build your modular grooving tool at **www.tool-builder.com**

Threading

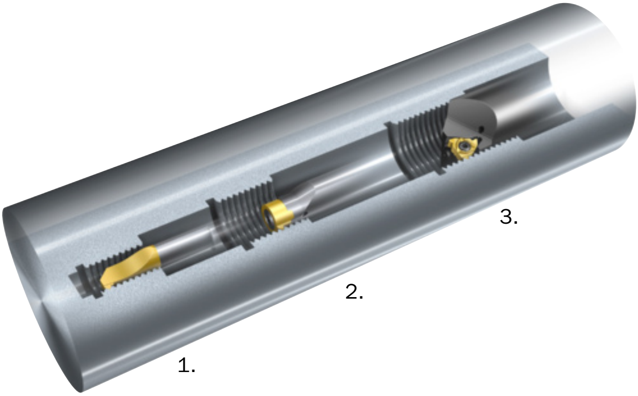
External, different systems

1. CoroCut® XS
   Pitch area 0.2–2 mm
2. CoroThread® 266
   Pitch area 0.5–8 mm, 32–3 t.p.i



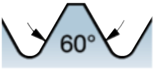
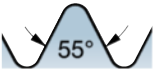
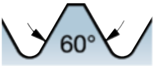
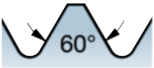
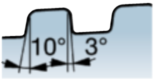
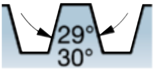
Internal, different systems

1. CoroTurn® XS
   Pitch area 0.5–3 mm, 32-16 t.p.i.
   DMIN Ø4 mm (0.157 inch)
2. CoroCut® MB
   Pitch area 0.5–3 mm, 32-8 t.p.i.
   DMIN Ø10 mm (0.393 inch)
3. CoroThread® 266
   Pitch area 0.5–8 mm, 32-3 t.p.i.
   DMIN Ø12 mm (0.472 inch)



Thread forms

Sandvik Coromant standard assortment

Application	Thread form	Thread type
Connecting General usage		ISO metric, American UN
Pipe threads	 	Whitworth, British Standard (BSPT), American National, Pipe Threads, NPT, NPTF
Food and fire		Round DIN 405
Aerospace		MJ, UNJ
Oil and gas		API Rounded, API Buttress, VAM
Motion General usage		Trapezoidal, ACME, Stub ACME

CoroThread® 266

- First choice tooling system for thread turning
- Guide-rail interface between the insert and tip seat eliminates insert movement caused by cutting force variation
- CoroThread® 266 provides accurate and repeatable thread profile as a result of rigid insert stability.

Tool feed direction

A thread can be produced in a number of ways. The spindle can rotate clockwise or anticlockwise, with the tool fed towards or away from the chuck. The thread turning tool can also be used in normal or upside-down position (the latter helps to remove chips).

- Most common set-up marked with green (below).

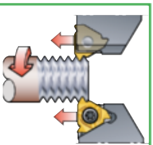
Working away from the chuck (pull threading)

Using right-hand tools for left-hand threads (and vice-versa) enables cost savings through tool inventory reduction.

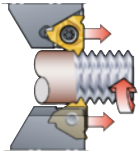
- Negative shim must be used in set-up marked with red (below).

External

Right hand threads

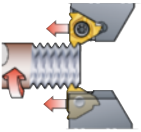


Right hand tool/insert

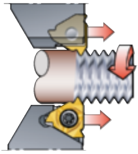


Right hand tool/insert

Left hand threads



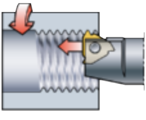
Left hand tool/insert



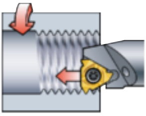
Left hand tool/insert

Internal

Right hand threads

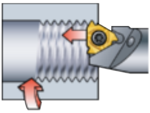


Right hand tool/insert

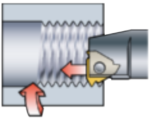


Right hand tool/insert

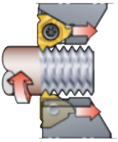
Left hand threads



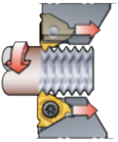
Left hand tool/insert



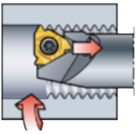
Left hand tool/insert



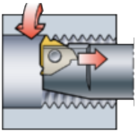
Left hand tool/insert



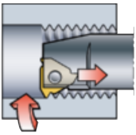
Right hand tool/insert



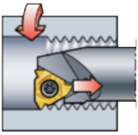
Left hand tool/insert



Right hand tool/insert



Left hand tool/insert

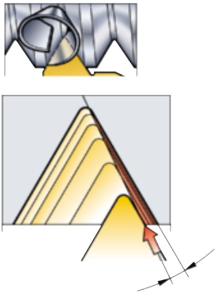


Right hand tool/insert

A negative shim must be used.

