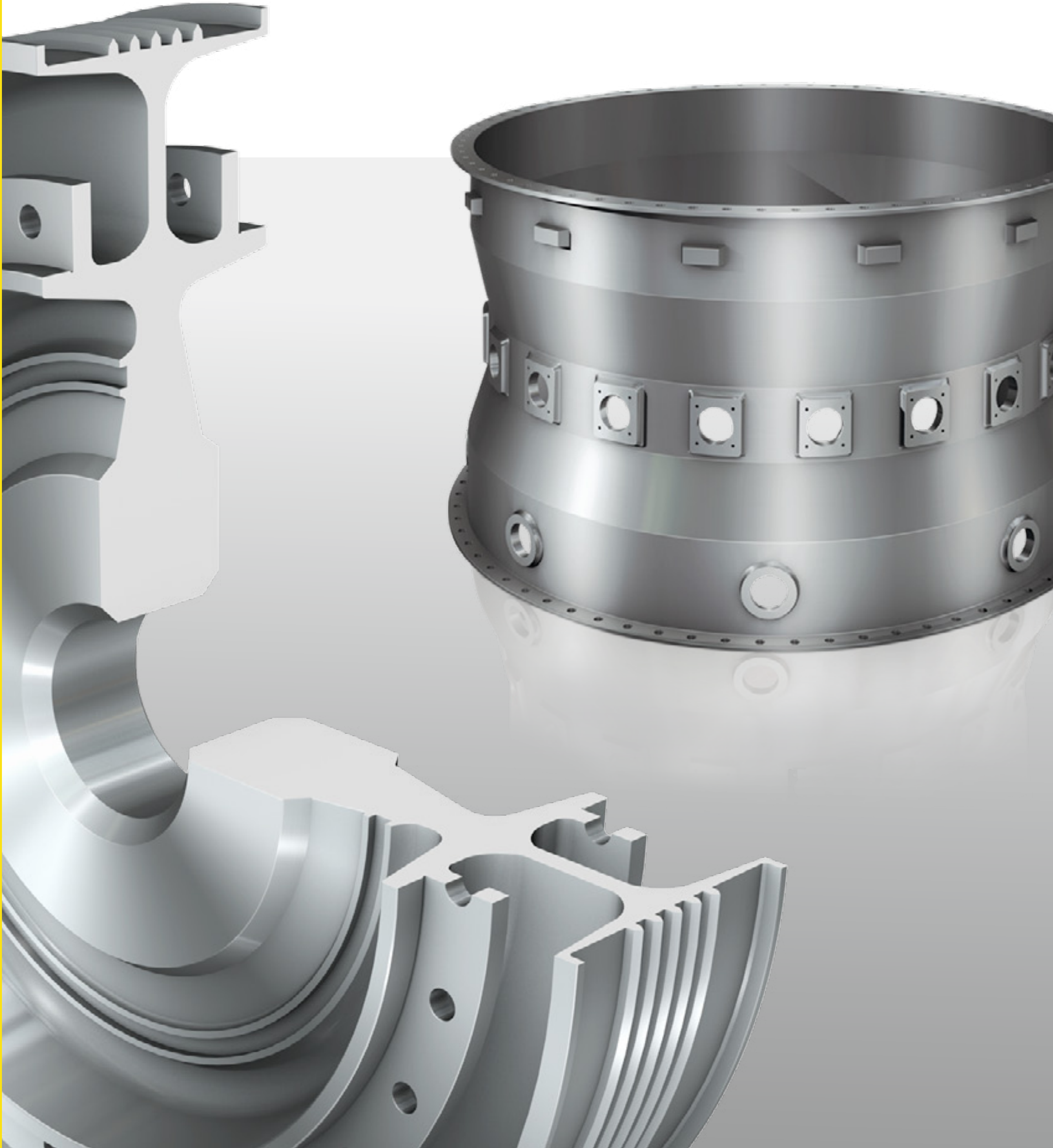
APPLICATION GUIDE

Heat resistant super alloys



More information

Useful information and application techniques can be found in our catalogues, handbooks and application guides, such as PluraGuide. CoroGuide web is an internet-based catalogue including a cutting data module (also available on CD) where you can find cutting data recommendations for your specific application.

Visit our websites for the latest news!

www.sandvik.coromant.com

www.aero-knowledge.com

Contents

Introduction

This application guide concentrates on optimizing machining of heat resistant super alloys (HRSA).

For one of the most challenging material groups to machine, optimized tools are naturally a prerequisite, but equally important is how to apply them.

We will guide you through the most common materials and machining applications. We aim to give you application and process recommendations that will help you use our products in the most productive manner with maximum process reliability and component quality.

Our goal is to support customers with complete tooling solutions that meet cost reduction and quality improvment initiatives.

Productivity along with quality and reliability are our focus. When we talk about productivity you will see that we measure this in terms of cm^3/min. It is important to understand the relationship between the combination of speed, feed and depth of cut and not just cutting speed alone which is often the most damaging parameter when considering tool life.

1. Heat resistant super alloys – HRSA

Heat resistant super alloys (HRSA) are a family of alloys utilised in various industry segments:

Aerospace engine	– combustion and turbine sections.
Stationary gas turbines	– combustion and turbine sections.
Oil and gas	– marine applications.
Medical	– joint implants.

The properties which make them attractive are:

- Retension of strength and hardness at high temperatures.
- Corrosion resistance.

Aerospace engine

Stationary gas turbines

Oil and gas

Medical

Alloy groups

HRSA materials fall into three groups: nickel-based, iron-based and cobalt-based alloys. The physical properties and machining behaviour of each varies considerably, due both to the chemical nature of the alloy and the precise metallurgical processing it receives during manufacture. Whether the metal is annealed or aged is particularly influential on the subsequent machining properties.

Nickel-based are the most widely used, and currently constitute over 50% of the weight of advanced aircraft engines. The trend is that this will increase in new engines in the future.

Common types include:

- Inconel 718, Waspaloy, Udimet 720 – precipitation hardened
- Inconel 625 – solution strengthened (not hardenable)

Iron-based have been developed from austenitic stainless steels. Some have very low thermal expansion coefficients (such as Incoloy 909) which make them especially suited for shafts, rings, and casings. However, they have the poorest hot strength properties of the three groups.

Common types:

- Inconel 909
- A286
- Greek Ascoloy

Cobalt-based display superior hot corrosion resistance at high temperatures compared to nickel-based alloys. They are more expensive and also more difficult to machine due to their great wearability.

The use in turbines is restricted to combustion parts in the hottest engine areas.

Their main use is seen in surgical implants, which utilise their inherent corrosion resistance.

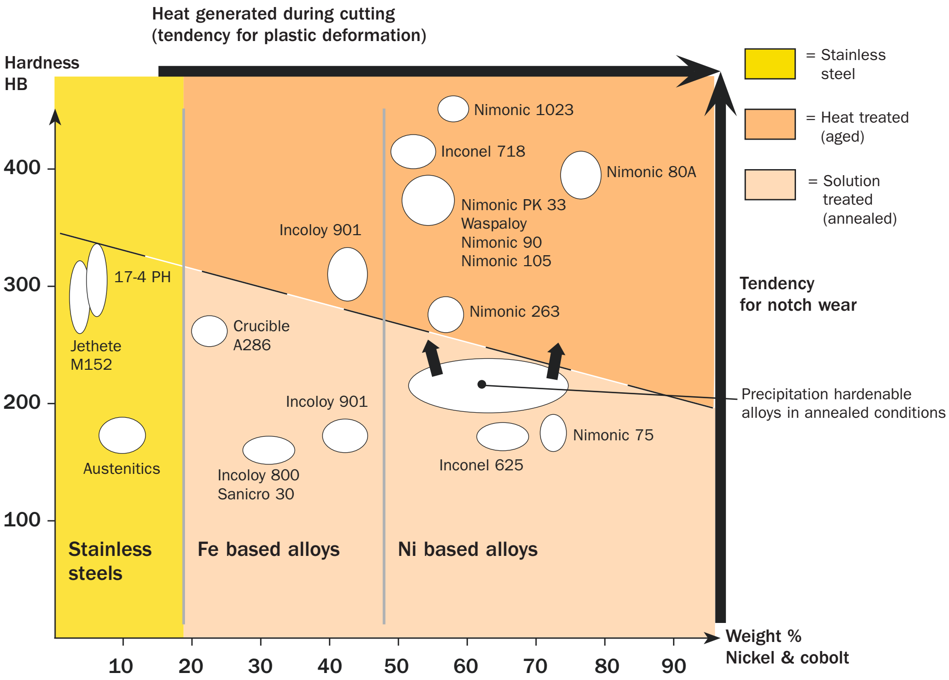
Common types:

- CoCr
- Haynes 25
- Stellite 31

The most common HRSA alloys (see page 120 for the complete list)

Alloy group	Code		Material	Hardness HB Ann.	Hardness HB Aged
Nickel	MC S2.0.Z.AN	CMC 20.2	**Inconell 718**		425
			Inconell 706		285
			Inconell 625	200	
			Hastelloy S		
			Hastelloy X	160	
			Nimonic PK33		350
			Udimet 720		
			Waspaloy		
Iron	MC P5.0.Z.AN	CMC 05.3	**Greek Ascoloy**		300
	MC M1.0.Z.PH	CMC 05.4	**A286**		300
	MC S2.0.Z.AN	CMC 20.21	**Incoloy 909**		
Cobalt	MC S3.0.Z.AG	CMC 20.3	**Haynes 25**		
			Stellite 21	280	340
			Stellite 31		

With such a wide spread of materials under the generic heading of HRSA the machining behaviour can vary greatly even within the same alloy group. In fact the same material can have numerous machining recommendations.



Machinability/raw material condition

Heat treatment

Annealing	– heating to controlled temperature then cooling at controlled rate.	<30HRC
Solution treatment	– heating followed by rapid cooling	<30HRC
Ageing	– slow cooling after solution treatment	up to 48HRC

The state of heat treatment affects the hardness of the component and hence the wear mechanisms. The formation of the chip is a good indicator of the hardness – with hard materials it is easier to break the chip.

Hardened materials have increased cutting temperatures and show a tendency to notching of the cutting edge at the depth of cut. The combination of a low entering angle and a hard substrate with a coating offering a heat barrier is required.

Softer materials machine similarly to the stainless steel family.

Insert grades with greater toughness and reduced hot hardness – resistance to high temperatures – are required due to reduced cutting temperatures and increased chip hammering. Here, damage to areas outside the actual cutting edge is caused by the chip breaking against the insert.

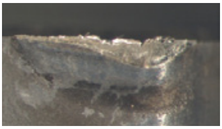
Comparison of wear depending upon material hardness and insert grade

CNMX 1204A1-SM – v_c 50 m/min, f_n 0.25 mm/r, a_p 1.5 mm

Hard material

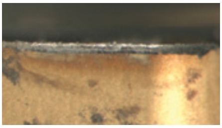
GC1105

6 min



S05F

12 min



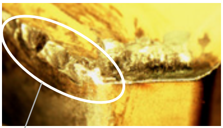
Soft material

GC1105

7 min

S05F

3.5 min



Chip hammering

Raw material production method

Depending upon the size, shape and strength requirements of the component, various production methods for the blank material will be adopted.

The production method varies the machinability of the material and will change the wear characteristics.

Material	Components	Advantage/suitability	Machinability
Forging	large	high strength	medium
Casting	complex shape	low strength	poor
Bar stock	<200 mm diameter	availability/strength	good

Each of these raw material types directly affects the alloy's micro structure, and so also affects the subsequent machining behaviour:

Forged materials have a finer grain size than in castings, which improves the strength and grain flow of the component. When machining forgings, reducing the speed and increasing the feed generally gives the maximum possible metal removal rate with good tool life.

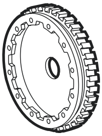
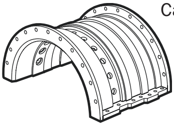
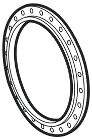
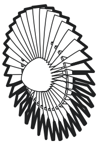
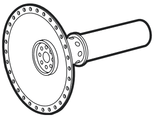
In castings the opposite applies, and applying low feeds (0.1 mm chip thickness) and higher speeds can be beneficial. Castings have poor machinability and tend to be most sensitive to notch wear and abrasive wear. They can be easily identified due to their visibly mottled surface (the 'orange peel' effect).

Bar stock material is the easiest form of raw material to deal with. Notching is not so much of a problem, which allows harder and more wear resistant insert grades to be used than for forgings.

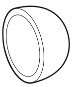
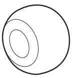
Common component types

Typical HRSA components, and an indication of the different machining methods involved for each include:

Aerospace and gas turbine – nickel based

Component		Turning	Milling	Drilling	Others
	Discs	60%	10%	5%	25%
	Casings	45%	40%	15%	
	Rings	95%		5%	
	Blades Blisks Impellers	10%	50%		40%
	Shafts	70%	5%	25%	

Medical – CoCr

Component		Turning	Milling	Drilling	Others
	Cup	90%		10%	
	Head	90%		10%	

Coolant requirements

Coolant should be applied in all operations excluding milling with ceramics. The volume should be high and well directed.

High pressure coolant HPC (up to 80 bar) shows positive results in terms of tool life and consistency.

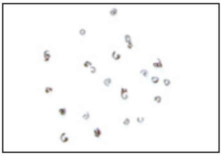
Dedicated HPC-tools with fixed nozzles give parallel laminar jets of coolant with high velocity accurately directed at the right zone between insert and chip.

For milling and drilling, all tools with internal coolant supply can benefit from HPC even if tools prepared for nozzles give higher possibility to use smaller nozzle diameters for high pressure.

- Turning, use at least 20 l/min and a basic pressure of 70 bar.
- Milling and drilling, use at least 50 l/min to accomodate the extra nozzles on the milling cutter and the largest drill diameters.

HPC improves the chip control

CNGG 120408-SGF
v_c 65 m/min, a_p 1.0 mm, f_n 0.2 mm/r
Inconel 718



CoroTurn HP tool, 80 bar

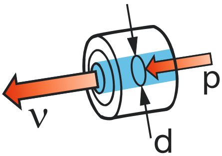
Conventional tool

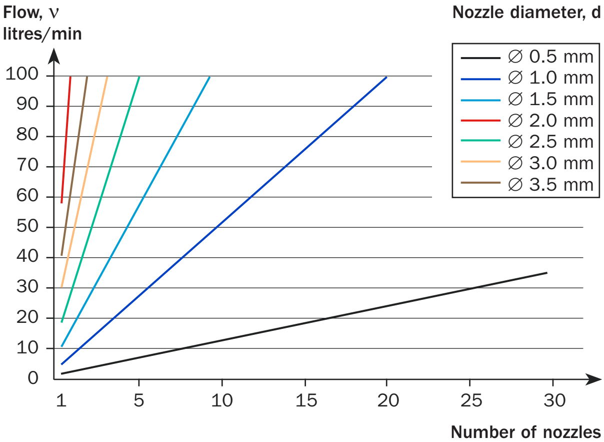
Compared chips made with a CoroTurn HP tool versus a conventional tool and standard coolant pressure.

Flow required for specified nozzle diameter and 80 bar high pressure pump

The pressure (p) hitting the cutting zone is dependent on the number of nozzles, the nozzle diameter (d) and the flow (ν) given from the pump.

A higher flow rate is needed for tools with many outlets or large hole diameter for the coolant.





2. Turning of nickel-based materials
– aerospace engine and stationary gas turbines

Classification of machining stages

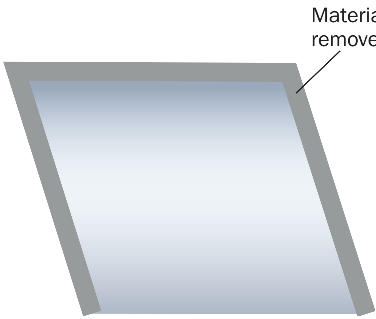
The production cycle for the machining of a HRSA component can be broken into three distinct stages, each with specific demands on the tooling and the machined surface.

FSM – First stage machining – up to 10 mm depth of cut

The forged components often have rough, uneven skin or scale. They are generally machined in the soft condition (the hardness is typically 26 HRC) at the foundry to the basic component shape. Coated carbide grades are used at high feed rates, large depths of cut and low speeds.

Ceramics may also be used with good machine and forging conditions. Here the main priorities are productivity and large stock removal – in the total machining process, up to 80% of the original weight will be machined away, and the bulk of it will be removed during FSM.

The component shapes are simple and standard general turning tool holders can be used. Many critical parts have a ring removed at this stage for material analysis. CoroCut insert in geometry -TF and grade GC1105 is outstanding for this operation.



ISM – Intermediate stage machining – 0.5 to 5 mm depth of cut

At this stage the material is mainly in the final hard/aged condition, (the hardness is typically 36 to 46 HRC) having undergone some form of heat treatment after FSM.

The ISM process involves profiling of the component with varying depths of cut at moderate tolerances, where productivity is important but insert security is equally vital.

In this area, ceramics offer the best productivity where stability allows. Due to the complex shape of the components, ISM may involve a high degree of grooving/ recessing and profiling, which can require a large amount of special tool holders.

LSM – Last stage machining – 0.2 to 1 mm depth of cut

LSM represents the least amount of material removal, but imposes the highest demands on surface quality. For this critical stage of production, the tools, tool paths and cutting data are sometimes certified by end producers of the aerospace engine.

These parts should be machined with cemented carbide grades, to ensure a minimal deformation zone and correct residual stresses in the finished component surface.

Factors which affect the residual stresses most are:

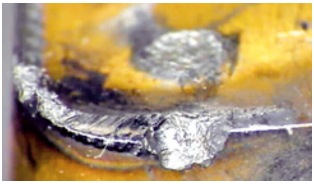
Speed – not above 60 m/min for critical parts.

Tool wear – maximum 0.2 mm. Use S05F for best tool life.

Chip thickness (feed/radius combination) – too low chip thickness (below 0.1 mm) generates more heat and work-harden the surface. For round inserts and large radius increase the feed. (see page 19).

Typical wear mechanisms

With carbide grades two wear mechanisms dominate – plastic deformation and notch wear. It is important to identify which is the most prominent before selecting the correct grade and strategy.



Notch wear on main cutting edge is a mechanical wear which is concentrated at the depth of cut. The extent of notch wear is directly related to:

	Least notch ⟷		Most notch
Entering/lead angle	Round inserts		C/DNMG 95 degree
Depth of cut	Below nose radius		Above nose radius
Geometry	Positive		Negative
Material hardness	Soft condition		Hardened
Material condition	Bar stock	Forged	Cast
Grade	PVD fine grain carbide (GC1115, GC1105)	CVD (S05F)	Ceramic (CC6060/CC6065/CC670)

Because of these factors, notching is the critical wear for ISM where the material is hard and the depth of cut is relatively high. To reduce notching, use as small entering angle as possible.

Plastic deformation (PD)/even flank wear – as a result of combined high temperatures and high pressure on the cutting edge. This wear is much more of a grade-specific issue than notch wear, which is more application related. Good wear resistance and hot hardness will reduce the likelihood of plastic deformation.

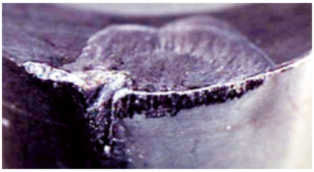
In case of excessive flank wear, use a more wear resistant grade or reduce the cutting speed.

Flank wear resistance Hot hardness ↑		**Turning**	**Grooving**
	Ceramic	**CC6060, CC6065**	
		CC670	**CC670**
	Carbide	**S05F**	
		GC1105	**GC1105**
		GC1115	
		H13A	**H13A**
		GC1125	**GC1125**
		GC2025	**GC2135**
↓ **Bulk toughness**		**GC2035**	**GC1145**

Top slice wear – this type of wear is common for ceramics in HRSA. Small slices of the cutting tool material are lost around the insert's top face. When the flank of the insert is worn, the workpiece pressure against the periphery will be high enough to break small slices away along the edge line.

The recently sliced area will then form a new sharp edge that again cuts well, and the cutting process can continue under these circumstances for a long time without posing a threat to the overall quality, in less sensitive roughing or semi-finishing operations.

In finishing operations, where surface quality and/or burr formation is important, top slicing can be critical. This tendency increases with high feed rates due to increased radial pressure.



Small slices are lost around the insert's top face

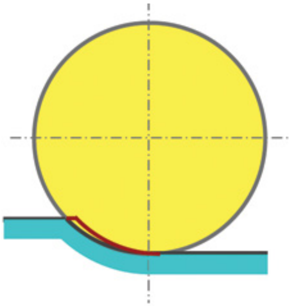
To reduce top slice

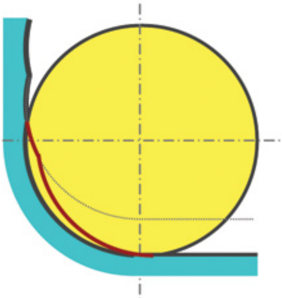
In stable conditions:

- Lower the cutting pressure by reducing the chip area:
  - – feed rate
  - – depth of cut, a_p
  - – arc of engagement
- Use optimized programming techniques
- Use CC670 which is stronger due to whisker reinforcement.