Infeed methods

Modified flank infeed

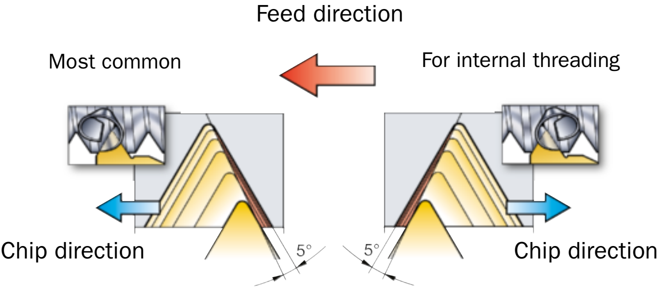


The modified flank infeed is the first choice method giving longest tool life and best chip control.
Most CNC machines have dedicated threading cycles.
Example:

- G92, G76, G71, G33 and G32
- For flank infeed it can be G76, X48.0, Z-30.0, **B57** (Infeed angle), D05 etc.

- Chip is generated only on one side of the insert giving excellent chip control
- Fewer passes needed as less heat is transferred to the insert
- Use 1-5° infeed angle.

Opposite flank infeed



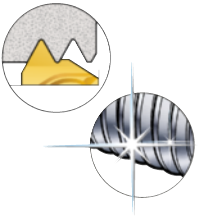
- Insert can cut using both flanks – the chip can be steered in both directions depending what flank is used
- Improved chip control
- Helps to ensure continuous, trouble-free machining, free from unplanned stoppages.

Radial and incremental are other often used methods.

Type of inserts

Full profile insert

First choice



Advantages:
- The full thread profile is cut by the insert
- Root and the crest is controlled by the insert
- No deburring required
- Use 0.05–0.07 mm (0.002–0.003 inch) for stock.

Disadvantages:
- Each insert can only cut one pitch.

V-profile insert

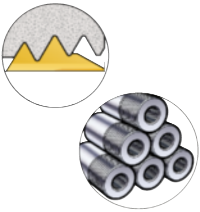
Flexible

Advantages:
- Flexibility, one insert for several pitches
- Minimum tool inventory.

Disadvantages:
- The outer/inner diameter must be turned to the right diameter prior to threading
- Burr formation
- Insert nose radius is smaller to cover the range of pitches which reduces tool life.

Multi-point insert

Productive

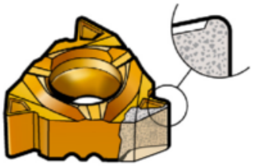


Advantages:
- Similar to full profile insert, two pointed give double productivity etc.
- Very high productive rate
- Double tool life.

Disadvantages:
- Need stable conditions due to increased cutting forces
- Need sufficient room behind the last thread to clear the last tooth of the insert, generating a full thread.

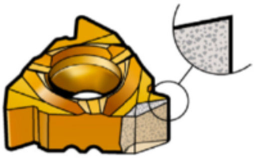
Geometry

Geometry A



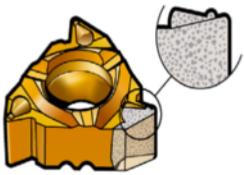
- Edge rounded cutting edge for safe and consistent tool life
- Full profile and V-profile
- Good chip control and edge security.

Geometry F



- Sharp cutting edge
- Clean cuts in sticky or work-hardening materials
- Low cutting forces and good surface finish
- Reduced built-up edge.

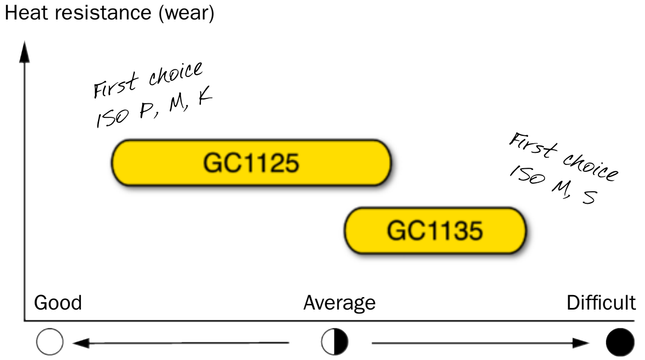
Geometry C



- Chip breaking
- Optimized for low carbon and low alloyed steels
- Maximum chip control, minimum supervision required
- High security for all threading, particularly internal
- High cutting forces
- To be used with 1° modified flank infeed only.

Grade

The insert grade is primarily selected according to:
- Component material
- Machine (stability, e.g. good, average or difficult).



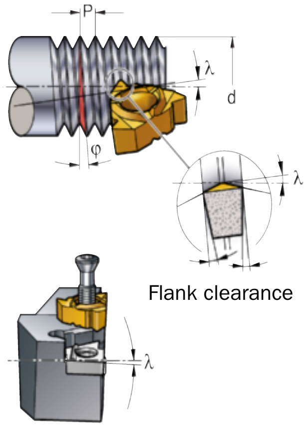
Use GC1125 grade if higher need of heat resistance, because of higher cutting speed and longer engagement time.

Use GC1135 grade for secure machining.

H13A and CB7015 for ISO N and H material.

Flank clearance

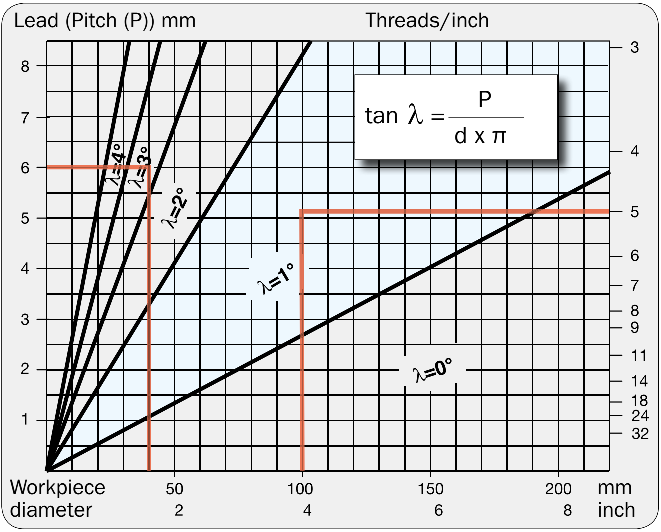
- The helix angle, φ, is depending on and related to the diameter (d) and pitch (P)
- By changing the shim, the flank clearance of the insert is adjusted
- The angle of inclination is lambda, λ. The most common angle is 1° which is standard shim in the tool holder.



Flank clearance

Shim

- Need to be adjusted to the actual thread pitch and diameter
- Available shims -2° to 4° (1° steps)
- Negative inclination shims are available when turning left-hand threads with right-hand tools and vice versa (pull threading).



Example, for a pitch of:

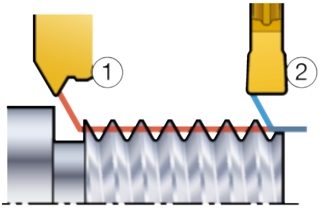
- 6 mm and workpiece Ø40 mm, a 3° shim is required
- 5 threads per inch and workpiece Ø4 inches, a 1° shim is required.

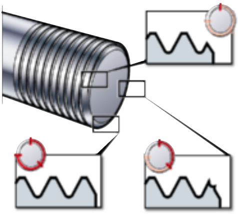
Application tips

Thread deburring

Burrs tend to form at the start of a thread before the insert creates the full profile

- Make the threading in normal ways (1)
- Deburring (2) is achieved with standard turning tools. Use thread cycle for the first 2/3 revolution
- Correct positioning of the deburring insert is important.



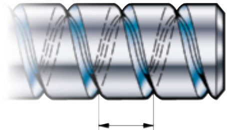


Multi-start threads

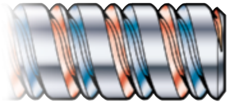
Threads with two or more parallel thread grooves require two or more starts. The lead of this type of thread will then be twice that of a single-start screw.