In unstable conditions where top slice is caused by vibrations:

- Reduce engagement angle with programming techniques
- Use CC6065 rather than CC6060



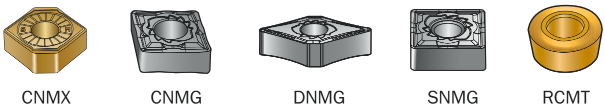


Insert shape selection

Entering angle – κ_r

With a standard C/D/SNMG style insert for roughing, the entering angle is constant regardless of depth of cut.

However, with round inserts the entering angle varies from 0 to 90° depending upon the ratio between depth of cut and diameter.

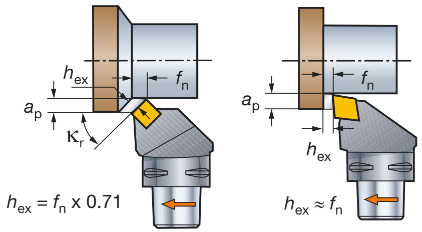


Effect of entering angle

Notch wear on the inserts is the major problem when machining HRSA. The worst notching occurs when the depth of cut is greater than the nose radius, and the entering angle is 90°. (The depth of cut is the influencing factor – with a depth of cut smaller than the nose radius, the effective entering angle is reduced even when the angle on the insert itself is 90°).

By following some general rules the wear can be controlled allowing more productive grades to be used.

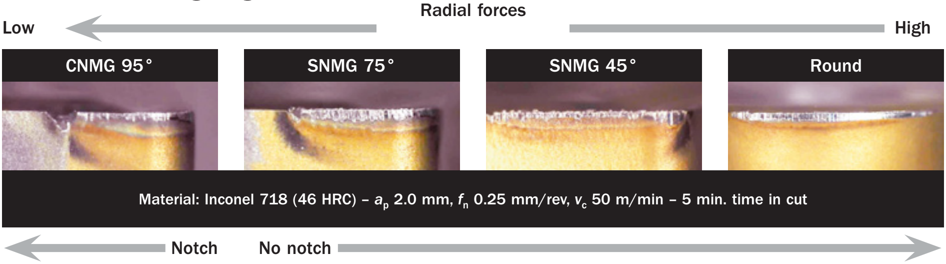
- Use as small entering angle as possible (max 60°, min 25°)
  – eg. SNMG, CNMX where κ_r = 45°.
- Round inserts – use no greater entering angle than 45° or 0.15 x diameter.



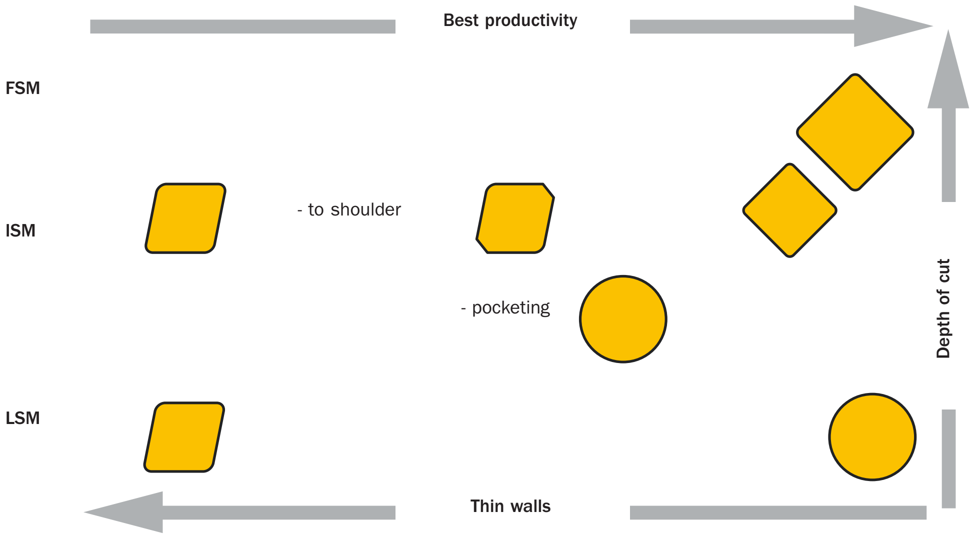
h_{ex} = chip thickness

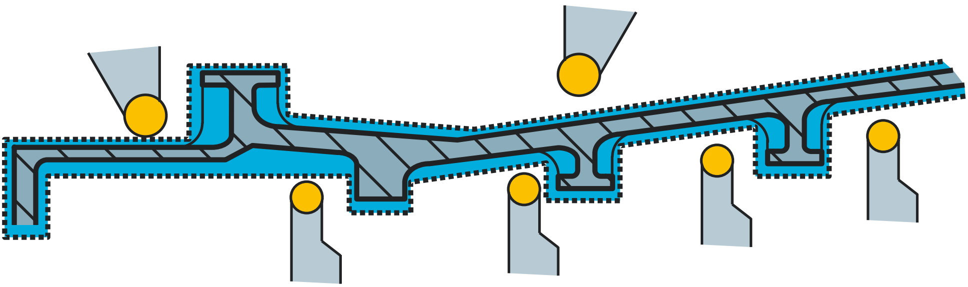
- Ramping – program a varying depth of cut into the cutting operation. This spreads the notching over the whole cutting edge, giving longer tool life and more predictable wear. This method is used predominantly with ceramics, and mainly with round inserts.

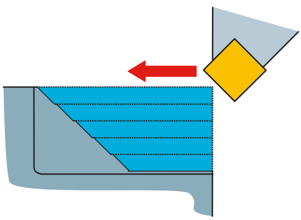
Effect of entering angle on wear mechanism

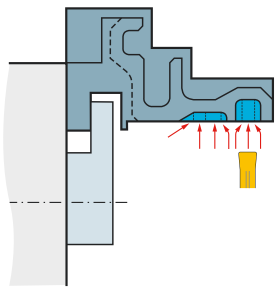


Selecting the right insert for the job



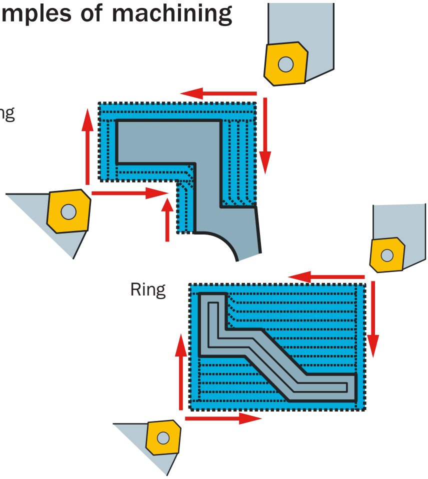






Xcel insert – high productive machining into a corner

Examples of machining



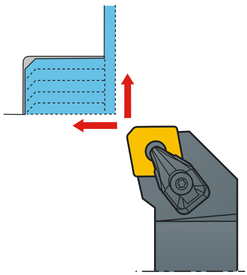
A tooling solution for semi-rough turning into shoulders, combines a host of design advantages combing the benefits of a square and rhomboid insert into one single tool:

- reduced notch wear and increased feeds compared to rhombic inserts.
- reduced radial forces and constant chip thickness compared to round inserts.
- greater accessibility in confined spaces than square inserts, in addition to having the ability to machine in two directions, and provide a known offset point for precise positioning of the cutting edge.

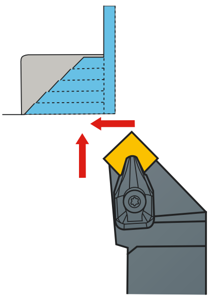
Longer tool life, secure machining, and increased cutting data – all the best possible advantages are available from a single tool.

Two chamfer sizes are available, to suit depths of cut up to 2.7 mm in intermediate stage machining, ISM. They are offered in several grades, all proven performers in ISO S materials.

C-style CNMX-SM



S-style SNMG



Insert code	Max a_p mm
CNMX 1204A1-SM	1.7
CNMX 1204A2-SM	2.7

Grade recommendations – first choice

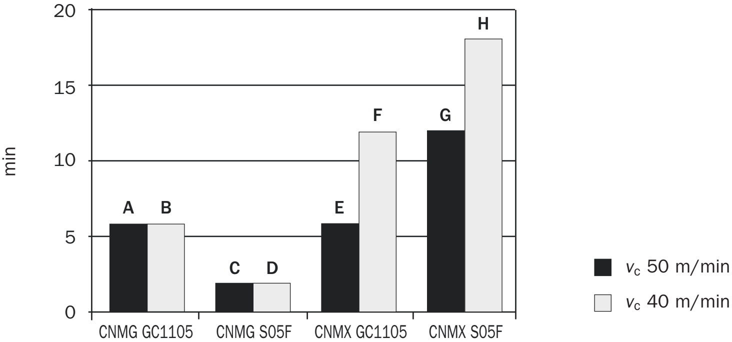
<35 HRC	S05F
>35 HRC	GC1105
Titanium	H13A
Iron based	GC2015

The inserts fit into a standard CNMG holder but require a new shim to accommodate their design.

New shims

5322 234-07 for T-Max P lever design holders
5322 234-08 for CoroTurn RC holders

Tool life test – Inconel 718 (46 HRC) – a_p 1.7 mm, f_n 0.25 mm/r

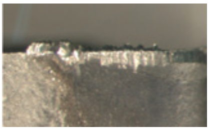


CNMG

■ v_c 50 m/min

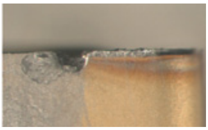
GC1105

A 6 min



S05F

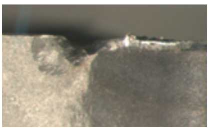
C 2 min



□ v_c 40 m/min

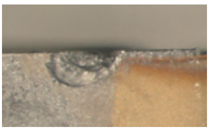
GC1105

B 6 min



S05F

D 2 min

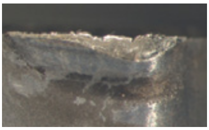


CNMX

■ v_c 50 m/min

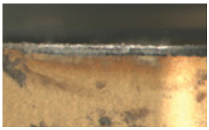
GC1105

E 6 min



S05F

G 12 min



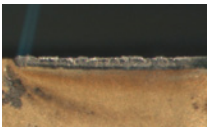
□ v_c 40 m/min

GC1105

F 12 min

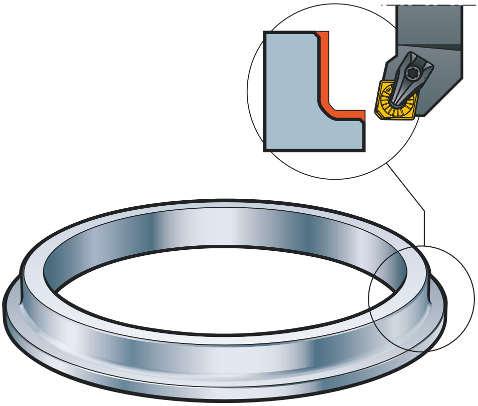
S05F

H 18 min



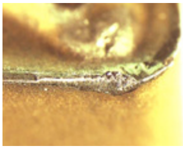
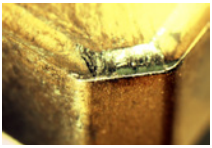
Turning of heat resistant super alloys

Xcel allows productivity to be doubled in nickel-based HRSA, through increased cutting data and a longer-lasting tool life.



Data

Material: NIMONIC PE 16
Ring, diameter 650 mm
Cutting depth: 1.7 mm

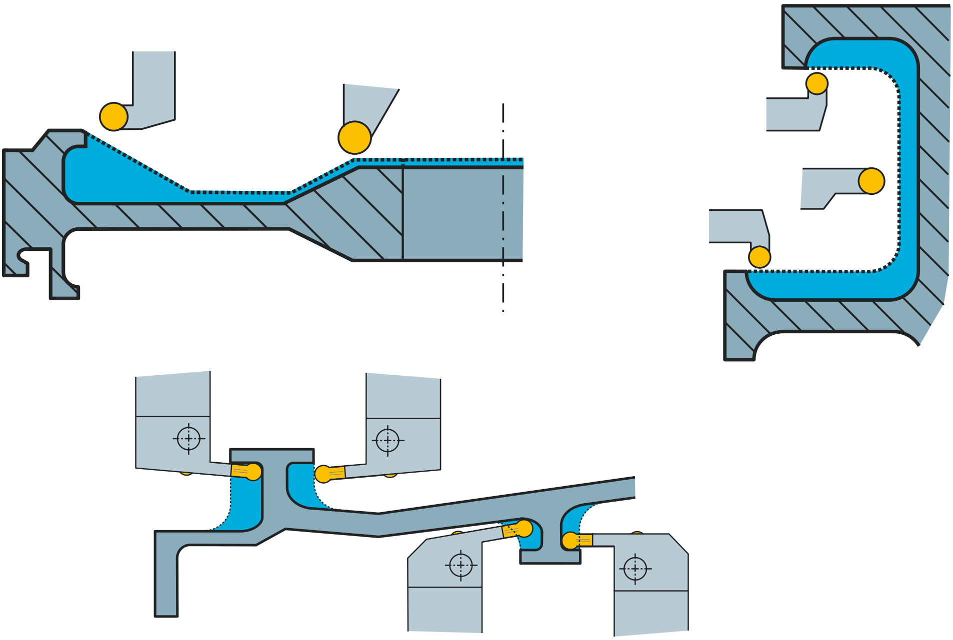
	Competitor	Xcel
		

Insert:	**Competitor**	**CNMX 12 04 A2-SM grade GC1005**
Feed:	0.2 mm/r	0.3 mm/r
Cutting speed:	32 m/min	50 m/min
Number of components/edge:	1 piece	2 pieces

Result
Over 100 % productivity increase
Time in cut reduction: 8 min to 3.5 min

Round inserts and CoroCut® inserts with -RO geometry

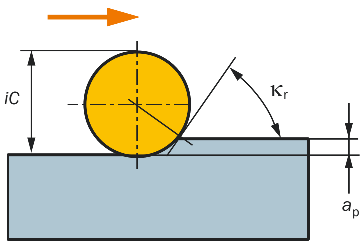
Round inserts are the strongest inserts available, and allow high productivity. Typically aerospace components are large, with large radii and blending profiles designed to eliminate high stress points allowing round inserts to be used.



Entering angle – κ_r

The best performance is achieved when the entering angle remains under 45°. This gives a depth of cut of 0.15 x insert diameter (the maximum depth of cut should be no greater than 0.25 x diameter).

For larger depths of cut than 25% of the diameter, it is better to use square inserts with a constant 45° entering angle.



$$\cos\kappa_r = \frac{(0.5iC - a_p)}{0.5iC}$$

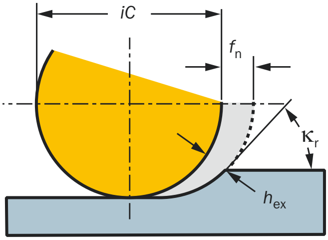
Note: this principle also applies to standard inserts when the depth of cut is less than the nose radius.

Chip thickness

The chip thickness varies with round inserts, and depends upon the entering angle. With low a_p/iC ratios, the feed can be increased in order to raise the chip thickness to a desired level.

Recommended chip thicknesses h_{ex} for HRSA are:

Carbide	0.1 to 0.35 mm
Ceramic	0.08 to 0.15 mm



$$f_n = \frac{h_{ex}}{\sin\kappa_r}$$

Depth of cut to diameter ratio a_p/iC	Depth of cut for insert diameter, in mm								Entering angle κ_r	Feed modification value	Feed min/max mm/r	
	3	4	5	6	8	10	12	16			h_{ex} 0.1 mm	h_{ex} 0.35 mm
0.25	0.75	1	1.25	1.5	2	2.5	3	4	**60°**	**1.16**	**0.12**	**0.41**
0.2	0.6	0.8	1	1.2	1.6	2	2.4	3.2	**53°**	**1.25**	**0.13**	**0.44**
0.15	0.45	0.6	0.75	0.9	1.2	1.5	1.8	2.4	**46°**	**1.4**	**0.14**	**0.49**
0.1	0.3	0.4	0.5	0.6	0.8	1	1.2	1.6	**37°**	**1.66**	**0.17**	**0.58**
0.05	0.15	0.2	0.25	0.3	0.4	0.5	0.6	0.8	**26°**	**2.3**	**0.23**	**0.81**

Example

6 mm diameter CoroCut RO insert.

Depth of cut 0.9 mm gives a maximum entering angle $\kappa_r = 46°$.

To machine with:

Minimum 0.1 mm chip thickness the correct feed is 0.14 mm/r.

Maximum 0.35 mm chip thickness the correct feed is 0.49 mm/r.

Surface finish

The surface finish generated has a direct relationship to both the nose radius size and the feed rate. To achieve a certain surface finish, a small nose radius requires a lower feed rate than a large nose radius – which in practical terms means that small nose radius inserts lead to lower productivity.

Therefore, for the highest productivity the nose radius should be as large as possible – the largest of all being round inserts.

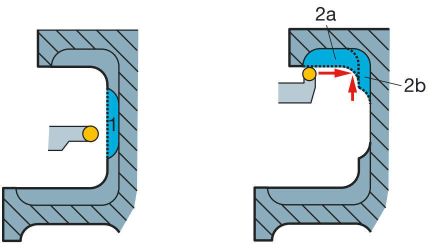
Maximum feed f_n mm/r to achieve surface finish R_{max} 8.0 – R_a 1.6 µm – N7								
	Nose radius size mm				Insert diameter mm			
	0.4	0.8	1.2	1.6	8	10	12	16
f_n	0.17	0.22	0.27	0.32	0.5	0.57	0.62	0.7

Application hints

‘Wrap-around’ is a problem which occurs with round inserts when plunging or profiling into corners. Due to high angular engagement creating high cutting pressures, the feed needs to be reduced. To reduce this problem, optimized programming strategies together with smaller insert radii should be used.

Recommendations

1) Never plunge straight into cut.
2) Roll in and out of cut.
3) Roughing – programmed radius same size as insert diameter.
4) Finishing – insert diameter no greater than 1.75 times programmed radius.



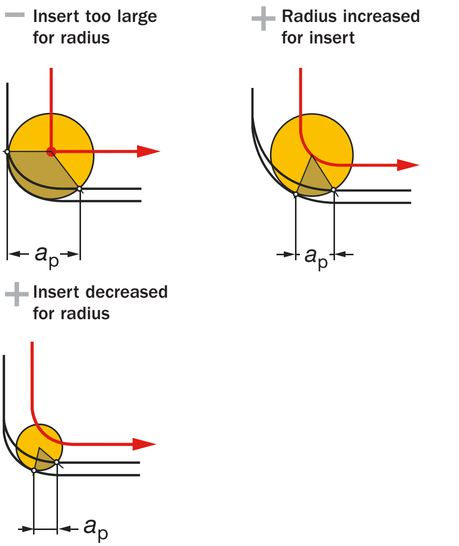
Programming recommendations

Because of the wrap around problem in the radii, the feed needs to be reduced as the size of cut increases. The larger the difference between the radius and the insert diameter, the less the feed needs to be reduced.

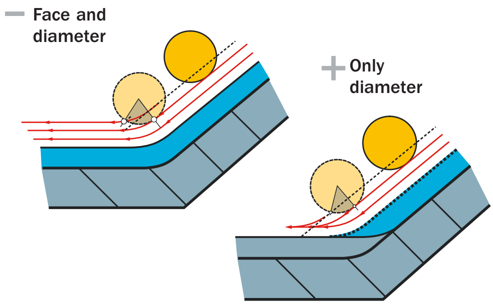
However a good starting point is to use 50% feed rolling into radius compared to parallel cuts.

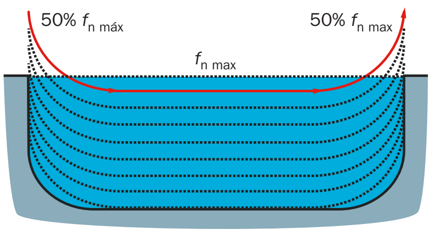
‘Trochoidal turning’

By breaking the part into manageable pieces this method can be used for all profiling requirements. The direction of passes can be alternated when using CoroCut and RCGX inserts to best utilise the edge.



The component feature to be machined needs to be broken down into stages so that a face and diameter are not machined simultaneously as this will naturally increase the depth of cut and cause ‘wrap-around’



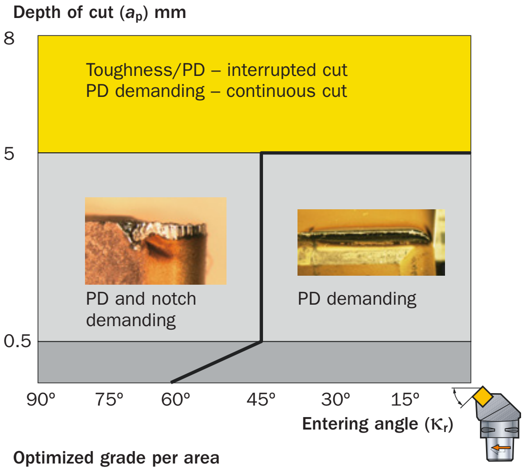


Programmed radius = insert diameter

Cutting tool materials

Grade selection when machining HRSA should not be considered in terms of finishing and roughing. Plastic deformation (PD) is present in all operations, however the formation of notch wear is driven by the entering angle of the insert.