Important is to use the right shim.

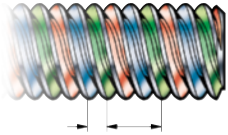
First threading groove

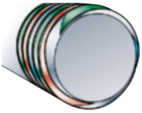


Second threading groove



Third threading groove





A multi-start thread with 3 starts

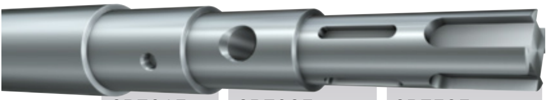
Advanced materials

Hard part turning with CBN inserts

Using a very broad definition, hard part turning (HPT) refers to hardened steels at 55 HRc and greater. Many different types of steel (carbon steels, alloy steels, tool steels, bearing steels etc.) can achieve such a high hardness. HPT is usually a finishing or semi-finishing process with high dimensional accuracy and surface quality requirements.

A CBN insert can withstand the high cutting temperatures and forces and still retain its cutting edge. This is why CBN delivers long, consistent tool life and produces components with excellent surface finish.

Sandvik Coromant offers a comprehensive program of unique CBN products for finish turning, grooving and threading of hardened steels.



Grade selection	CB7015	CB7025	CB7525
Cutting speed	■■■■■	■■■■□	■■□□□
Toughness demand	■■□□□	■■■□□	■■■■■
First choice Edge preparation: Negative insert 	S01030 S0330	S01030 S0330	T01020 T0320
First choice Edge preparation: Positive insert 	S01020 S0320	S01020 S0320	T01020 T0320

Why hard part turning?

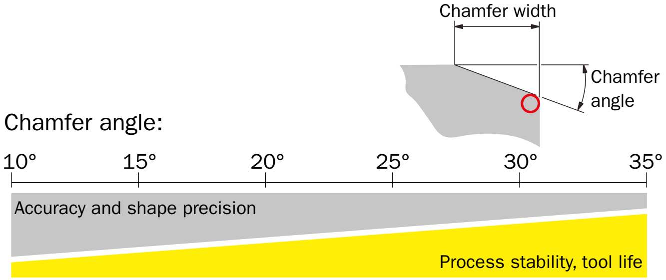
- High quality
- Reduced production time per component
- Process flexibility
- Lower machine investment
- Reduced energy requirements
- Possibility to eliminate coolant
- Easier swarf handling
- Possibility to recycle chips.

Application tips

Chamfer size

A wide chamfer spreads the cutting forces over a larger area providing a more robust cutting edge, allowing for higher feed rates. Use a large chamfer when process stability and consistent tool life are the most important factors.

If surface finish and dimensional accuracy are the main requirements a small chamfer will provide better results. Cutting forces and temperature will be reduced and there will be less vibration.



The cutting edge

Use the largest nose radius allowed, based on your process requirements:

- Small nose radius, e.g. 0.2, 0.4 mm (1/128, 1/64 inch) provides good chipbreaking
- A large nose radius provides better surface, greater edge strength and therefore extended tool life.

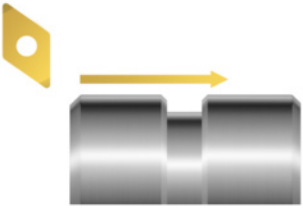
Wiper inserts provide two possibilities for process improvements:

- Improved surface finish with conventional cutting conditions
- Maintained surface finish at higher feed rate.

Xcel inserts allow the highest feed rates, 0.3–0.5 mm/r (0.012–0.020 inch/r), while still producing high quality surface finish.

Prepare the component in the soft state

- Avoid burrs
- Keep close dimensional tolerances
- Use chamfer and make radii in the soft state.



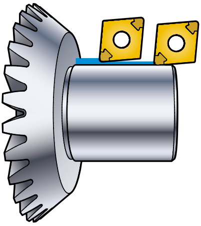
Maintain a rigid machine set-up

- Use wide clamping jaws (no hardened jaws)
- Use Coromant Capto®
- Tool holders must be in excellent condition.

Two-cut strategy

A two-cut strategy is likely to be the best option:

- When the machine set-up is unstable
- If there is any inconsistency in the component
- If a very high final tolerance or surface quality is required.