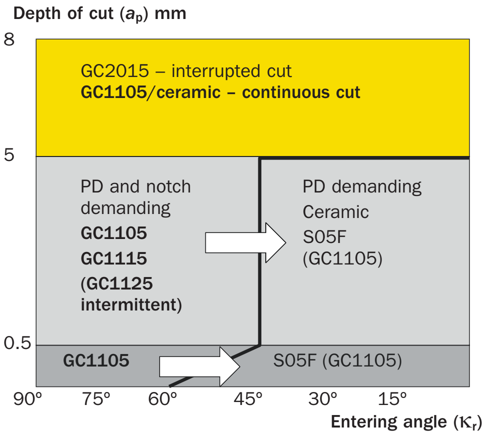
Wear mechanisms per area



In practice this means that the grade selection is optimized depending upon the shape of the insert.

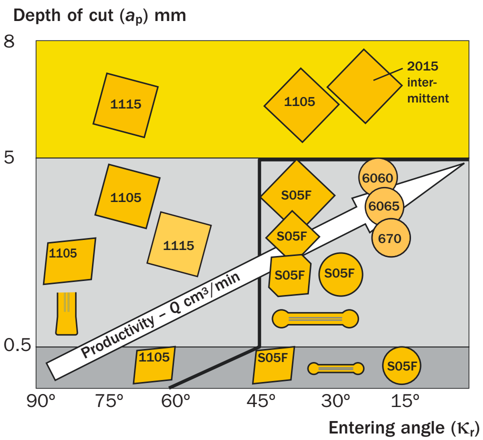
For an ‘all-round’ grade, working in all areas GC1105 is the best choice.

Optimized grade per area



The diagram shows the most productive choice of grade relative to the insert shape.

Optimized insert shape per area



= FSM 26 HRC

= ISM 46 HRC

= LSM 46 HRC

Notch = depth of cut notching

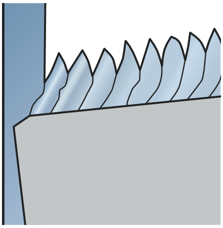
Ceramic insert grades

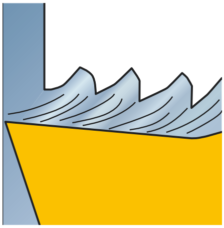
Ceramic cutting materials offer excellent productivity in roughing operations in FSM and ISM. Their application differs greatly compared to carbide due to their:

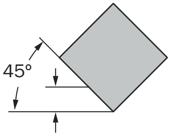
- High temperature resistance – allows high cutting speed to be applied to produce a highly plasticised and sheared chip.
- Low toughness – can lead to edge frittering, top slice and notch wear.

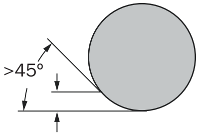
Both of these factors mean that for successful application the following rules need to be applied:

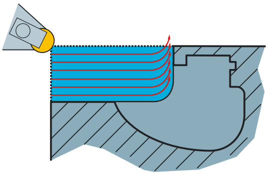
- Optimize entering angle at about 45° to reduce notch, round or square inserts should be used.
- Maximum chip thickness – between 0.08 to 0.15 mm.
- Optimized programming techniques
  - to minimize notch on entry and long passes.
  - control the cutting arc of engagement in corners.

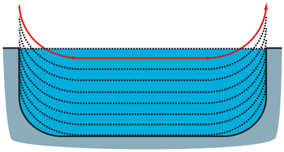

Ceramics


Carbide



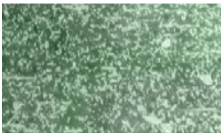


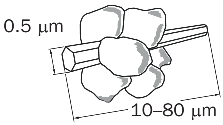




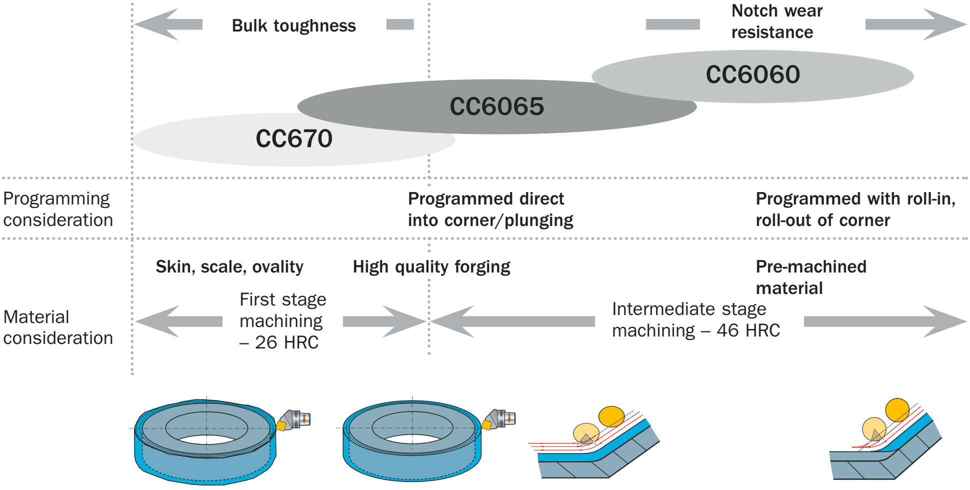
There are 2 types of ceramics developed for use in HRSA:

- **Sialon** – (**Si**licon, **Al**uminium, **O**xygen, **N**itrogen) – a mixture of silicon nitride and aluminium oxide. It has the best chemical stability resisting notch wear:
  - CC6060 – optimized choice for long cutting lengths in clean material and for profiling/pocketing with optimized programming techniques.
  - CC6065 – optimized for heavy roughing applications, plunging and machining direct into corner.
- **Whiskered ceramic** – to provide the improved toughness and bulk strength compared to the traditional ceramic, fibres are included:
  - CC670 – first choice for machining of forged components with rough scale and ovality.





Grade application areas



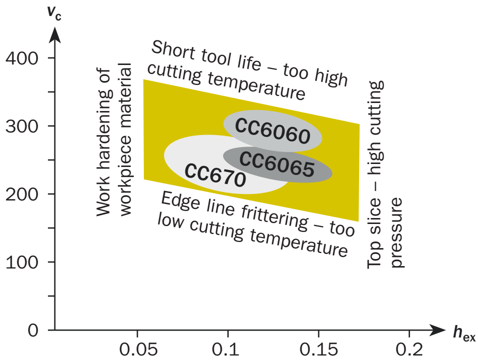
Cutting parameters

The speed should be balanced to create enough heat in the cutting zone to plasticise the chip but not too high to unbalance the ceramic.

The feed should be selected to give a chip thickness which is high enough to not work-harden the material but not be too high to cause edge frittering.

Higher feeds and depths of cut require a reduction of the cutting speed.

These boundaries will change depending upon the component material hardness and grain size.



Start cutting data recommendations (RNGN 12, RCGX 12) – Inconel 718 (38 to 46 HRC)

Grade	Cutting speed, v_c	Cutting depth, a_p	Feed, f_n
CC670	200 to 300 m/min	2 mm	0.1 to 0.15 mm/r
CC6065	200 to 250 m/min	2 mm	0.15 to 0.2 mm/r
CC6060	250 to 300 m/min	2 to 3 mm	0.15 to 0.2 mm/r

Application hints for round inserts

Trouble shooting – wear mechanisms

Top slice

Reason	Remedy
Too high cutting pressure	Reduce feed Reduce a_p on round insert Use CC670

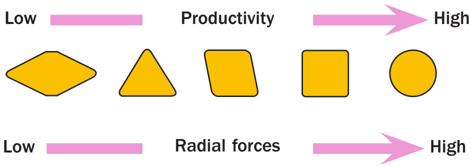
Notch wear

Reason	Remedy
Sensitive cutting tool material Specific to HRSA	Careful programming techniques Reduce entering angle Use CC6060, CC6065

Application hints for ceramic inserts

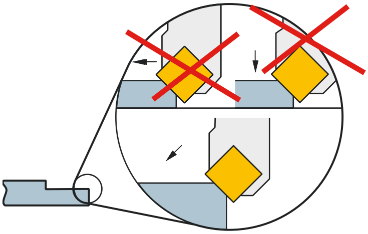
Insert selection

- Where possible use round or square inserts with a small entering angle and large nose radius.
- Always use the strongest insert nose angle.
- Thick inserts give additional strength.



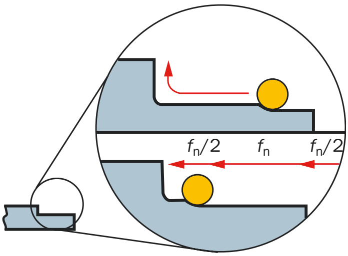
Pre-chamfering

- Protects the insert when it first enters from initial chipping/notch formation.
- To avoid notch wear when chamfering, use a direction feed at 90° to the produced chamfer.



Turning to a shoulder either:

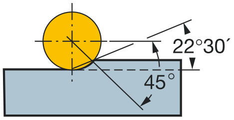
- Roll up to the shoulder with a radius the size of the insert diameter to prevent increased depth of cut.
- Reduce feed by 50% ($f_n/2$) when approching a shoulder due to the depth of cut increasing.



Notch wear

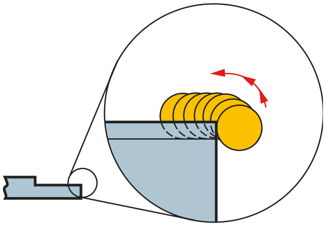
Notch wear can be minimized with good planning and some general advice:

- Use round inserts whenever possible – ensure the relationship between depth of cut a_p and insert diameter does not exceed 25%.

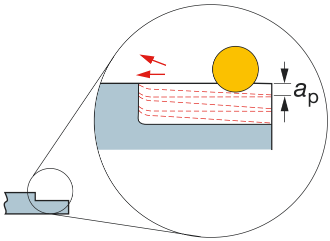


mm	mm
6.35	0.9
9.52	1.4
12.70	1.9
19.06	2.8
25.40	3.8

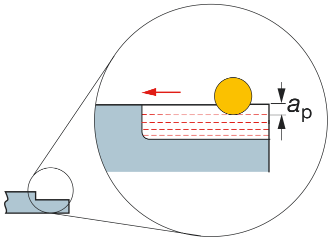
- Use 45° entering angle when depth of cut exceeds 25% of *iC*.
- “Roll over action” in programming to eliminate the need for pre-chamfering and minimize the notch wear. There will be one contact point where the insert hits the hard scale/surface at the corner of the component and one different point at the a_p line.



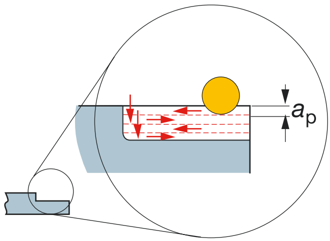
- Ramping ensures that any damage is spread out along the cutting edge. The depth of cut should be varied between 25% *iC* to 15% (do not ramp to zero).



- Multiple passes with varying a_p can be an alternative.



- For RCGX/RPGX inserts, program in both directions to utilise more edges on the insert.



Carbide insert grades

GC1105

A TiAlN PVD coated fine-grain carbide with good hot hardness and toughness properties. It is optimized for inserts with 95° entering angle but is an all-round grade giving effective performance through FSM, ISM and LSM areas.

The coating gives extremely good adhesion which is required for lower feeds and depths of cut. SGF is a ground insert with sharp edges, this combined with GC1105 is outstanding in extreme finishing on vibration sensitive components with low feeds.

GC1115

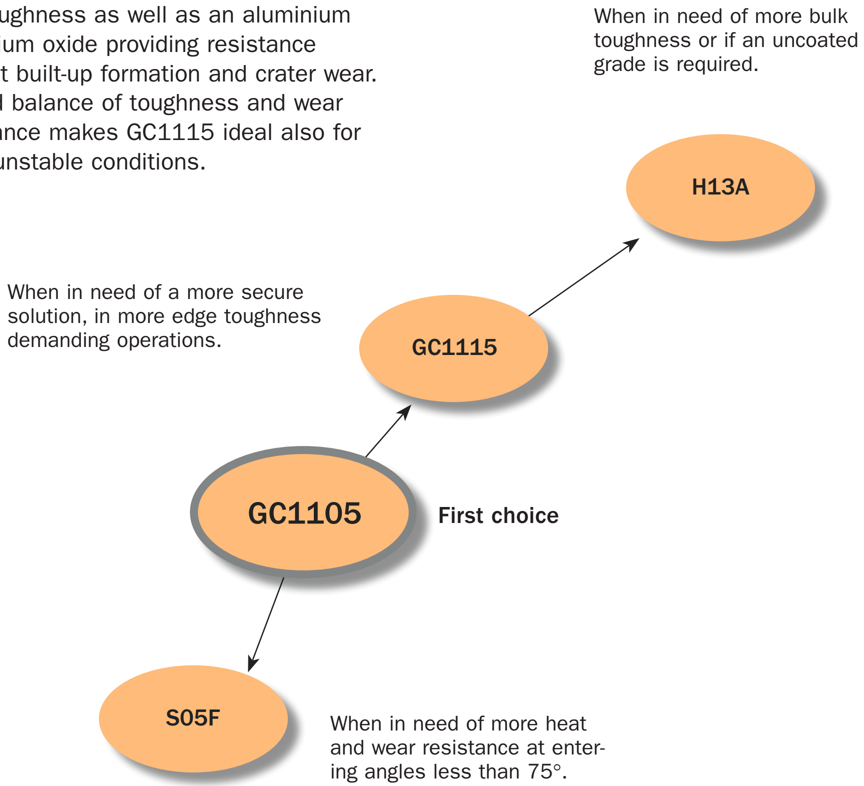
A micro-grained carbide providing security in more edge toughness demanding operations, e.g. reduce problems with notch wear or chip hammering.

The PVD coating contains TiAlN for edge line toughness as well as an aluminium chromium oxide providing resistance against built-up formation and crater wear. A good balance of toughness and wear resistance makes GC1115 ideal also for more unstable conditions.

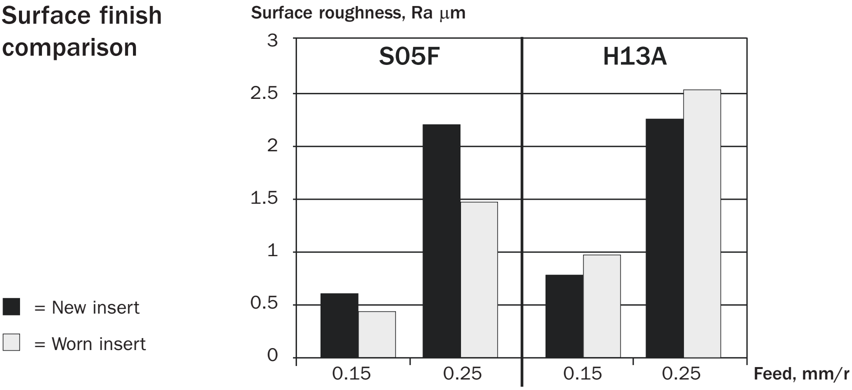
S05F

A CVD coated fine grain carbide with excellent hot hardness properties. It is optimized for applications with a 45° approach (square, round, CoroCut RO and finishing). The CVD coating provides an excellent heat barrier allowing greater productivity and tool life through ISM and LSM.

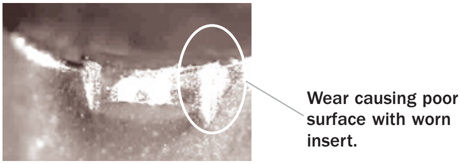
It is a grade for optimizing productivity which does not allow for applications with entering angles over 75° due to poor notch wear resistance.



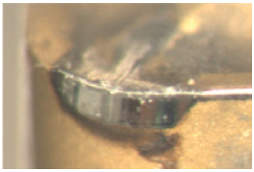
Surface finish comparison



Surface integrity tests have shown that S05F, applied at v_c 40 m/min, gives extremely consistent deformation depth and residual stress profiles, when comparing new and worn inserts. The dimensional accuracy and lack of any coning/taper is also seen to eliminate the need for re-cutting or spring passes. Both of these are the result of minimal wear on the trailing edge compared to other grades.

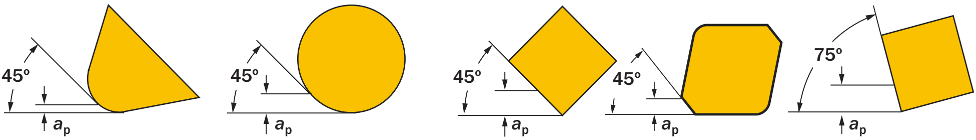


CNMG 120408-23 H13A



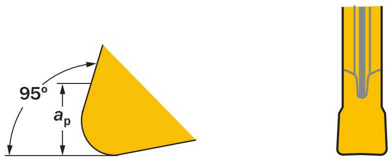
CNMG 120408-SF S05F

S05F applications – PD demanding



CNMG/DNMG **RCMX/RO** **SNMG/CNMX** **SNMG**

GC1105 applications – notch demanding



CNMG/DNMG **TF/GF**

Spiral cutting length (SCL) – predictive machining

SCL was introduced by Sandvik Coromant for the machining of HRSA due to the typically short tool life. One insert normally machines one pass and is then indexed. It is important to be able to predict for:

- Roughing – adding a stop in the program to change the insert
- Finishing – cutting data which ensures that the pass is completed with a predictable wear so as not to have to change during the cut or recut the pass.

SCL is a method of calculating the length of cut required for a particular feature and then confirming with our specific recommendations to ensure a reliable process.

Each SCL graph is unique and only applicable for that insert, geometry, grade, depth of cut and material. For finishing we give you a range of cutting speeds to allow for different length of cut requirements. For roughing we have identified the optimum parameters for each insert style and give you one length.

Application – process flow

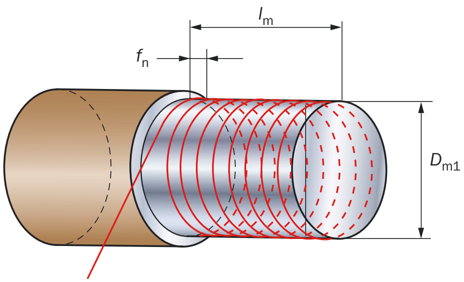
Roughing

1) Select optimum insert style to suit component/process.
2) Use optimized v_c, a_p and f_n for that insert shape/application and note SCL capability eg CNMX 1204A1-SM S05F v_c 50 m/min, f_n 0.35 mm, a_p 1.7 mm.
3) Note SCL capability for that insert SCL = 450 m, see page 30.
4) Calculate SCL for component feature D_{m1} = 450 mm, l_m =150 mm.

$$SCL = \frac{D_{m1} \times \pi}{1000} \times \frac{l_m}{f_n}$$

$$SCL = \frac{450 \times 3.14}{1000} \times \frac{150}{0.35} = 606 \text{ m}$$

5) Confirm calculated SCL with insert capabilities – program required number of insert changes
eg. 606/450 – 2 edges required.



Finishing

1) Select optimum insert style to suit component/process.
2) Use optimized a_p for that insert shape/application choose feed
eg. CNMG 120408-SF 1105 – f_n 0.15 mm, a_p 0.25 mm.
3) Calculate SCL for component feature
eg. Dia 450 mm, l_m = 150 mm.

$$SCL = \frac{450 \times 3.14}{1000} \times \frac{150}{0.15} = 1414 \text{ m}$$

4) Select cutting speed from:
CNMG 120408-SF 1105 a_p 0.25, f_n 0.15 mm' diagram eg. v_c = 50 m/min, see page 30.

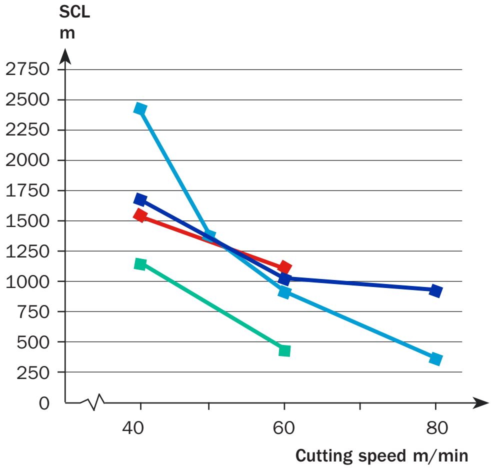
Note. How to calculate, see page 118-119.

SCL recommendations

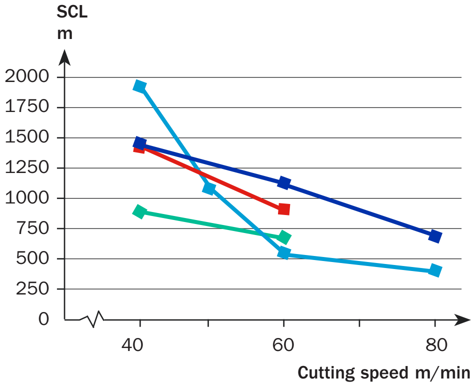
All cutting trials for these data recommendations are in Inconel 718 (46 HRC) and have been found to be true for other nickel alloys in the same hardness – Udimet 720, Waspaloy.