Use of coolant

Dry cutting is one of the most significant advantages of hard part turning. However, there are some situations where coolant is required, for example:

- To facilitate chip breaking
- To control the thermal stability of the workpiece
- When machining big components (to remove heat).

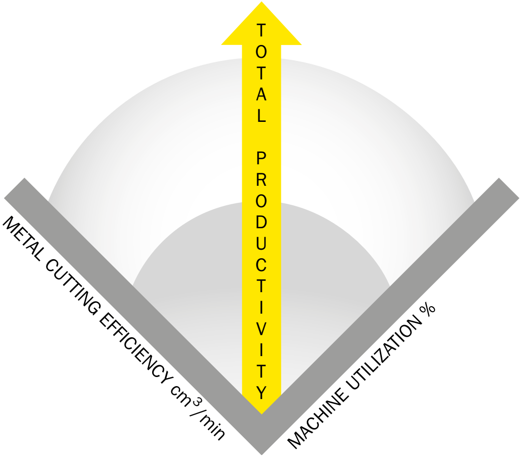
The coolant must always be applied as a consistent flow throughout the entire cutting length.

Additional information

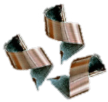
Winning the productivity race

With productivity, much like in a car race, both having high speed and keeping stops few and short are important. Understanding your situation and offering productivity-enhancing solutions based on your challenges is where Sandvik Coromant excels.

Total productivity can be enhanced by increasing metal cutting efficiency or machine utilization. Or in some situations – both.



Metal cutting efficiency – go fast!



Metal cutting efficiency is all about speed and high chip removal rate. Still, increasing speed with the downside of frequent stops is not efficient.

To reach high productivity, you need high-performance grades, rapid methods and to not let vibration slow you down.

For high speed: GC4325, GC4315 and Silent Tools™.

Machine utilization – more machining time!

Keeping planned stops short is a true productivity booster. Manual tool change is time-consuming and sometimes really tricky, especially when using machines with limited space or when tool position is not repeatable. In the worst case, it can take up to 10 minutes to get the tool in place and the position right.

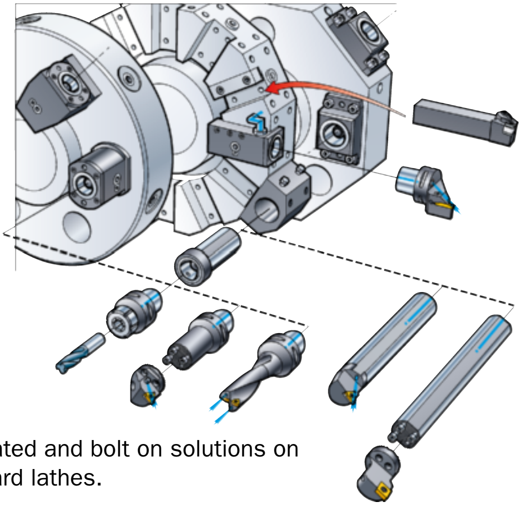
For the pit stop: Quick change with Coromant Capto® and QS™ holding system.

Unplanned stops are real time thieves. A flat tire destroys your chances at winning in a car race. Similarly, chip problems and tool breakage can really damage efficiency in a workshop.

To keep you on the track: GC4325, GC4315, CoroTurn® HP and Silent Tools™.

Quick change

Quick change clamping units will optimize your machine utilization by reducing both set-up time and tool change time significantly.



Integrated and bolt on solutions on standard lathes.

Coromant Capto® directly integrated in the spindle increases stability and versatility. The same tools can thus be used in the entire workshop, providing flexibility, optimal rigidity and minimized tool inventory.

The modularity function means less need for expensive special tools with long delivery times:

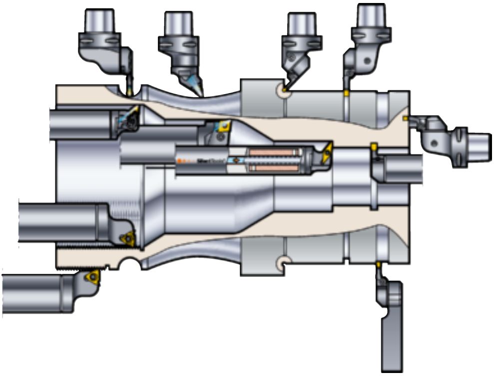
- Available in six sizes: C3-C10, diameter 32, 40, 50, 63, 80, and 100 mm.

Through-tool delivery of high pressure coolant, from machine to cutting edge:

- Up to 400 bar (5802 psi) together with Coromant Capto® HP clamping units.

CoroTurn® SL

CoroTurn® SL is a universal modular system of boring bars, Coromant Capto® adaptors and exchangeable cutting heads intended to build customized tools for different types of machining applications.

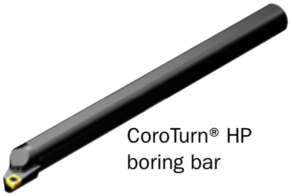


- For general turning, parting and grooving and threading
- Robust, serrated interface between adaptor and cutting head is comparable with a solid tool in performance, in regard to vibration and deflection
- Cutting heads with CoroTurn® HP
- Solid steel, dampened Silent Tools™ and dampened reinforced carbide adaptors
- Quick change in combination with Coromant Capto®
- SL cutting heads combined with CoroTurn® SL adaptors makes it possible to build a large variety of tool combinations
- Build your own modular tool at **www.tool-builder.com**.

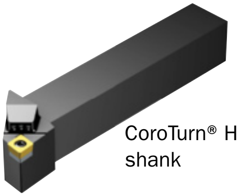
CoroTurn® HP

CoroTurn HP is a programme of tool holders with high precision coolant.

The tool holders have fixed nozzles for improved chip control, process security and high productivity giving extended tool life.



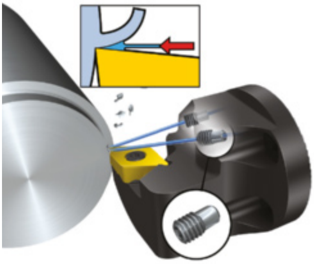
CoroTurn® HP
boring bar



CoroTurn® HP
shank

- Boring bars for internal turning
- Shanks for fine to medium turning
- Quick change in combination with Coromant Capto®
- Increased tool life due to dedicated inserts for T-Max® P and CoroTurn® 107.

- Integrated nozzles for exact coolant jets
- Coolant pressure range: 5-275 bar (75-3990 psi)
- Number of nozzles: 1-3.

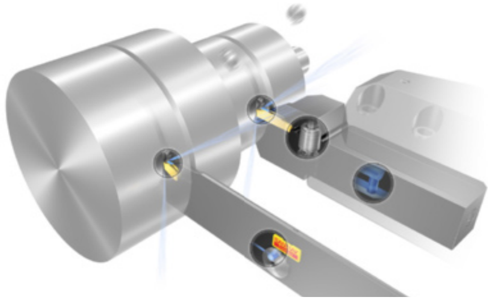


High precision nozzles direct coolant exactly at the cutting zone.

Parting and grooving – plug and play coolant

CoroCut® QD and CoroCut® 1-2, parting blades and shank tools are available with plug and play adaptors for easy coolant connection

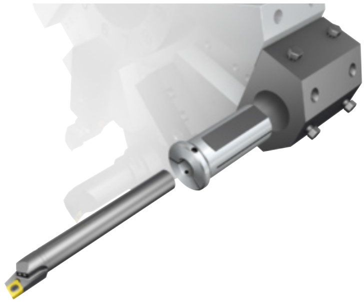
- High precision over and under coolant for improved chip control, surface finish and tool life
- No connection hoses or pipes needed
- Adaptors available for most machine types.



EasyFix™

EasyFix sleeves reduce set-up time when using cylindrical boring bars. A spring plunger guarantees the correct centre height.

- Existing coolant supply system could be used
- A metallic sealing offers good performance for high coolant pressure
- EasyFix sleeves fits all cylindrical boring bars.



Silent Tools™

Silent Tools adaptors minimize vibration through a dampener inside the tool maintaining good productivity and close tolerances even at long overhangs.

The adaptor can be combined with different CoroTurn® SL cutting heads.