LSM/finishing

CNGG 120408-SGF 1105
CNMG 120408-SF 1105
CNMG 120408-SF S05F
CNMG 120408-SM 1105

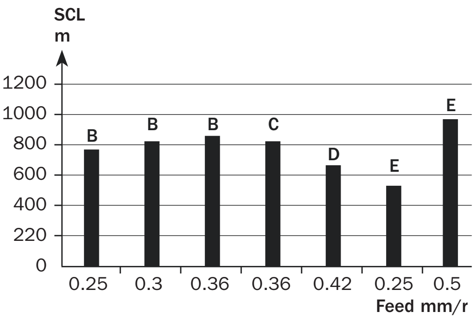


a_p 0.25 mm – f_n 0.15 mm/r



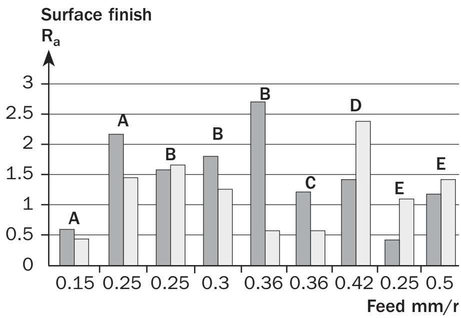
a_p 0.25 mm – f_n 0.25 mm/r

SCL for CoroCut RO geometry S05F –
a_p 0.25 mm – f_n various



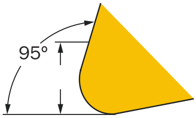
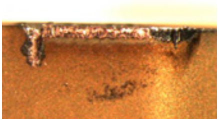
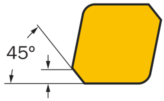
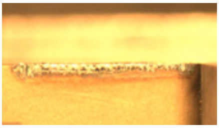
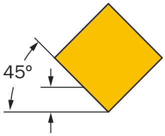
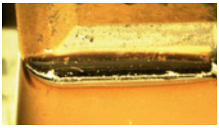
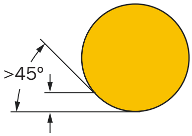
A = CNMG 120408-SF S05F
B = N123H2-0400-RO S05F
C = N123J2-0600-RO S05F
D = N123L2-0800-RO S05F
E = RCMT 1204M0-SM S05F

Surface finish for
radius size



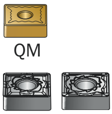
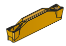
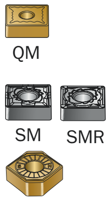
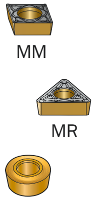
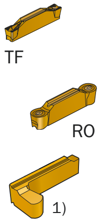
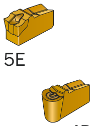
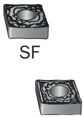
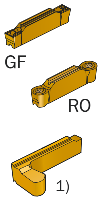
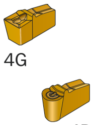
= New
= Worn

Roughing

	v_c m/min	a_p mm	f_n mm/r	Tool life min	SCL m	Q cm³/min	Q_{tot} cm³	
 **CNMG 120408-SMR 1105**	50	2	0.25	**5**	**250**	**25**	**125**	
 **CNMX 1204A2-SM S05F**	50	2.7	0.35	**9**	**450**	**47**	**425**	
 **SNMG 120408-SMR S05F**	50	3	0.35	**9**	**450**	**53**	**473**	
 **SNMG 190616-SM S05F**	50	5	0.35	**9**	**450**	**88**	**788**	
 **RCMT 1204M0-SM S05F**	50	2	0.5	**5**	**250**	**50**	**250**	
 **RNGN 120700 T01020 6060**	250	2	0.2	**4**	**1000**	**100**	**400**	

Geometries and chip breaking

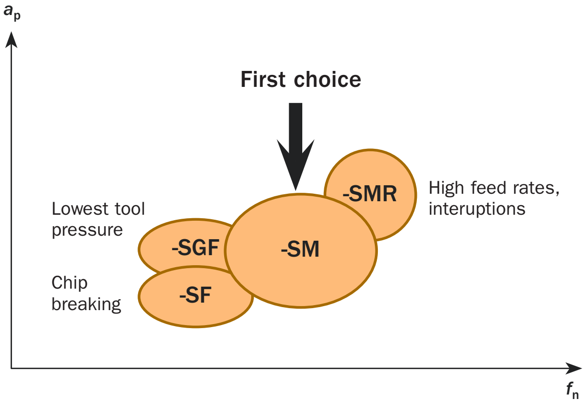
Recommendation for tool geometry:

Application area	Feed range, mm	Edge rounding	Geometry requirements	Geometry recommendation			
				General turning		Grooving	
				Double sided	Positive	CoroCut®	Q-Cut
FSM	0.20 to 0.4	Medium	Direct pressed – primary land	 QM, SM, SMR		 TF	
ISM	0.15 to 0.25	Medium to small	Direct pressed positive rake angle	 QM, SM, SMR, Xcel	 MM, MR, SM	 TF, RO, 1)	 5E, 4P
LSM	0.1 to 0.2	Small	Direct pressed ground	 SF, SGF	 MF, *CGT-UM, SM	 GF, RO, 1)	 4G, 4P

1) CoroCut angled inserts, see page 37.
Note: the ground inserts -SGF and *CGT-UM should be used for thin walled components to minimize the cutting forces and hence risk of distortion.

ISO S geometries

First choice recommendations for general turning with double sided inserts.

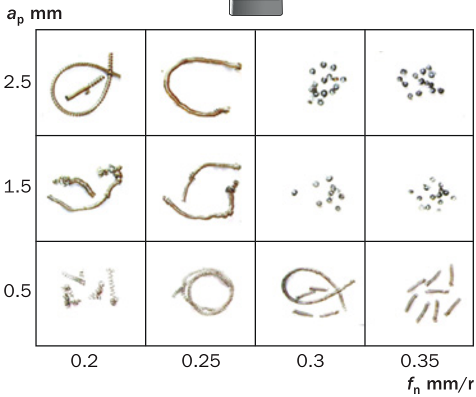


Chip breaking diagrams

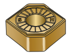
Cutting speed 65 m/min, Inconel 718 – 44 HRC

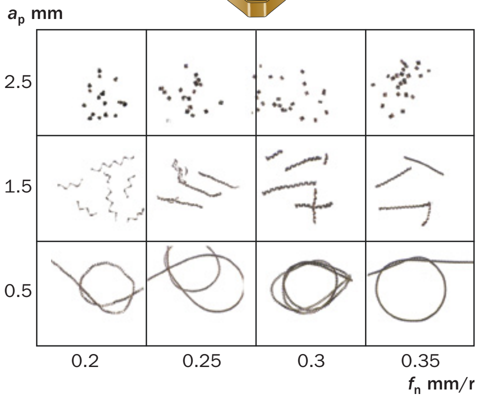
κ_r 45° – ISM

SNMG 120408-SMR



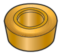
CNMX 1204A2-SM

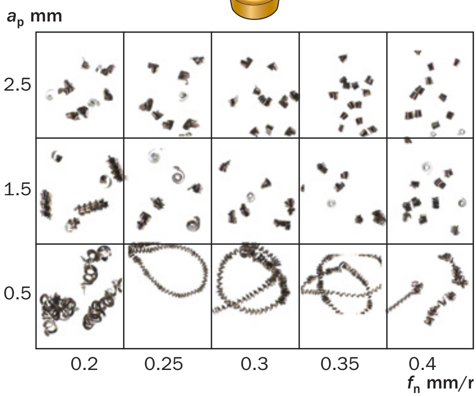




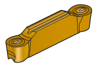
Round inserts – ISM/LSM

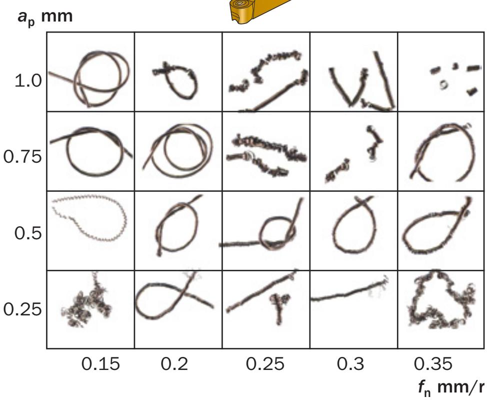
RCMT 1204M0-SM





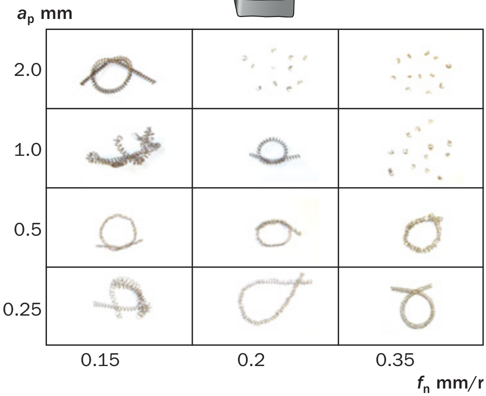
N123J2-0600-RO



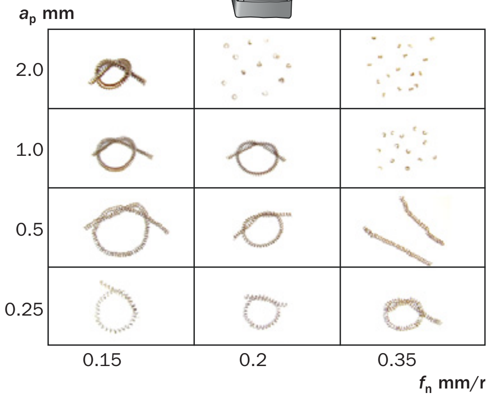


κ_r 95° – ISM

CNMG 120408-SM

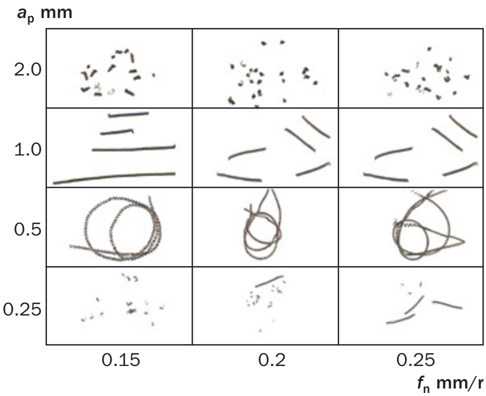


CNMG 120408-SMR

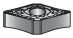


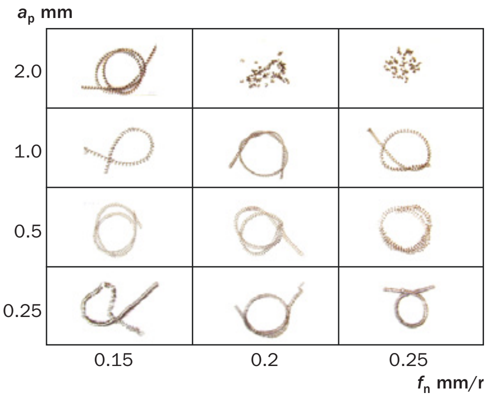
κ_r 95° – LSM

VBGT 160408-UM

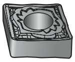


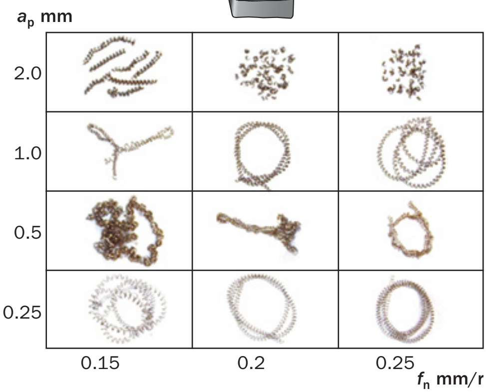
DNGG 150608-SGF



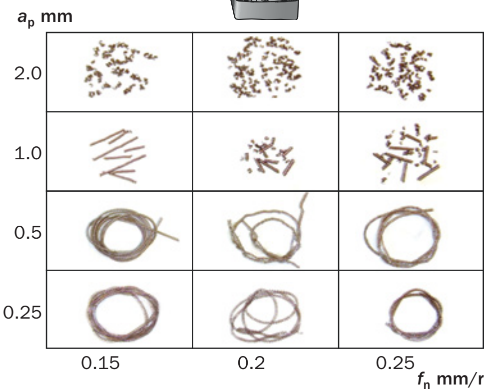


CNGG 120408-SGF



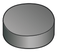
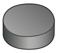
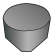
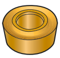
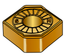
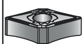
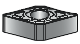
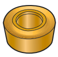


CNMG 120408-SF

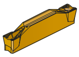
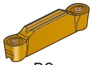
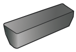
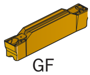
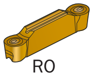


Recommended starting choices for nickel based HRSA

General turning

	1st choice	2nd choice	Cutting speed, m/min	Feed, mm/r	Depth of cut, mm	Insert style	Metal removal rate, cm^3/min	Comments
FSM	GC1105	GC1115	30–40	0.3–0.4	Up to 10	 SNMG 15, 19 -SMR	120	Use 45° approach to reduce chip thickness and notching.
	CC670	CC6065	150–200	0.15–0.2	Up to 5	 RNGN 19  SNGN 19	200	Use 45° approach to reduce chip thickness and notching – first choice on good quality forgings.
ISM	CC6060	CC6065	200–300	0.1–0.2	1 to 3	 RNGN 12	120	Use round inserts wher-ever possible to minimize notching.
	CC6060	CC6065	200–250	0.1–0.2	1 to 3	 RPGX	80	For pocketing use positive inserts.
	S05F S05F	GC1105 GC1105	40–60 40–60	0.3–0.45 0.25–0.35	3 to 5 3 to 5	 SNMG 15, 19 -SMR  SNMG 15, 19 -SM	90 90	Where large depths of cut are possible.
	S05F	GC1105	40–60	0.2–0.45 0.2–0.35 0.2–0.5 0.2–0.35	1 to 3 1 to 3 1 to 3 1 to 2.5	 SNMG -SMR  SNMG -SM  RCMT -SM  CNMX -SM	50 50 50 50	Always use an entering angle less than 75° if pos-sible to reduce notching ie Round RCMT, square SNMG, CNMX.
	GC1105	GC1115	40–60	0.15–0.25	1 to 3	 SNMG -SMR  SNMG -SM  DNMG -SMR  DNMG -SM	25	For 90° entering angle.
LSM	S05F	GC1105	40–60	0.25–0.5	0.25 to 0.5	 RCMT -SM		Profiling
	S05F	GC1105	40–80	0.15	0.25	 *NMG -SF		Use speed according to the SCL required for each operation.
	GC1105	GC1115	40–60	0.15	0.25	 *NGG -SGF		Use the ground -SGF for thin wall components.

Grooving and profiling

Machining stage	1st choice	2nd choice	Cutting speed, m/min	Feed, mm/rev	Depth of cut, mm	Insert style	Comments
ISM	GC1105	GC1125	40– 60	0.1–0.15		 TF  5E	For rough grooving - use largest rad possible.
	S05F	GC1105	40– 60	0.2–0.4	Max a_p 0.15 x *D*	 RO  4P	For profiling.
	CC670		200–400	0.05–0.1		 150.23	For rough grooving under good conditions.
LSM	GC1105	GC1125	40– 60	0.1–0.15	0.25 to 0.5	 GF  4G	For finish grooving. Use 0.4 radius where possible.
	S05F	GC1105	40– 60	0.15–0.35	0.25 to 0.5	 RO  4P	For finish profiling.

Tailor Made

Within many of Sandvik Coromant's product families, the available range of inserts and tools is not limited to those specifically listed in the catalogues. Tooling designed to individual customer requirements is available through the Tailor Made service, allowing customers to specify their cutting tool requirements for particular machining operations.

The Tailor Made service will provide a quotation and proposal drawing within 24 hours of a customer specifying their requirements, and the finished tools within 10 to 20 working days. Ordering can be carried out via the internet, ensuring the most rapid response to customer proposals.

Details of the Tailor Made offer and order forms are available in Sandvik Coromant publications, and on the Internet at www.coromant.sandvik.com.

Finishing groove with Tailor Made CoroCut.

Tailor Made with CoroCut.

Finish groove with small radius.

Engineered solutions for the aerospace industry

We have developed engineered solutions to specific application requirements in the aerospace industry. The solutions will be developed to a specific component – contact your Sandvik Coromant technical representative for more details.

CoroCut® angled inserts for grooving

Thin walls and complex shapes for components in the aero engine leads to a requirement for grooving and profiling inserts within tightly confined spaces. Standard inserts and tool holders often do not offer the combination of sufficient accessibility and rigidity, which is required in tough-to-machine materials.

To overcome these obstacles Sandvik Coromant has developed engineered inserts, which utilise the outstanding stability of the CoroCut tip seat design and optimized grades providing security and productivity.

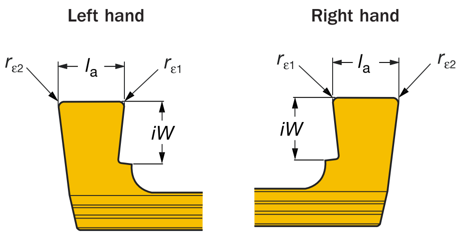
Order finished ground engineered inserts according to your specific requirement:

Width l_a
Depth iW
Radii $r_{\varepsilon 1}$, $r_{\varepsilon 2}$
Chipformers
Width/depth limitations

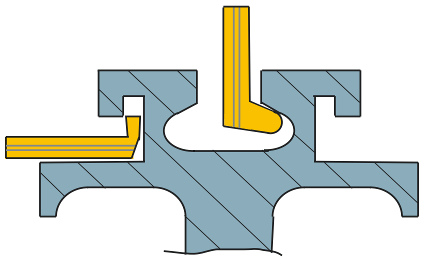
Available grades:

GC1105 – first choice for HRSA applications

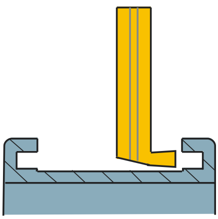
H13A – for interrupted cuts



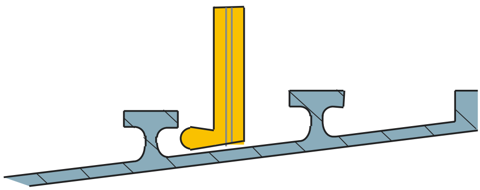
Typical features/applications are found in casings, discs, rings and shafts



Disc, shaft



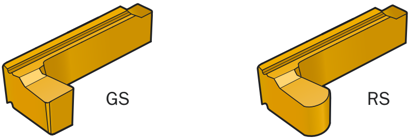
Vane/stator segment



Casing

Standard inserts and blanks for do-it-yourself grinding

An assortment of both left and right handed CoroCut 90° inserts are available as standard in grade GC1115.



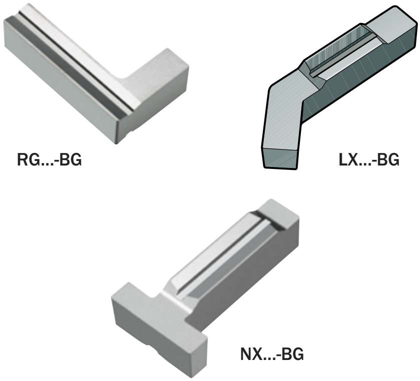
Standard assortment:

- Right- and left-handed 90° design
- Seat sizes H and L
- Width (l_a) 2, 3 and 4 mm
- GS and RS geometry

For do-it-yourself grinding, blanks are available as standard in grades H13A, H10F and H10.

90° blanks

- Seat sizes H and L
- Width (l_a) 6 mm
- 45° and T-shape blanks
- Seat size L
- Width (l_a) 6 mm

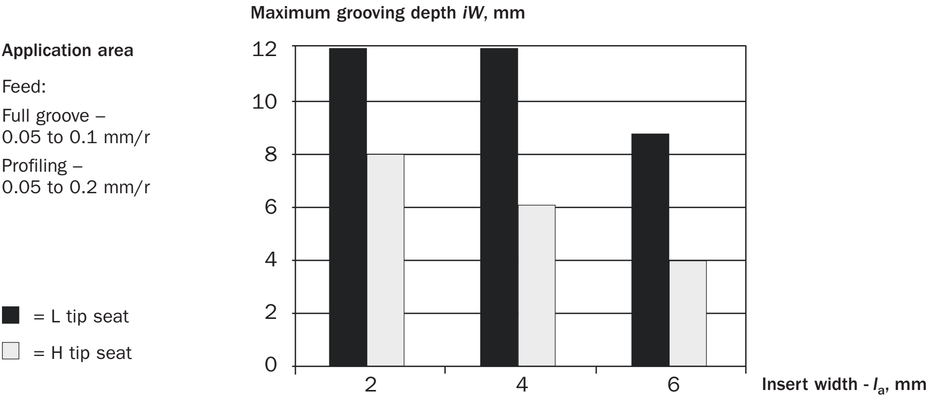


Application area

Feed:

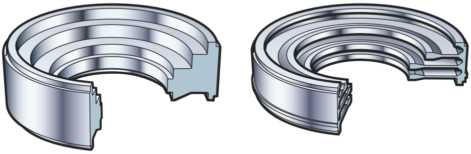
Full groove –
0.05 to 0.1 mm/r

Profiling –
0.05 to 0.2 mm/r



Deep grooving and profiling

Discs and spool components have deep cavities which have to be machined from solid or finish machined after welding. The long depth to width ratio required for the tool means that there is a high tendency to vibration.

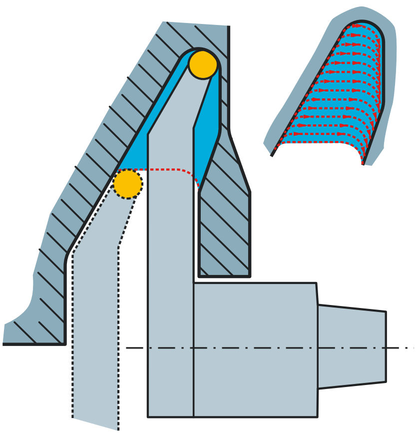


The blade solution is developed to optimize this difficult application offering maximum rigidity and minimized vibration allowing high productivity where long slender tools are required.

- Dampened mechanism incorporated in the blade.
- Coromant Capto® (C6 or C8) spindle adaptor.
- Oval shaft.
- Serration coupling.
- High blade.
- Over and under coolant.

Dampened mechanism

A patented solution is applied when the blade length to width ratio is greater than 5:1. Carbide discs are assembled into the blade which opposes the vibration induced by the cutting process. The effect is that typically 4 times greater depth of cut can be achieved.

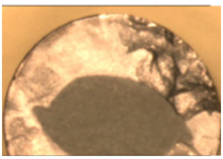


Effect on insert

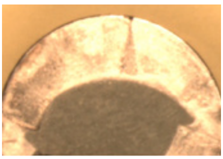
Ceramic RCGX 090700 T01020 670

v_c 250 m/min, a_p 1.5 mm

f_n 0.15 mm/r (G1), f_n 0.075 mm/r (G2/G3)



Non damped



Damped

S-RCMX -SM

Fits in ceramic holders which reduces both the number of tools required and tool change over:

- Finishing operations
- Mixed production of titanium and HRSA

Secure solution with chip breaker giving excellent productivity and chip control.

S-RCMX 060600-SM, dia 6 mm

S-RCMX 090700-SM dia 9 mm

S-RCMX 120700-SM, dia 12 mm

S05F – first choice HRSA

H13A – first choice titanium

S-SNMM-SR

An optimized geometry for machining HRSA materials, with forged skin, in the soft condition (26 HRC). To be able to take largest depth of cut possible to minimize interrupted cutting in scale use: square *iC* 19 or 25 – κ_r 75°

S-SNMM 250924-SR 2015 or 2025
S-SNMM 190616-SR 2015 or 2025

S-WCMX-GM

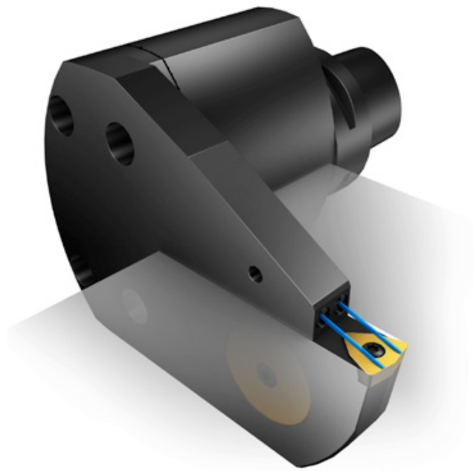
Optimized geometry with excellent chip control. High productivity for plunging operations.

- Wide, deep grooving – 16 mm
- Screw down inserts – no top clamp
- Can be used with HPC or UHPC

S-WCMX 120408-GM

GC1105 – first choice for HRSA

H13A – first choice for titanium



CSGX Ceramic grooving inserts

Fits in RCGX/RPGX ‘V bottom’ holders.

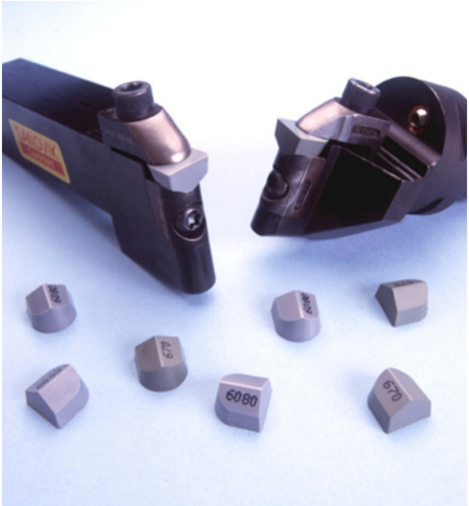
CSGX 060608 T01020 670,
width 6.35 mm

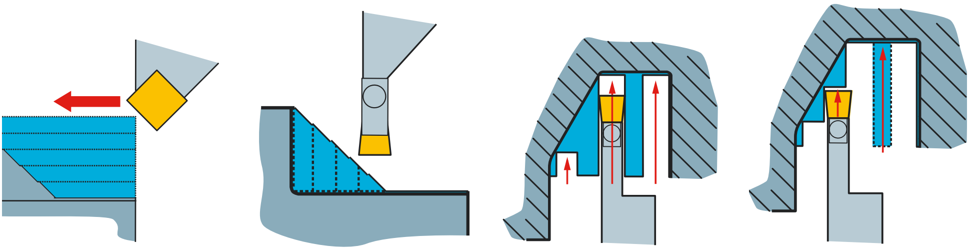
CSGX 090708 T01020 670,
width 9.75 mm

CSGX 120708 T01020 670,
width 12.7 mm

High metal removal – effective use in:

- Machining into corner after square insert
- Wide grooving – use turret method





Seal-fin

Special tool holder for standard inserts.

440 310211R44 right hand

440 310211L45 left hand

- Optimized for good stability and accuracy profiling cuts
- High pressure nozzle
- Good chip control
- Depth of cut (a_r) 8 mm

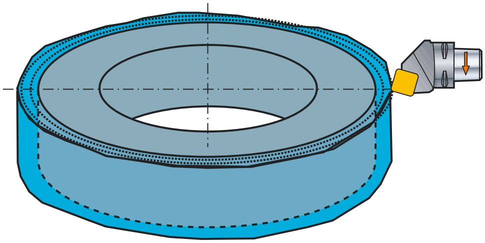
N123E2-0200-RO
width 2.0 mm

GC S05F – first choice for HRSA

Sandvik Coromant component feature solutions

FSM – 26 HRC

Removal of skin. κ_r 75° for larger depth of cut (a_p).

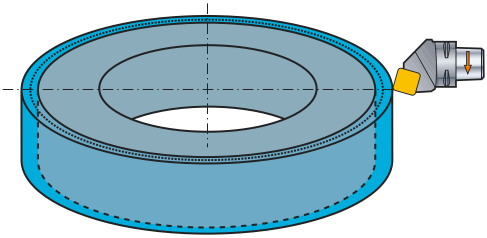


Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm^3/min	SCL, m
S-SNMM 190616 S-SNMM 250924	SR SR	GC2015 GC2015	20 20	10.00 15.00	0.60 0.60	20 20	120 180	400 400

*For Inconel 718 (26 HRC).

FSM – 26 HRC

Clean material.



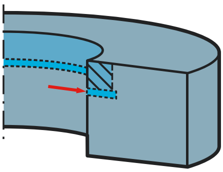
Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm^3/min	SCL, m
SNMG 190616 RNGN 190700	SM	GC1105 CC6065	40 200	8.00 5.00	0.40 0.25	15 5	128 250	600 1000

*For Inconel 718 (26 HRC).

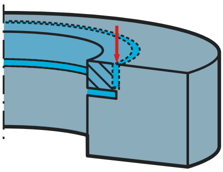
FSM – 26 HRC

Removal of test ring.

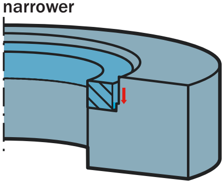
Groove 2 mm past outside dia. of the test ring



Stop groove 2 mm short of breaking through



Breaking through with carbide insert 1 mm narrower

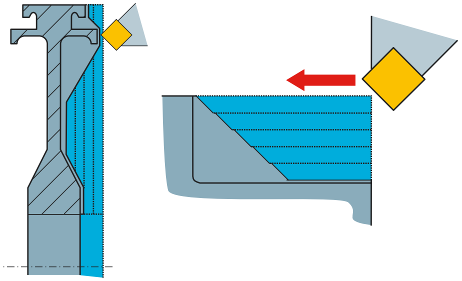


Ceramic 6.35 mm
150.23 063508 T01020 CC670
v_c 300 m/min f_n 0.07 mm/r

Carbide 6 mm
N123K2-0600-0004-TF
1105
v_c 50 m/min
f_n 0.12 mm/r

ISM – 46 HRC

Roughing with high depth of cut.



Max. ramping angle

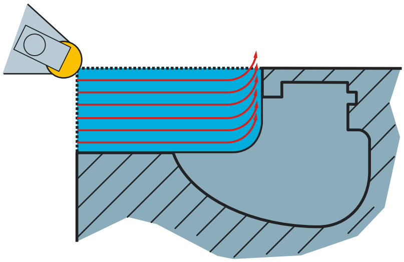
40° for DSDNN
25° for DSSNL/R

Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate cm^3/min	SCL, m
SNMG 190616	SM	S05F	50	5.00	0.35	8	87.5	400
SNMG 120408	SMR	S05F	50	3.00	0.35	8	52.5	400

*For Inconel 718 (46 HRC).

ISM – 46 HRC

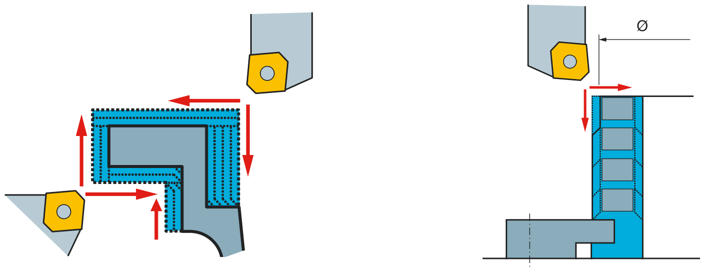
Machining into a corner – Ceramic.



Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm^3/min	SCL, m
RNGN 120700		CC6060	250	2.00	0.2	4	100	1000
RNGN 120700		CC6065	250	2.00	0.2	3	100	750

*For Inconel 718 (46 HRC).

Carbide.

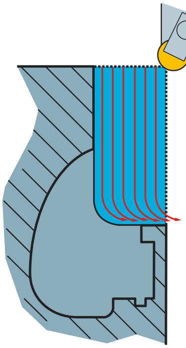


Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm^3/min	SCL, m
CNMX 1204A1	SM	S05F	50	1.50	0.35	8	29.75	400
CNMX 1204A2	SM	S05F	50	2.50	0.35	8	47.25	400

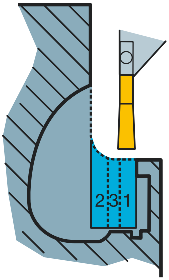
*For Inconel 718 (46 HRC).

ISM – 46 HRC

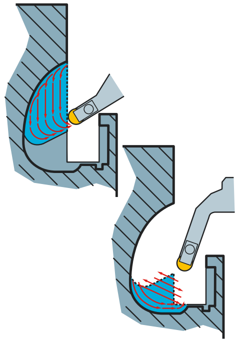
Profiling and pocketing. Ceramic.



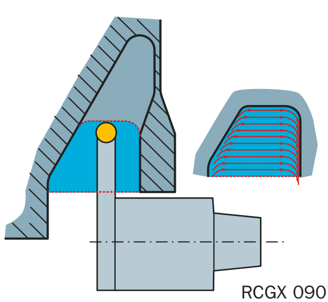
RNGN 120700 6060
v_c 300 m/min
a_p 2.0 mm
f_n 0.15 mm/r

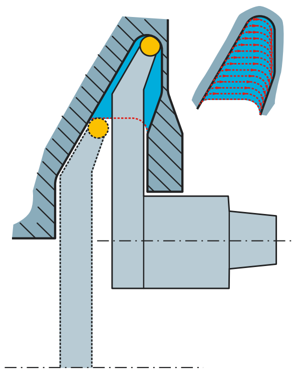


150.23 9.5 mm 670
v_c 300 m/min
l_a 9.5/6 mm
f_n 0.07 mm/r



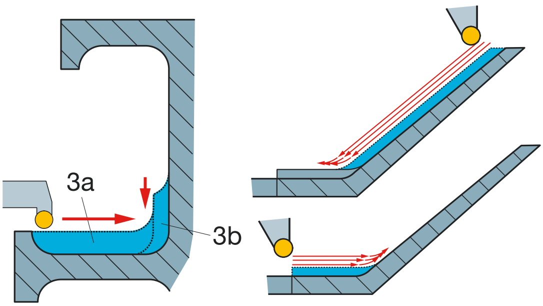
RCGX 090700 6060
v_c 300 m/min
a_p 1.5 mm
f_n 0.15 mm/r





RCGX 090700 6060
v_c 300 m/min
a_p 1.5 mm
f_n 0.15 mm/r
Q 68 cm^3/min

Carbide.

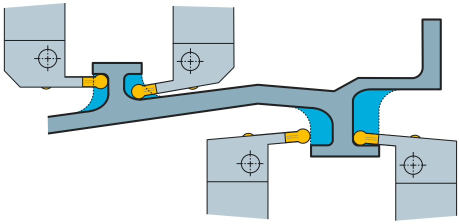


Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm^3/ min	SCL, m
RCMT 10T3M0	SM	S05F	50	1.50	0.35	5	26.25	250
RCMT 1204M0	SM	S05F	50	2.00	0.50	5	50	250

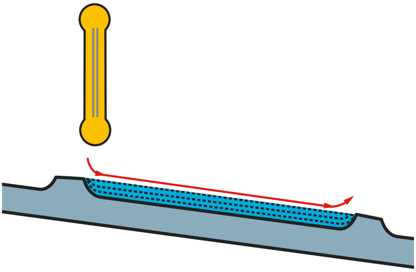
*For Inconel 718 (46 HRC).

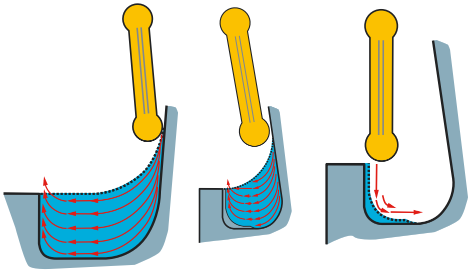
ISM – 46 HRC

Profiling and pocketing. CoroCut.



Use trochoidal turning method described on page 20.





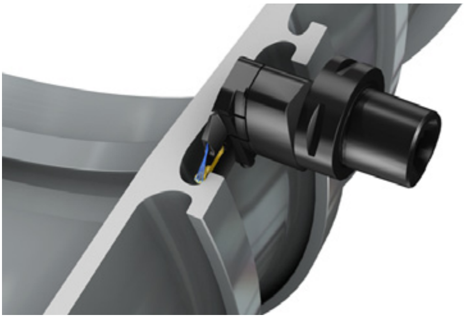
Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm^3/min	SCL, m
N123L2-0800	RO	S05F	50	1.20	0.50	6	33	300
N123J2-0600	RO	S05F	50	1.00	0.40	6	22.5	300
N123H2-0400	RO	S05F	50	0.60	0.30	6	10.5	300

*For Inconel 718 (46 HRC).

CoroTurn® SL70

CoroTurn SL70 blade and adaptor system is designed to fit most profiling and pocketing features in complex components, without the need for special or modified tools.

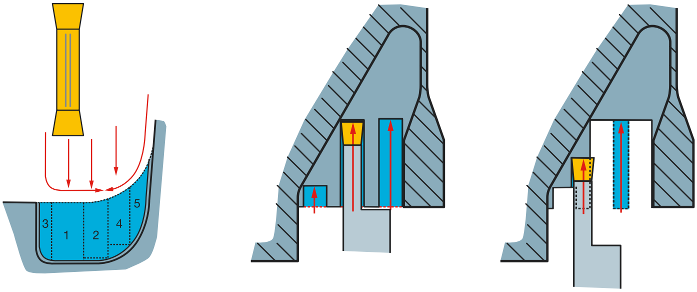
- Offers excellent stability and accessibility.
- Turning tools and inserts for discs, casings, turbine wheels and shafts in HRSA.
- Coolant over clamp for ceramic inserts.
- Blades for carbide inserts have high pressure coolant nozzles as standard.



CoroTurn® SL70 for flexible machining

ISM – 46 HRC

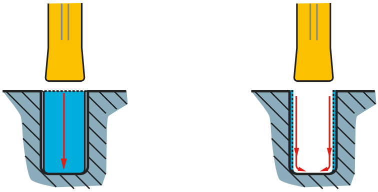
Wide grooving.



Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm³/min	SCL, m
150.23-0635	T01020	CC670	300	6.35	0.07	3	133.4	900
150.23-0950	T01020	CC670	300	9.50	0.07	3	199.5	900
CSGX 090708	T01020	CC670	300	9.50	0.07	3	199.5	900
N123K2-0600-0004	TF	GC1105	50	6.00	0.12	8	36	400
N123L2-0800-0008	TF	GC1105	50	8.00	0.15	8	60	400

*For Inconel 718 (46 HRC).

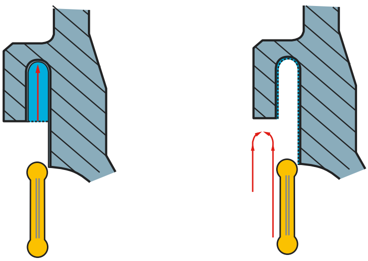
Narrow grooving.



Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm³/min	SCL, m
N123G2-0300-004	GF	GC1105	50	3.00	0.07	8	10.5	400
N123G2-0300-004	TF	GC1105	50	3.00	0.10	8	15	400

*For Inconel 718 (46 HRC).

Narrow grooving full radius.

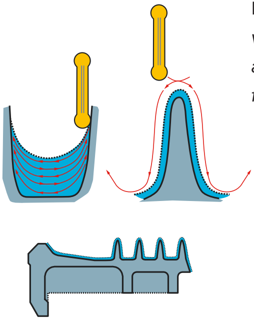


Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm³/min	SCL, m
N123F2-0300	RO	S05F	50	3.00	0.08	12	12	600

*For Inconel 718 (46 HRC).

ISM – 46 HRC

Seal-fin grooves - special holders, standard inserts.



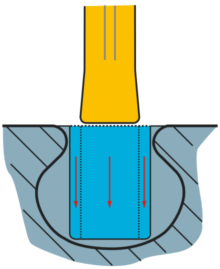
N123E2-0200-RO S05F

v_c 50 m/min

a_p 0.5 mm

f_n 0.25 mm/r

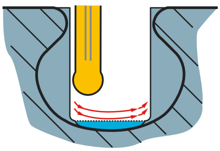
Blade grooves on disc/spool.



N123H2-0400-0004-TF 1105

v_c 40 m/min

f_n 0.1 mm/r

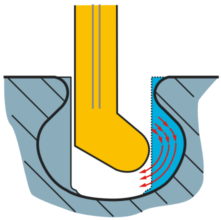


N123G2-0300-RO S05F

v_c 50 m/min

a_p 0.5 mm

f_n 0.25 mm/r



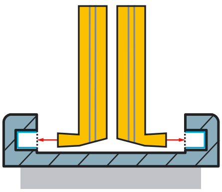
CoroCut engineered insert GC1105

v_c 50 m/min

a_p 0.5 mm

f_n 0.25 mm/r

Grooves on stator vanes.

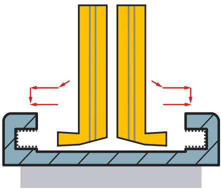


CoroCut 90° insert GC1115

v_c 30 m/min

l_a 2 mm

f_n 0.1 mm/r



CoroCut 90° insert GC1115

v_c 30 m/min

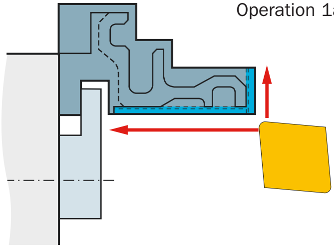
a_p 0.25 mm

f_n 0.15 mm/r

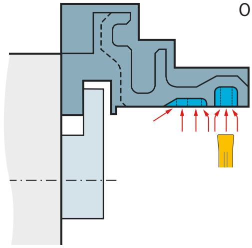
ISM – 46 HRC

Rings

Operation 1a

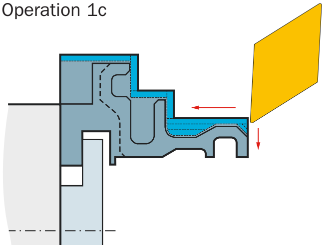


Operation 1b

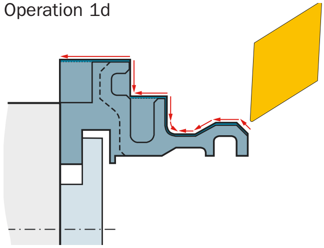


Operation	Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r
1a 1b	CNMG 120412 N123G2-0300-0004	SM GF	GC1105 GC1105	50 50	1.0	0.20 0.08/0.12

Operation 1c

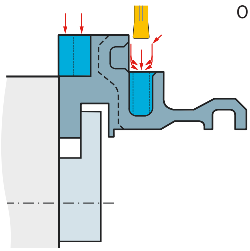


Operation 1d

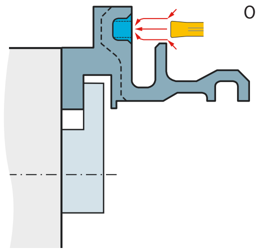


Operation	Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r
1c 1d	DNMG 150612 DNMG 150612	SM SM	GC1105 GC1105	50 50	1.0 0.25	0.20 0.20

Operation 1e



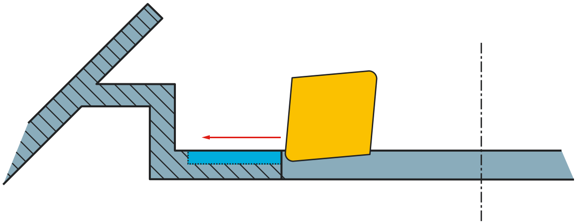
Operation 1f



Operation	Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r
1e 1f	N123G2-0300-0004 N123G2-0300-0004	GF GF	GC1105 GC1105	50 50		0.08/0.12 0.08/0.12

ISM – 46 HRC

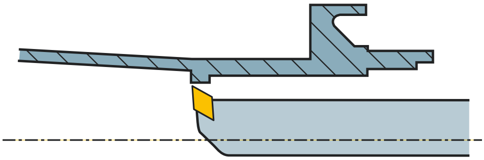
Thin walls/unstable.