Maximum recommended overhang:

Bar type	Turning	Grooving	Threading
Steel	4 x DMM	3 x DMM	3 x DMM
Carbide	6 x DMM	6 x DMM	6 x DMM
Steel dampened	10 x DMM	5 x DMM*	5 x DMM*
Carbide rein-forced dampened	14 x DMM	7 x DMM	7 x DMM

*570-4C bars

Overhangs up to 10 x DMM are usually solved by applying a steel dampened boring bar to accomplish a sufficient process.

Overhangs over 10 x DMM require a carbide reinforced dampened boring bar to reduce radial deflection and vibration.

Internal turning is very sensitive to vibration. Minimize the tool overhang and select the largest possible bar size in order to obtain the best possible stability and accuracy.

For internal turning with steel dampened boring bars, the first choice is bars of type 570-3C.

For grooving and threading where the radial forces are higher than in turning, the recommended bar type is 570-4C.

SOUTHERN CUTTING TECHNOLOGY

SANDVIK Coromant

Metal Cutting E-Learning

A comprehensive course accessible anytime, anywhere. With this course, you can enhance your competency in metal cutting, improve your productivitiy and profitability.

This course covers theory and application within:

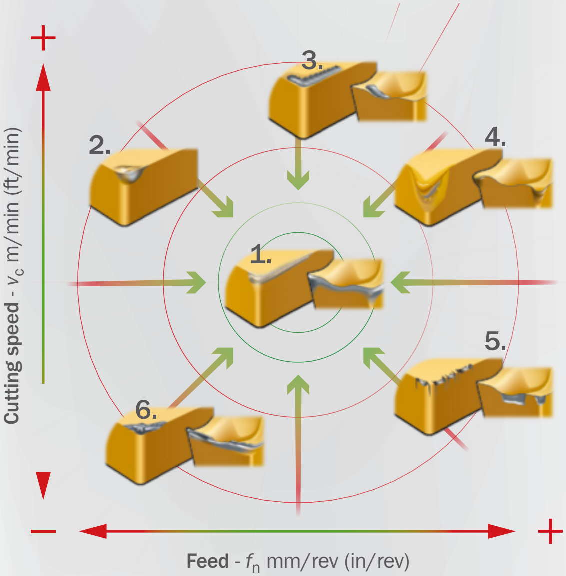
- Machinability
- Turning
- Parting & Grooving
- Threading
- Milling
- Tool holding
- Drilling
- Boring
- Tool wear & maintenance

Scan QR code to sign up the course.

For more info, please contact your local SCT representatives.

Unit 3C, 200 Maces Road, Bromley,
Christchurch, New Zealand.
Phone: +64 3 338 4169
Email: admin@southerncut.co.nz
https://www.southerncut.co.nz/

Wear optimization

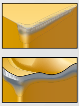


1. **Flank wear** (abrasive) — Preferable wear for predictable tool life
2. **Plastic deformation** (impression)
3. **Crater wear**
4. **Plastic deformation** (depression)
5. **Chipping**
6. **Built-up edge**

Information about wear types on backside ▶

Wear types

1. Excessive flank wear



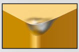
Cause

- Cutting speed too high
- Insufficient wear resistance
- Too tough grade
- Lack of coolant supply

Solution

- Reduce cutting speed
- Select a more wear resistant grade
- Improve coolant supply

2. Plastic deformation (impression)



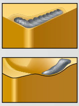
Cause

- Cutting temperature too high
- Lack of coolant supply

Solution

- Reduce cutting speed (or feed)
- Select a more wear resistant grade
- Improve coolant supply

3. Crater wear



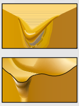
Cause

- Too high cutting speed and/or feed
- Too tough grade

Solution

- Reduce cutting speed or feed
- Select a positive insert geometry
- Select a more wear resistant grade

4. Plastic deformation (depression)



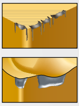
Cause

- Cutting temperature too high
- Lack of coolant supply

Solution

- Reduce feed (or cutting speed)
- Select a more wear resistant grade
- Improve coolant supply

5. Chipping



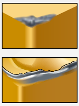
Cause

- Unstable conditions
- Too hard grade
- Too weak geometry

Solution

- Select a tougher grade
- Choose a geometry for higher feed area
- Reduce overhang
- Check centre height

6. Built-up edge



Cause

- Too low cutting temperature
- Adhesive workpiece material

Solution

- Increase cutting speed or feed
- Select a sharper edge geometry