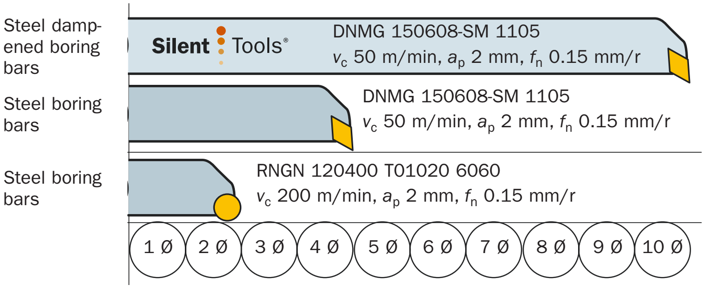
Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm^3/min	SCL, m
CNMG 120408	SM	GC1105	50	1.50	0.20	5	15	250
CNMG 120408	SM	GC1105	50	1.00	0.20	5	10	250

*For Inconel 718 (46 HRC).

Internal machining.



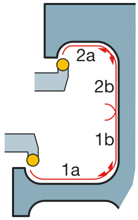
Bar type selection



For overhangs up to 14 x dm_m, use Silent Tools carbide reinforced boring bars.

LSM – 46 HRC

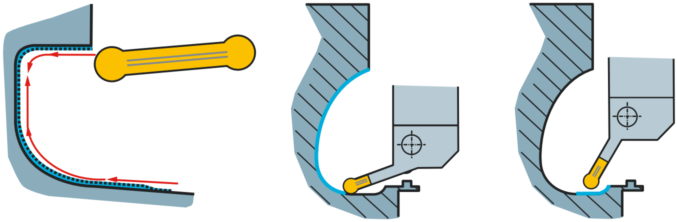
Round inserts.



Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm³/min	SCL, m
RCMT 1204M0	SM	S05F	40	0.25	0.50	20	5	800
RCMT 10T3M0	SM	S05F	40	0.25	0.45	20	4.5	800
RCMT 0803M0	SM	S05F	40	0.25	0.40	20	4	800

*For Inconel 718 (46 HRC).

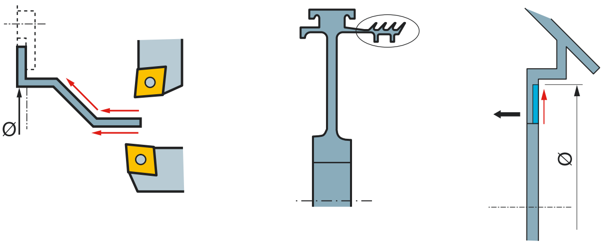
CoroCut.



Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/r	Tool life, min*	Metal removal rate, cm³/min	SCL, m
N123J2-0600	RO	S05F	40	0.25	0.35	20	3.5	800
N123H2-0500	RO	S05F	40	0.25	0.30	20	3	800
N123H2-0400	RO	S05F	40	0.25	0.25	20	2.5	800
N123G3-0300	RO	S05F	40	0.25	0.20	20	2	800

*For Inconel 718 (46 HRC).

Thin wall components.

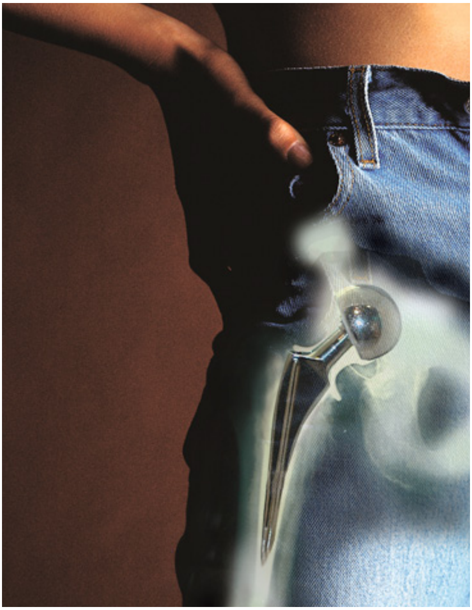


Insert	Geo-metry	Grade	Cutting speed, m/min	Depth of cut, mm	Feed, mm/rev	Tool life, min*	Metal removal rate, cm³/min	SCL, m
CNGG 120404	SGF	GC1105	40	0.25	0.10	20	1	800
CNGG 120408	SGF	GC1105	40	0.25	0.15	50	1.5	2000

*For Inconel 718 (46 HRC).

3. Turning of cobalt-based materials – medical

Material properties



+ resistant to wear
  (hard material 45-50 HRC).
+ can be cast into complex shapes.
+ high corrision resistance.
+ stronger than stainless steel.
– twice the weight of stainless steel.
– very brittle under impact loading.

The cobalt-based alloys in the medical area are mainly of two types: CoCrMo and CoNiCrMo.

CoCrMo alloys are used in applications such as fixation screws, bone plates, shoulder, knee and hip replacement (coated or uncoated, cemented or uncemented).

In this chapter of the application guide we focus on the machining of hip joints in the material group CoCrMo.

Chemical compositions of a few Co-based alloys for implants

CoCr28Mo6 ASTM F75

Vitallium (Howmedica, Inc)
Haynes-Stellite 21 (Cabot Corp.)
Protasul-2 (Sulzer AG)
Micrograin-Zimaloy (Zimmer)

Co	Cr	Mo	Mn	Si	Ni	Fe	C
58.9–69.5	27.0–30.0	5.0–7.0	Max 1.0	Max 1.0	Max 1.0	Max 0.75	Max 0.35

CoCrMo ASTM F799

Forged CoCrMo
Thermomechanical CoCrMo
FHS (Forged high strength)

Co	Cr	Mo	Mn	Si	Ni	Fe	C	N
58–59	26.0–30.0	5.0–7.0	Max 1.0	Max 1.0	Max 1.0	Max 1.5	Max 0.35	Max 0.25

Process considerations

Component condition

The forgings are manufactured from castings or bar stock. This has some impact on the process, whilst the cast blank has less material to remove, the tough skin, with a certain degree of ovality, can cause difficulties if the insert is not fully engaged on the first pass. Bar stock requires much more material to be removed and normally there is a drilling operation to remove the bulk, see engineered solution page 58.

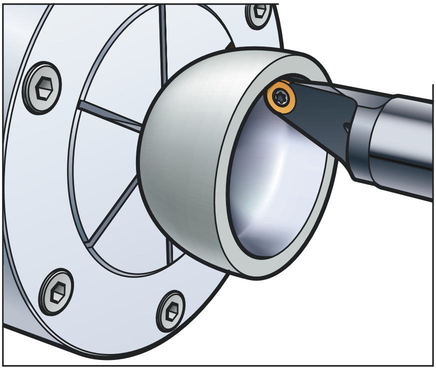
Machining limitations

The process method and productivity are limited by the poor machinability of the material, difficult to access the internal sphere and also weak fixturing. Normally the casting has a spigot for fixturing whilst machining the internal feature, this spigot is later removed.

Machining stage – classification

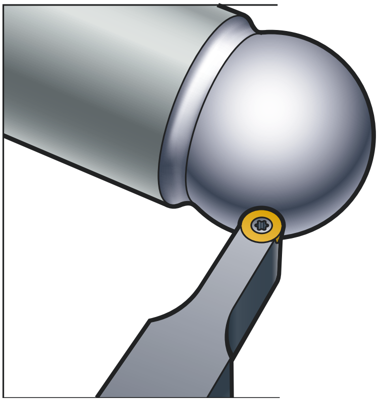
Process/operation 1 – Internal cup

- Rough drilling – bar stock.
- Rough turning – up to 1 mm depth of cut.
- Semi-finish turning – 0.1–0.15 mm depth of cut.
- Finish turning – 0.05–0.15 mm depth of cut.
- Part off spigot.



Process/operation 2 – External head

- Rough turning – up to 1 mm depth of cut.
- Semi-finish turning – 0.1–0.25 mm depth of cut.
- Finish turning – 0.05–0.15 mm depth of cut.
- Part off – bar stock.

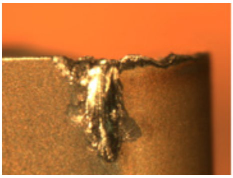


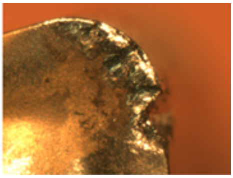
Typical wear mechanisms

Notch wear

A mechanical wear which is concentrated at the depth of cut. This wear reduces the tool life drastically and produces a burr on the component.

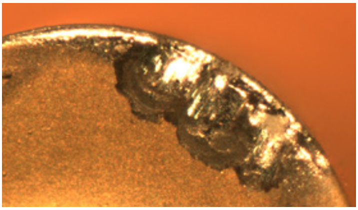
Remedy: It is an application related wear which is easiest solved by changing the entering angle (insert shape) rather than insert grade.





Abrasive wear

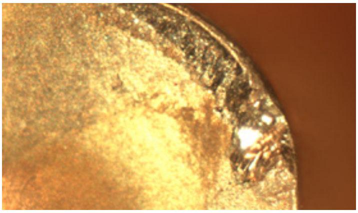
Mainly caused by the hard particles in the workpiece material rubbing or grinding the edge.



Crater wear

Formed through the tool material being removed from the chip face by the hard particle grinding action.

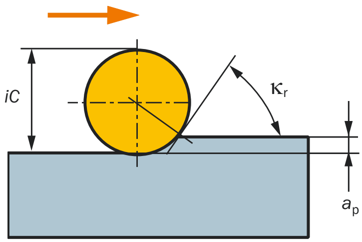
Remedy: Select a positive insert geometry. Reduce the speed to obtain a lower temperature.

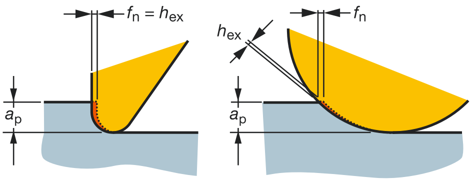


Insert shape selection

From the wear mechanisms it can be seen that a reduction in entering angle (κ_r) has two clear advantages:

- Notch wear reduced giving longer more predictable wear with improved productivity.
- Reduced chip thickness – with a V or D style insert the chip thickness (h_{ex}) is the same as the feed and the cutting length is the same as the depth of cut. Using a depth of cut below the radius reduces the chip thickness relative to feed and increases the cutting edge length. These give the end result of lower temperature and hence longer cutting length and high productivity capability.

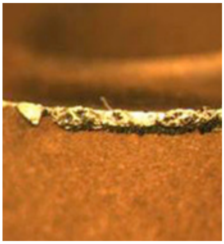




A reduction in entering angle without reducing the depth of cut is best achieved by using a large radius – in practical terms a round insert.

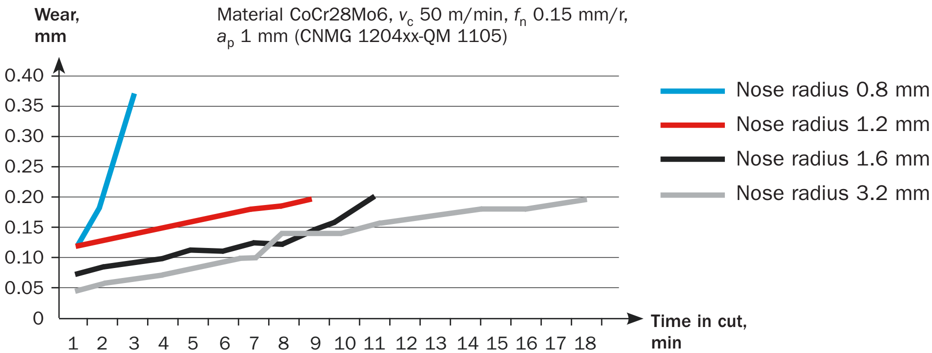
The diagram below shows the effect of the nose radius on tool life. An increase of 6 fold with the same grade.

Tool life: 3 min.
Radius: 0.8 mm



Tool life: 18 min.
Radius: 3.2 mm

Effect of nose radius



Optimized tools for internal machining

To capture the advantages offered by applying large radii, Sandvik Coromant has developed a range of tools which will pioneer traditional processes, giving both increased productivity and tool life.

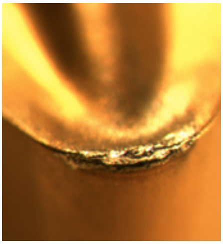
Available styles:

Insert style	**Insert size, mm**	**Boring bar diameter, mm**
DCMT	7	16
DCMT	11	20
Round	8	16
Round	10	20

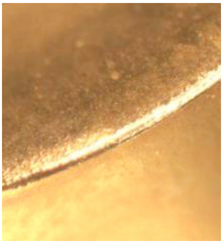
v_c 70 m/min, f_n 0.1 mm/r, a_p 0.5 mm,
material cobalt chromium

Holder: A20M-ADXCL 11-R
Insert: DCGT 11T308-UM 1115

Holder: A20M-SRXDL 08-R
Insert: R300-0828E-PL 1030



1 component.



10 components.

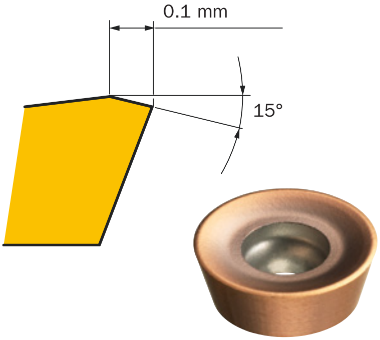
Round insert geometries

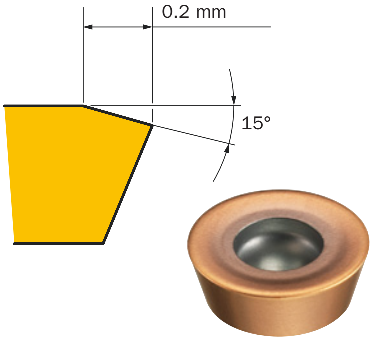
- R300-0828E-PL 1030 or 1010
- R300-1032E-PL 1030 or 1010

+ periphery ground.
+ light cutting.
+ low vibration/cutting forces

- R300-0828E-PH, 1030
- R300-1032E-PH, 1030

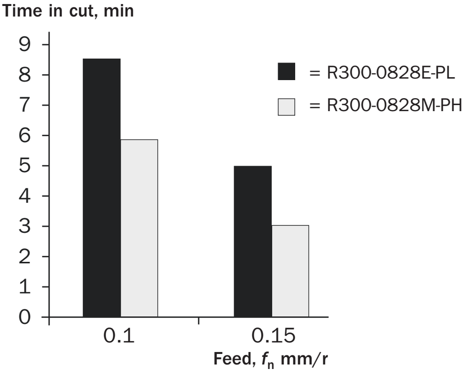
+ direct pressed.
+ need of stable fixture.

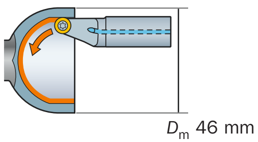




Effect of feed

Facing to centre – Cobalt chromium
D_c 46 mm, v_c 70 m/min, a_p 0.5 mm, GC1030





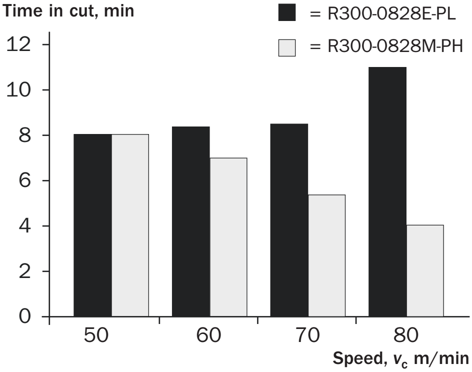
Example:

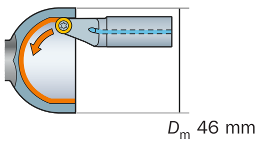
At v_c 70 m/min, f_n 0.1 mm/r, time per pass would be 29 sec.

At v_c 70 m/min, f_n 0.15 mm/r, time per pass would be 20 sec.

Effect of speed

Facing to centre – Cobalt chromium
D_c 46 mm, f_n 0.1 mm/r, a_p 0.5 mm, GC1030





Example:

At v_c 50 m/min, f_n 0.1 mm/r, time per pass would be 41 sec.

At v_c 80 m/min, f_n 0.1 mm/r, time per pass would be 25 sec.

Recommended starting choices for cobalt based HRSA

Machining stage	Feature	1st choice	2nd choice	Cutting speed, m/min	Feed, mm/r	Depth of cut, mm	Metal removal rate cm^3/min	Insert style
Rough turning	Internal	PL GC1010	PH GC1030	50–80	0.1–0.15	Up to 1.5	12	R300-08
	External	SM GC1105	SM S05F*	40–60	0.1–0.15	Up to 1.0	9	RCMT 10
		RO GC1105	RO S05F*	40–60	0.1–0.15	Up to 1.0	9	N123J2-0600
Semi-finish turning	Internal	PL GC1010		50–80	0.1–0.15	0.1–0.3	3	R300-08
		UM GC1105	UM GC1115	40–60	0.08–0.1	0.1–0.25	1.5	DCGT 11 DCGT 07
	External	SM GC1105	SM S05F*	40–60	0.1–0.15	0.1–0.25	2.25	RCMT 10
		RO GC1105	RO S05F*	40–60	0.1–0.15	0.1–0.25	2.25	N123J2-0600
		GC1105	S05F*	40–60	0.08–0.12	0.1–0.25	1.8	DNGG
Finish turning	Internal	PM CT530	PL GC1010	40–60	0.08–0.12	0.05–0.15	1.1	R300-08
		PF CT5015	UM GC1115	40–60	0.08–0.1	0.05–0.15	0.9	DCMT DCGT
	External	GC1105	S05F*	40–60	0.08–0.12	0.05–0.15	1.35	DNGG
		RO CB7015		180-230	0.04–0.08	0.05–0.10		N123J1-0600
		RO GC1105		60-100	0.1–0.15	0.05–0.15		N123J2-0600
Parting		CM GC1125		40–60	0.05–0.1	–		N123G2-0300-0002
Grooving		GF GC1105		40–60	0.05–0.1	–		N123G2-0300-0002

*S05F not recommended for facing to centre.

Carbide insert grades

- GC1105 – Unique thin PVD (TiAlN) coating on hard fine grained substrate.
  – Excellent adhesion to the substrate even on sharp edges, good hot hardness.
- GC1115 – Unique oxide PVD coated on micro-grained carbide.
  – Good resistance against built-up formation and crater wear. Ideal for more unstable conditions.
- GC1125 – PVD coated micro-grained carbide.
  – Good resistance to thermal shock and notch wear. First choice for parting-off operations.
- GC1030 – Unique multi-layer PVD (TiAlN) coating.
  – Improved wear resistance over GC1025 with same toughness. High edge line security.
- GC1010 – PVD coated micro-grained carbide.
  –Resistance to plastic deformation and flank wear due to hot hardness.
- GC5015 – Uncoated cermet.
  – Excellent resistance to built-up formation and plastic deformation.
- S05F – Thin CVD coating on hard fine grained substrate.
  – For applications where notch is not a significant problem ie round insert and finishing with small entering angle.
- CT530 – Uncoated cermet grade.
  – For finishing operations.
- CB7015 – PVD coated CBN grade with ceramic binder for continues cuts.
  – For good surface finish.

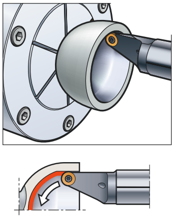
Engineered solution

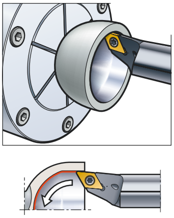
Developed to optimize the roughing from bar stock.

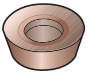
- Engineered Coromant U drill
  – dia 22-35 mm
  – WCMX, GC1020, H13A
- Start cutting data recommendation for titanium and cobalt chromium.
  – Speed v_c= 50-80 m/min.
  – Feed f_n = 0.08-0.12 mm/r.

Sandvik Coromant component feature solutions

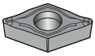
Cups with small radius requirement and/or unstable fixturing (min. dia = 34 mm)



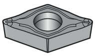




Roughing
Holder: A20M-SRXDR 08-R
Insert: R300-0828E-PL 1030
Cutting data: v_c = 50–80 m/min,
f_n = 0.1–0.15 mm/r, $a_p \leq$ 1 mm

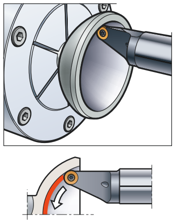


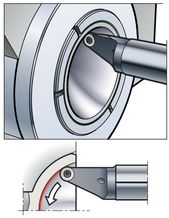
Semi-finishing
Holder: A20M-SDXCR 11-R
Insert: DCGT11T308-UM 1105
Cutting data: v_c = 40–60 m/min,
f_n = 0.08–0.1 mm/r, a_p = 0.1–0.25 mm

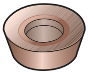


Finishing
Holder: A20M-SDXCR 11-R
Insert: DCGT11T308-UM 1105
Cutting data: v_c = 40–60 m/min,
f_n = 0.08–0.1 mm/r, a_p = 0.05–0.15 mm

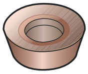
Cups with no radii restrictions and/or stable fixture (min. dia = 34 mm)



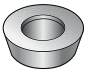




Roughing
Holder: A20M-SRXDR 08-R
Insert: R300-0828E-PL 1030/1010
Cutting data: v_c = 50–80 m/min,
f_n = 0.1–0.15 mm/r, $a_p \leq$ 1.5 mm

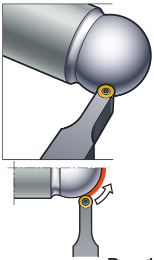


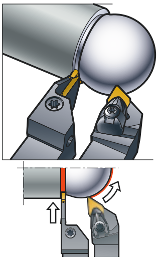
Semi-finishing
Holder: A20M-SRXDR 08-R
Insert: R300-0828E-PL 1030/1010
Cutting data: v_c = 50–80 m/min,
f_n = 0.1–0.15 mm/r, a_p = 0.1–0.3 mm

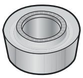


Finishing
Holder: A20M-SRXDR 08-R
Insert: R300-0828E-PL 530
Cutting data: v_c = 40–60 m/min,
f_n = 0.08–0.12 mm/r, a_p = 0.05–0.15 mm

Producing heads from bar material

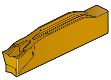






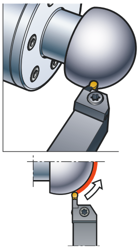
Roughing
Holder: SRDCN 2020K 10-A
Insert: RCMT 10 T3 MO-SM 1105
Cutting data: v_c = 40–60 m/min,
f_n = 0.1–0.15 mm/rev, $a_p \leq$ 1 mm

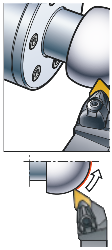
Finishing
Holder: RF123J13-2525MB
Insert: N123J1-0600-RE 7015
Cutting data: v_c = 180–230 m/min,
f_n = 0.04–0.08 mm/r, a_p = 0.05–0.10 mm

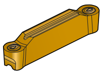


Parting off
Holder: RF123F20-1616B
Insert: N123F2-0250-0002-CM 1105
Cutting data: v_c = 40–60 m/min,
f_n = 0.05–0.1 mm/rev

Producing heads from forged material







Roughing
Holder: RF123J13-2525MB
Insert: N123J2-0600-RO 1105
Cutting data: v_c = 40–60 m/min,
f_n = 0.1–0.15 mm/rev, $a_p \leq$ 1 mm

Finishing
Holder: RF123J13-2525MB
Insert: N123J1-0600-RE 7015
Cutting data: v_c = 180–230 m/min,
f_n = 0.04–0.08 mm/r, a_p = 0.05–0.10 mm

Milling of HRSA materials

When milling super alloys, there are certain process requirements which must be observed.

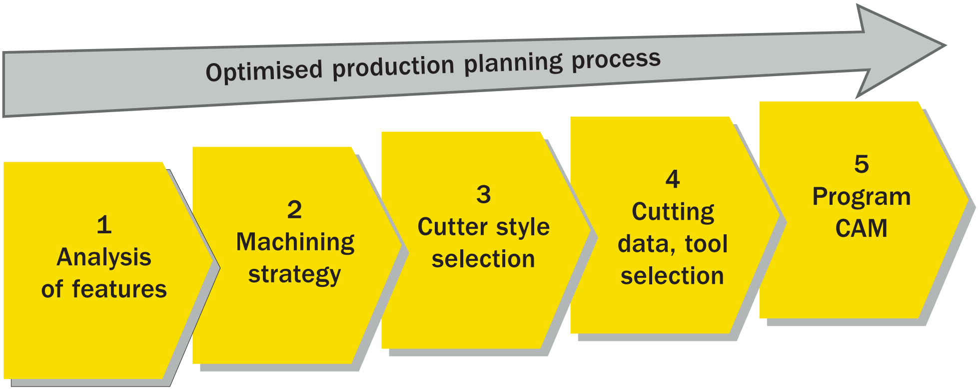
- Milling of high-temperature alloys often requires more rigid and powerful equipment than the milling of carbon steels.
- Cutter accuracy in both radial and axial directions is essential to maintain a constant tooth load and a smooth operation, and to prevent premature failure of individual cutter teeth.
- Cutting edges must be sharp with an optimised edge-rounding, to prevent chip adherence at the point where the edge exits the cut.
- The number of cutting teeth actually in cut during the milling cycle must be as high as possible. This will give good productivity provided that the stability is good enough.
- Cutting speeds for super alloys are generally low. Common practice is to employ a fairly low cutting speed in combination with a moderately high feed per tooth, to produce a chip thickness not less than 0.1 mm which prevents work-hardening of the material.
- Coolant should be applied in generous quantities around the cutting edge when the cutting speeds are low, in order to reduce chip adhesion. Coolant supplied through the machine tool spindle is recommended for HRSA materials. High pressure coolant (HPC) will give better tool life. (no coolant for ceramic milling)
- The cutting edge geometry should always be positive.
- For cutting depths below 5 mm, the entering angle should be less than 45°. In practice, a round, positive-rake insert is recommended.
- If special design cutters are being considered for an application, it is essential to allow sufficient space between each tooth for effective chip evacuation around the cutting edge.
- Flank wear around the cutting edge should not exceed 0.2 mm for R390, 0.3 mm for round inserts in carbide, and 0.6 mm in ceramics. Otherwise the chance of a catastrophic failure increases rapidly. Normal best practice is to index the cutting edges at frequent intervals, to ensure a reliable process.
- Down milling (climb milling) should be used, to obtain the smallest chip thickness where the edge exits cut and reduce any chip adherence
- The machining of iron-based super alloys, and solution treated, nickel-based alloys (Inconel 625), usually machine easier than nickel-based and cobalt-based super alloys.

Production planning process

To optimise a machining operation all aspects of the application should be considered:

- Type of operation?
- Tool paths?
- Conventional or climb milling?
- Cutting tool type and material?
- Change of operation sequence?

Optimising the above is of course necessary, however, to achieve an optimised process these must be combined with process and application 'know how' to achieve secure productive machining. This information considers in which order to build an optimised process for HRSA milling and discusses important 'success factors' for each stage.



Typical components

Because of HRSA's excellent metallurgical properities they are used in a variety of industries including:

Aerospace engine
– Combustion & turbine components, mounting brackets

Selected automotive components
– Turbo chargers, exhaust valves

Medical components
– Dentistry, prosthetic devices

Space vehicle components
– Aerodynamically heated skins, rocket engine parts

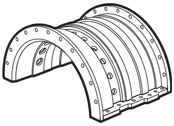
Nuclear power systems
– Valve stems, drive mechanisms

Oil and gas industry
– Marine applications

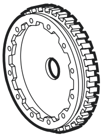
By far the most common application for HRSA materials is the aircraft engine. The use of HRSA in the combustion part of the engine is increasing. This is typified by the fact that, whereas in 1950 only about 10% of the total weight of an aircraft gas turbine engine was made of super alloys, this has now risen to 50% in today's modern engines.

It is predicted that HRSA will continue to be used extensively in the combustion parts of an aircraft engine and recent developments have seen the next generation of HRSA materials being developed and implemented in production.

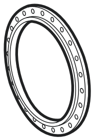
Typical components – aerospace



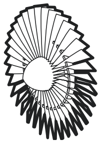
Combustion casings



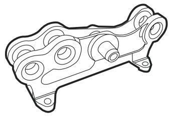
Discs



Rings



Blades/blisks



Mounting brackets

Machining strategy

Features

Most HRSA components are critical parts of the aircraft engine with complex features to be machined. For example when machining an engine casing the most time consuming operation is machining the band between the bosses. Careful planning and application of modern cutting tool materials can dramatically reduce cycle time. Planning the order of operations in order to reduce part distortion is also an important factor when planning the machining strategy.

Machine requirement

Horizontal/Vertical

For larger components such as engine casings where there are many different features and where access is an issue it is best to use a horizontal machine. This also makes it easier to evacuate the chips, preventing re-cutting of the chips, giving a more secure tool life. For some ring components and mounting brackets vertical machine tools can create improved stability.

Configuration – 3/4/5-axis

It is common to have a fourth/fifth axis on horizontal machines to give good accessibility e.g. for casings and closed faces.

For complex parts (3D profiles, blisks) 5-axis rigid machines are used, with fully simultaneous five access control.

Spindle speed

Three cutting strategies dictate the spindle speed requirements:

a) Solid carbide tools – (low torque) – the cutting speed (v_c) is between 30 to 100 m/min. For cutter diameters 8 to 16 mm this will give an rpm requirement between 4000 to 600 rpm

b) Carbide inserts – (high torque) - the cutting speed (v_c) is normally limited to a maximum of 40 m/min, for cutter diameters 25 mm to 80 mm this will give an rpm requirement of 500 rpm to 159 rpm.

c) Ceramic inserts – (high power and torque) – typically cutting speeds (v_c) can be as high as 1000 m/min, for cutter diameters 50 mm this will give an rpm of 6365.

Table feed

For roughing, using carbide inserts, the table feed is naturally relatively low, putting demands on stability rather than speed.

For roughing, with ceramics, the table feed can be up to around 2.5 m/min. Whilst this is not an extreme table feed, care must be taken that the control system can cope with direction change at this feed to avoid undercutting/shortcuts etc.

Power/torque requirement

Basically, the power requirement varies with the amount of metal to be removed, average chip thickness, cutter geometry and speed. The greater the metal removal rate (Q cm^3/min) the higher the power requirement.

With spindle speeds for roughing much lower than for less exotic materials, it puts great importance on ensuring that power and torque are available at low rpms (a machine with insufficient torque and power will give fluctuating chip thickness in turn giving unstable performance).

The diagram below shows how a typical machine power diagram looks with lower power at low rpms.

Spindle coupling

Most HRSA alloys work-harden during machining and have higher strength and 'gumminess' not typical in other materials. Heavy duty machining equipment and an ISO 50 taper (or equivalent) spindle is recommended to minimise chatter and work-hardening of the alloy ahead of the cutting.

Coolant

Unlike milling in most other materials, coolant is always recommended to assist in chip removal, control heat at the cutting edge and prevent re-cutting of the chips. High pressure coolant (70 bars) applied through the spindle/tools and externally has been seen to give good benefits compared to low pressure.

Fixturing

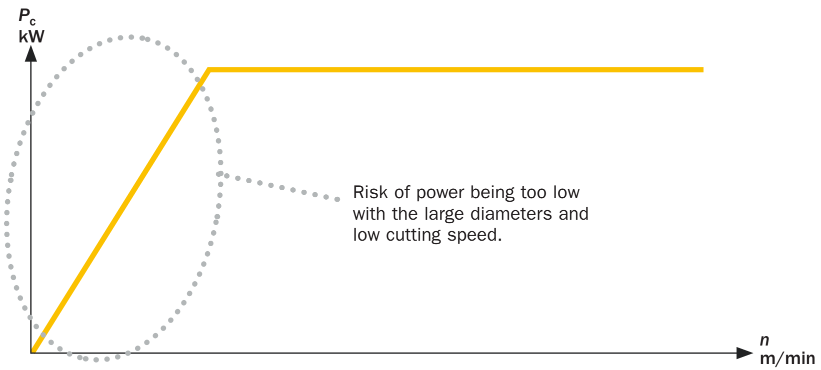
The shape and fixturing of the component is of great importance. Aerospace engine components are often thin-walled and have a lot of complex features which easily create distortion and vibration.

This is often the reason why engine casings have complex fixtures to reduce vibration and support the workpiece in relation to the direction of the cutting forces. Light cutting tools with positive geometries can help to ensure a safe, distortion-free machining operation.

Stability

The condition and stability of the machine has an effect on the quality of the surface and can impair tool life. Excessive wear of the spindle bearings or feed mechanism can result in a poor surface structure.

As well as ensuring a stable machine tool, other factors such as tool overhang, Coromant Capto coupling, tuned adaptors etc should be considered.



Cutter concept

Modern milling is a very universal machining method. During the past few years hand in hand with machine tool developments, milling has evolved into a method that machines a very broad range of configurations. The choice of cutter concept is no longer straightforward – in addition to all the conventional applications, milling is a strong contender for producing holes, threads, cavities and surfaces that used to be turned, drilled, or tapped etc. (See hole making chapter). Tooling developments have also contributed to the new possibilities along with the gains in productivity, reliability and quality consistency that have been made in indexable and ceramic inserts.

The aim is to have a strategy suited to the machine tool and feature to be machined giving a maximum metal removal rate (Q cm^3/min) that can be balanced with an economic tool life.

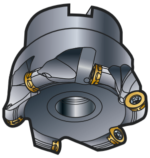
For milling there are many variables to consider:

D_c – cutter diameter
z_n – number of effective teeth
κ_r – entering angle
$a_{p\ max}$ – maximum cutting depth
$a_{e\ max}$ – maximum radial cut

Design types

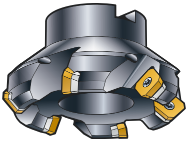
Face mills

Round inserts



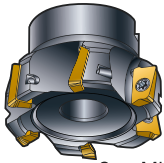
CoroMill® 300

45º



CoroMill® 245

90º



CoroMill® 390

End mills

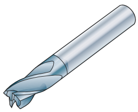
Round inserts

CoroMill® 300

90º

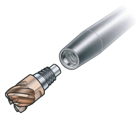
CoroMill® 390

Solid carbide



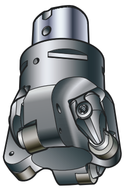
CoroMill® Plura

Exchangeable head



CoroMill® 316

Ceramic insert cutter



S-R120R/CoroMill® 300C

Ball nose

CoroMill® Ball nose 216

The cutter diameter is more or less selected by the operation and the machine capability. The choice of ceramic insert, carbide inserts or solid carbide is determined by productivity calculations, surface requirements and process limitations (machine, fixturing etc). The κ_r/insert style (round, 45°, or 90°) and number of teeth selected will have a dramatic effect on the machining strategy and ultimately on the tool life and *Q*. Therefore the feature type and machining strategy will influence which concept style should be selected.

Milling process

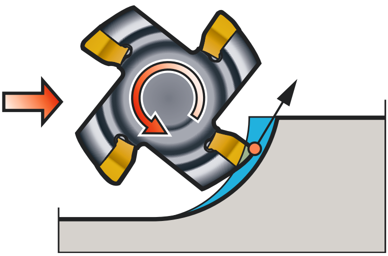
Milling direction

During a milling operation, the workpiece is fed either with or against the direction of rotation and this affects the nature of the start and finish of cut.

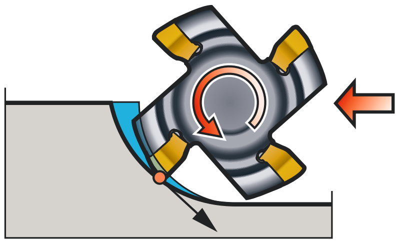
- In up milling (also called conventional milling) – the chip thickness starts at zero and increases to the end of the cut. There are high cutting forces which tend to push the cutter and workpiece away from each other. The insert is forced into cut, creating a rubbing or burnishing effect work-hardening the surface.
- In down milling (also called climb milling) – the insert starts its cut with a large chip thickness. This avoids the burnishing effect with less heat and minimal work-hardening tendencies.

Always use down milling for machining of HRSA materials with carbide inserts.

Up milling



Down milling



Milling with indexable inserts

Cutter diameter and position in face milling

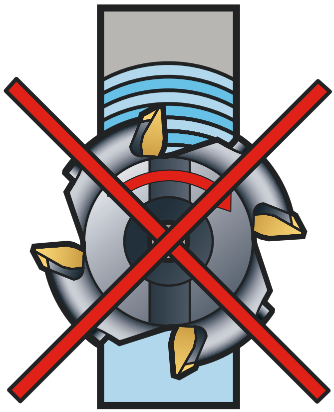
The selection of milling cutter diameter is usually made on the basis of the workpiece width with the power availability of the machine also being taken into account. The position of the cutter in relation to the workpiece engagement and the number of teeth in contact are all vital factors for successful operation.

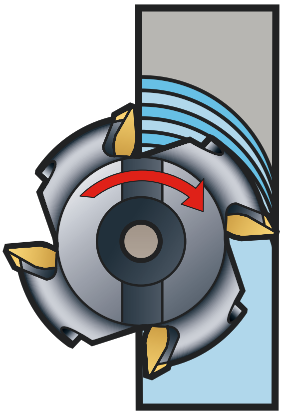
When machining in a single pass, a cutter diameter 20 to 30% larger than the workpiece width is recommended. The milling cutter should always be positioned off-centre producing the thinnest chip on exit. If several passes need to be taken, a radial cut (a_e) of 75% D_c should be applied to ensure good chip formation and suitable cutting edge load.

Avoid positioning cutter on-centre.

Entry into and exit from the workpiece considerations

Each time one of the milling cutter inserts enters into cut, the cutting edge is subjected to a varying shock load. The right type of initial contact, and final contact, between edge and material is an important aspect of the milling process. Positioning the cutter right as regards entry and exit of the cutting edge is important – always avoid a thick chip on exit.





When profile milling the positioning of the cutter can be pre-defined, however for face milling where the position of the cutter is more flexible it can be prone to misapplication.

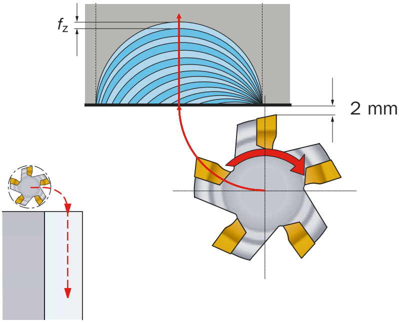
1) a_e should not be greater than 75% of the cutter diameter, and not less than 30% – at least 2 teeth in contact (if $z_n > 2$).

2) The cutter should be off-centre giving as close to zero chip thickness as possible on exit from cut.

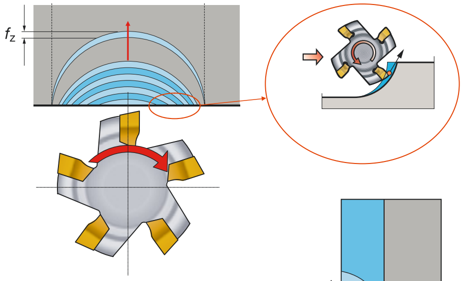
3) Entry into the workpiece should be programmed carefully until the cutter is in full cut by one of the following methods:

Roll on entry

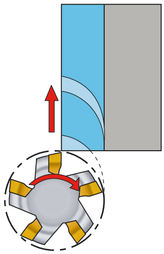
Prog. rad. = $\left(\frac{D_e}{2}\right) + 2$



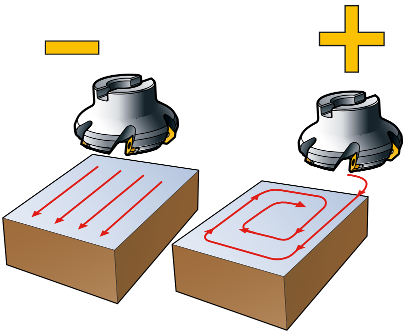
Reduced feed on entry



Thick chip on exit of cut until cutter is fully engaged

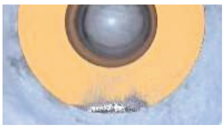


Also when milling large workpiece surface areas, select tool path to keep the milling cutter in full contact rather than perform several parallel passes. When changing direction, include a radial tool path to keep cutter moving, avoiding dwell and chatter tendencies.



Below, worn inserts with the same cutting data and tool life in Waspalloy demonstrate the impact of keeping the cutter in contact with the workpiece.

Many entries/exits.



Constant contact.

Face milling with carbide inserts

Effect of insert style

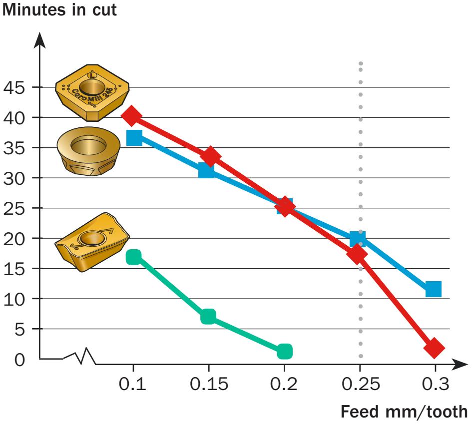
Chip thickness, cutting forces and tool life are affected by the choice of insert style when milling HRSA materials.

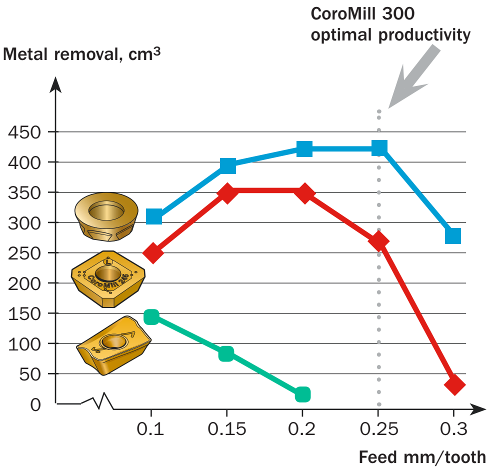
R245-050Q22-12H, z_n 5, R245-12T3E-ML GC2040

R300-063Q22-12H, z_n 7, R300-1240E-MM GC2040

R390-050Q22-11H, z_n 7, R390-11T308E-ML GC2040

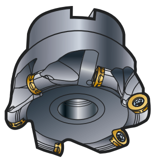
D_c 50 mm, v_c 30 m/min, a_e 32 mm (70%), a_p 2 mm
Material: Inconel 718 (42 HRC)



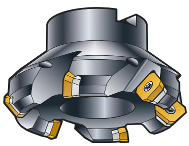


Recommendation – for general face milling in HRSA materials – use round insert cutters whenever possible to increase chip thinning effect – CoroMill 300.

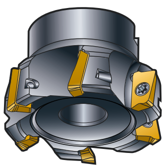
1st choice
Round inserts
CoroMill® 300



2nd choice
45°
CoroMill® 245

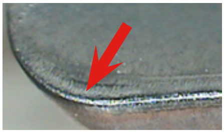


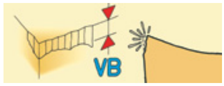
3rd choice
90°
CoroMill® 390



Typical wear patterns in HRSA milling

Flank wear





Rapid flank wear causing poor surface finish or out of tolerance.

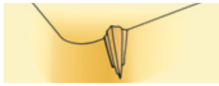
Cause: Cutting speed too high or insufficient wear resistance.

Remedy: Reduce cutting speed.
Select a more wear resistant grade.

Cause: Chip thickness too low.

Remedy: Increase feed.

Notch wear



Notch wear causing poor surface finish and risk of insert breakage.

Cause: Work hardening materials.

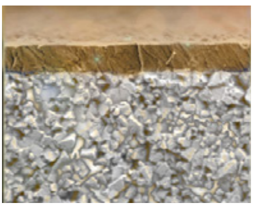
Remedy: Select round insert/reduce a_p.

The common causes of tool failure are excessive flank wear, notching at the cutting edge, and the inability to reach surface finish and accuracy requirements. Other contributing factors include excessive crater depth and destruction of the cutting edge by fracture. HRSA also tend to work-harden making subsequent passes more prone to notch wear.

First choice

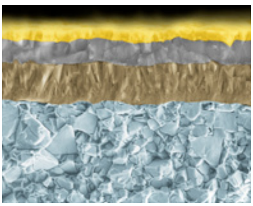
GC1030

PVD-TiAlN-coated carbide grade for milling of heat resistant super alloys at medium speeds. Good resistance to built-up edge and plastic deformation.



S40T

Combination of high toughness cemented carbide with a thin CVD coating resulting in a grade that withstands vibration and other difficult cutting conditions. Enables longer tool life and high security.



Complementary

S30T

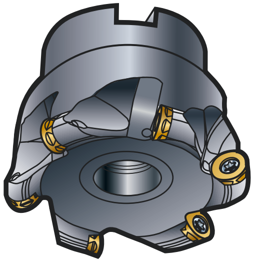
Combination of micro-grain carbide and a wear resistant PVD coating enables very sharp cutting edges that resist fatigue and micro-chipping. Enables higher cutting speeds and longer tool life.

GC2040

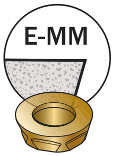
Tough MT-CVD coated carbide for milling of cast heat resistant alloys. Good resistance to high temperatures.

CoroMill® 300 – positive round face milling concept

Three main geometries are available for CoroMill 300 cutters when machining HRSA materials:



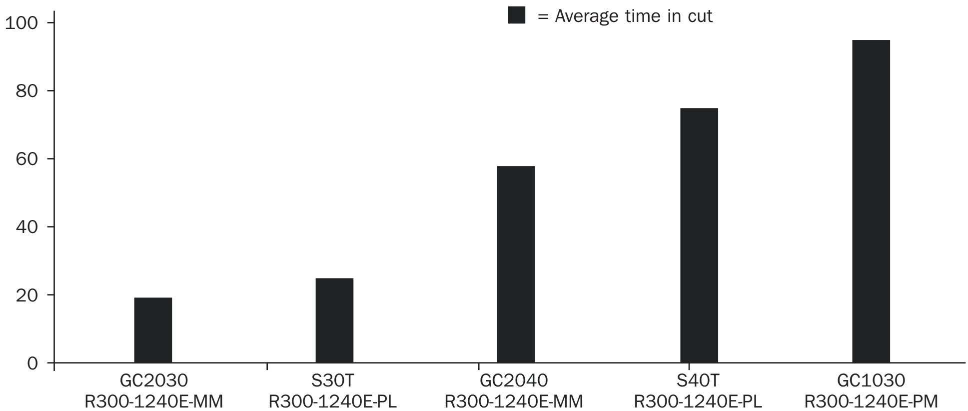
CoroMill® 300

Good conditions	**First choice**	Difficult conditions
		
High edge sharpness and precision. Positive geometry with edge reinforcement. First choice with S40T.	High edge sharpness and precision in combination with security. First choice for applications with S40T.	Good choice for general conditions. Reinforced edge security in grade GC2040 compared to E-PL and E-MM geometry.

Grade/geometry recommendations for milling HRSA

Tool R300-063Q22-12H
z_n 7, D_c 51 mm, v_c 30 m/min, a_e 36 mm,
a_p 1.5 mm, f_z 0.2 mm/tooth, h_{ex} 0.12 mm
Material: Inconel 718 (40 HRC)

Minutes in cut



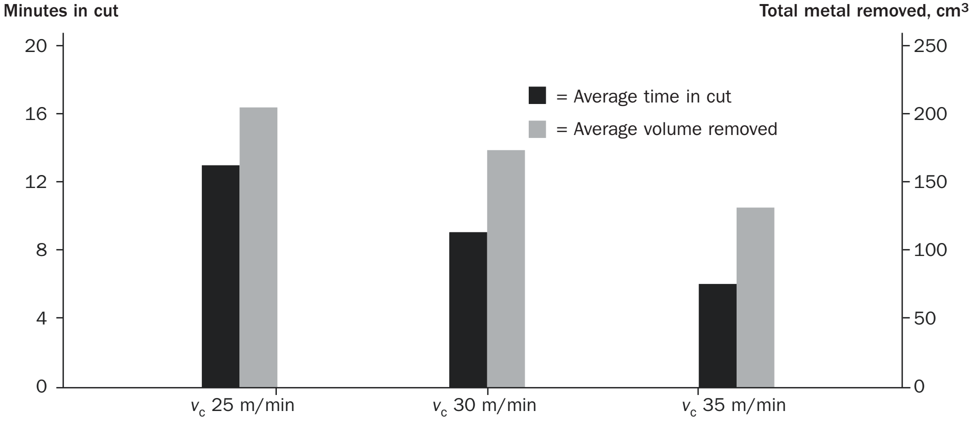
Recommendation - longest tool life and highest total of removed metal with the GC1030 E-PM insert.

Effect of cutting speed, v_c

Surface speed together with the material hardness are the most important factors in determining tool life when machining super alloys. Cutting temperatures for HRSA materials are typically 750 to 1020ºC. These temperatures are sufficiently high that oxidation and work-hardening become contributing factors to total tool wear.

The results below show that an increase of 5 m/min cutting speed reduces tool life and total material removed by approximately 30%.

Tool R300-063Q22-12H, insert R300-1240 E-PL S40T
z_n 7, D_c 51 mm, a_e 36 mm, a_p 2 mm,
f_z 0.24 mm/tooth, h_{ex} 0.18 mm,
Material: Inconel 718 (44 HRC)



Recommendation - for roughing using carbide inserts, speeds should not exceed (harder material choose the lower value):

First choice:

GC1030 max v_c 30-35 m/min
S40T max v_c 25-30 m/min

Complementary:

S30T max v_c 30-35 m/min
GC2040 max v_c 25-30 m/min

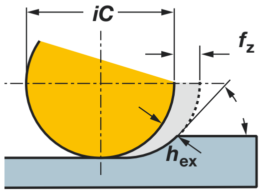
Effect of feed per tooth, f_z

As with other workpiece materials, feed and depth of cut are also important for tool life when machining HRSA materials. The feed per tooth value is calculated from the recommended maximum chip thickness (h_{ex}) – this deviates from the feed per tooth depending upon the following two factors:

The entering angle (κ_r) applied

For inserts with an entering angle of 45° and round inserts there is a chip thinning effect – this allows greater feed than inserts with κ_r 90°.

Max chip thickness $h_{ex} = f_z \times \sin \kappa_r$

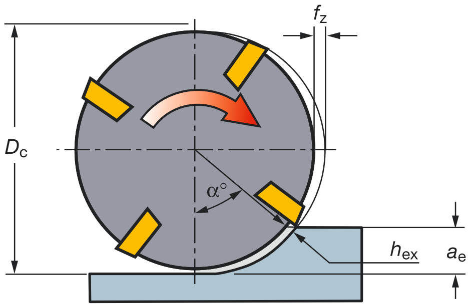


Round inserts

$$h_{ex} = f_z \times \sqrt{\frac{4a_p}{iC} - \left(\frac{2a_p}{iC}\right)^2}$$

Radial immersion, (a_e/D_c)

This is the width of the component engaged in cut in relation to the diameter of the cutter. It is the distance across the surface being machined covered by the tool. When the a_e is below half the diameter the maximum chip thickness is reduced relative to the f_z – therefore the feed can be increased.

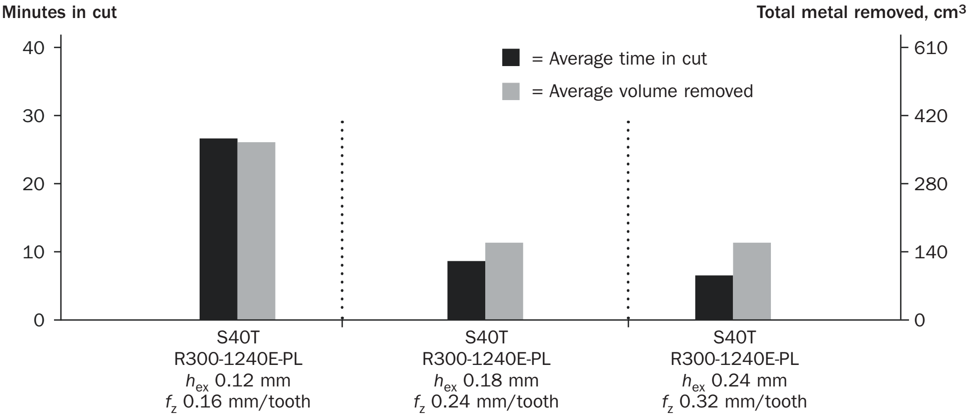


Depth of cut diameter ratio, a_e/D_c	Entering angle, α	Modification value, f_n	f_z mm/tooth for given chip thickness in mm: h_{ex} **0.1**	h_{ex} **0.15**	h_{ex} **0.2**
0.25	60°	1.16	0.12	0.17	0.23
0.2	53°	1.25	0.13	0.19	0.25
0.15	46°	1.4	0.14	0.21	0.28
0.1	37°	1.66	0.17	0.25	0.33
0.05	26°	2.3	0.23	0.34	0.46

Effect of geometry/f_z

The graph below shows the effect of the geometry depending upon the feed/tooth.

Tool R300-063Q22-12H, z_n 7, D_c 51 mm, v_c 30 m/min,
a_e 36 mm, a_p 2 mm,
Material: Inconel 718 (44 HRC)



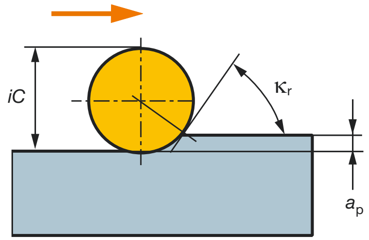
Recommendation – maximum productivity and metal removal rates were achieved using E-PL geometry at 0.16 mm feed/tooth (0.12 mm h_{ex}).

Geometry	Min (f_z)	Max (f_z)
E-PL	0.08 mm	0.18 mm
E-MM	0.1 mm	0.22 mm
M-PM	0.15 mm	0.3 mm

Effect of a_p/entering angle

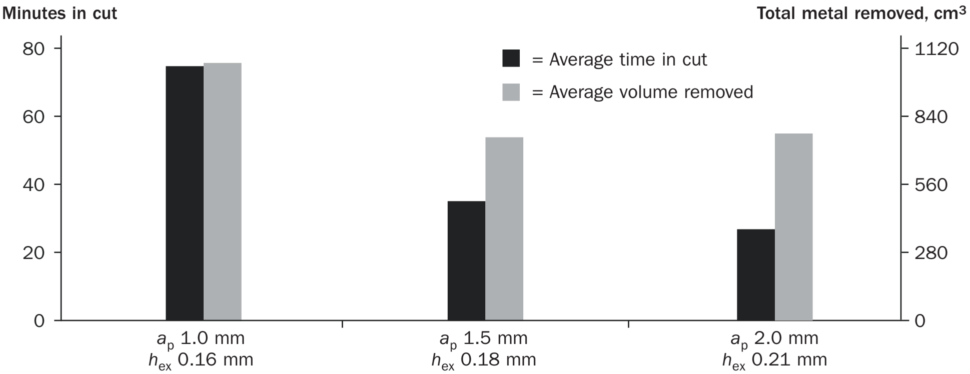
Selecting the correct cutting depth/entering angle when face milling in HRSA materials has an effect on tool life and productivity. Despite limitations on depth of cut when using round insert cutters these are still the most productive method when milling HRSA. Unlike typical titanium aerospace frame components HRSA components tend to have geometries that require high metal removal rates but not at high depth of cut, for example blades, casings etc. This allows optimisation of the entering angle at varying depth of cut.

Depth of cut to diameter ratio a_p/iC	Depth of cut for insert diameter, in mm				Entering angle κ_r
	8	**10**	**12**	**16**	
0.25	2	2.5	3	4	60°
0.2	1.6	2	2.4	3.2	53°
0.15	1.2	1.5	1.8	2.4	46°
0.1	0.8	1	1.2	1.6	37°
0.05	0.4	0.5	0.6	0.8	26°



Grade/geometry recommendations for milling HRSA

Tool R300-063Q22-12H, insert R300-1240E-MM 2040
z_n 7, D_c 51 mm, v_c 30 m/min,
a_e 36 mm, f_z 0.3 mm/tooth
Material: Inconel 718 (40 HRC)



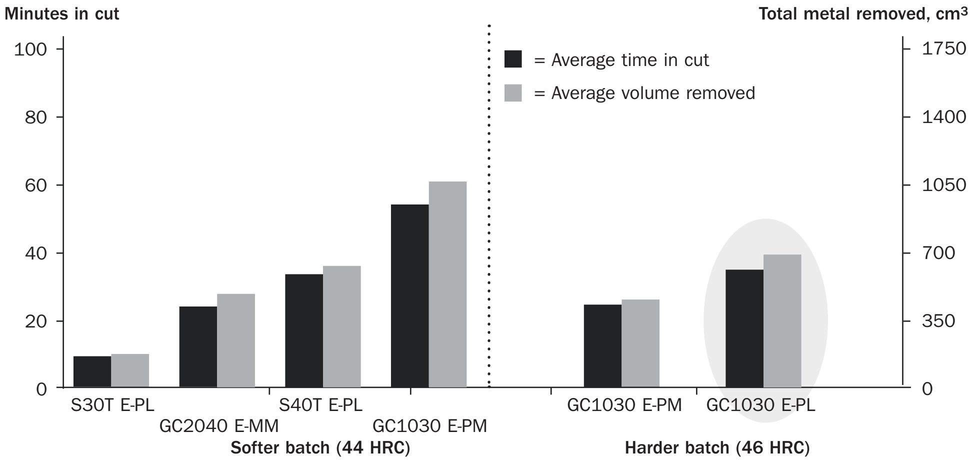
Recommendation – most productive depth of cut is 1 mm, this gives an entering angle of 33° when milling using a round insert.

Effect of hardness with grade and geometry, E-PL 1030

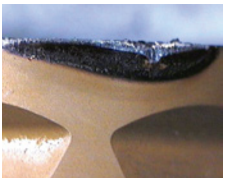
It can be seen that in harder materials new grade and geometry E-PL 1030 withstands the heat created during cutting much better than E-PL S40T.

The harder substrate combined with optimised micro geometry has better resistance to notch wear and plastic deformation.

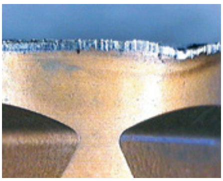
Tool R300-063Q22-12H,
D_c 51 mm, f_z 0.24 mm/tooth, h_{ex} 0.18 mm,
a_p 2 mm, a_e 36 mm, v_c 30 m/min
Material: Inconel 718



Hardness 40 HRC

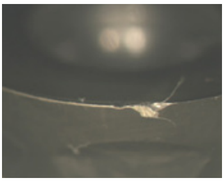


E-MM 2040 – 25 min



E-PM 1030 – 55 min

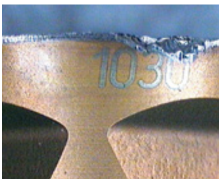
Hardness 44 HRC



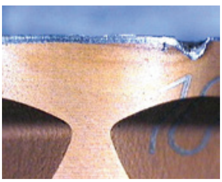
E-PL S30T – 10 min

E-PL S40T – 33 min

Hardness 46 HRC



E-PM 1030 – 24 min

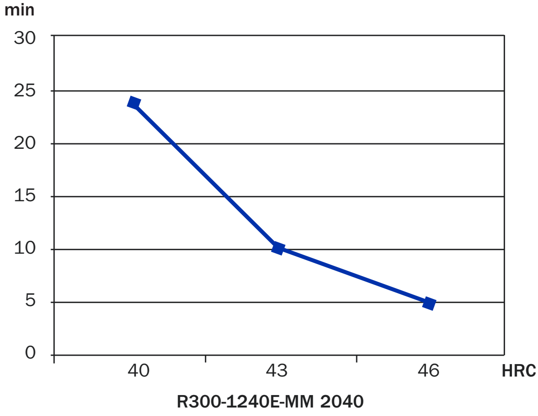


E-PL 1030 – 36 min

Effect of material hardness

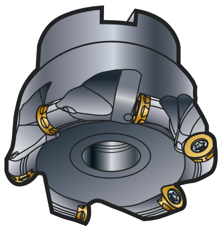
Small variations in material hardness will have a significant effect on tool life.

Tool R300-063Q22-12H,
D_c 51 mm, f_z 0.24 mm/tooth, h_{ex} 0.18 mm,
a_p 2 mm, a_e 36 mm, v_c 30 m/min
Material: Inconel 718



Summary – Face milling using carbide inserts in HRSA materials

- CoroMill 300 (round insert cutter) gives optimum performance.
- Grade S40T/GC1030 has the best performance.
- Use E-MM/E-PL geometry except at the highest chip loads (>0.3 mm/tooth) where M-MH offers better edge strength.
- Increase in cutting speed reduces tool life significantly; 25 m/min offers the best balance of tool life and productivity.
- Close pitch cutters with through spindle coolant supply give maximum productivity.
- Tool life reduces with an increase in axial depth of cut (a_p). Volume of material removed is less affected and running at a_p = 1 mm achieves 30% more volume removed as at 1.5 mm.
- Optimum parameters:



CoroMill® 300

Insert GC1030 E-PM	Insert S40T E-PL
0.2 mm/tooth feed rate	0.2 mm/tooth feed rate
25 m/min cutting speed	25 m/min cutting speed
1 mm axial depth of cut	1 mm axial depth of cut
Best toughness/security	**Best productivity/tool life**

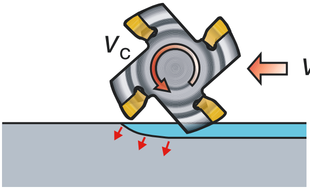
CoroMill® 390 – end milling/90 degree approach

90 degree entering angle is the least favourable design for milling in HRSA due to the high notching tendencies. However shoulder milling and profiling operations are demanded by the component features e.g. mounting brackets, casings, rings (scallops), circular interpolation of larger holes, slotting etc.

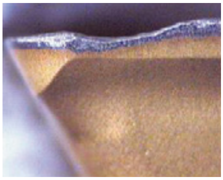
Grade, geometry and cutting data should be selected based upon the percentage radial immersion.

Edging/finishing – low radial immersion

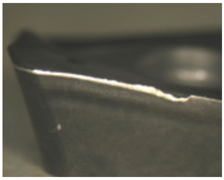
a_e = 12.5% of D_c.



Grade GC2040 E-ML



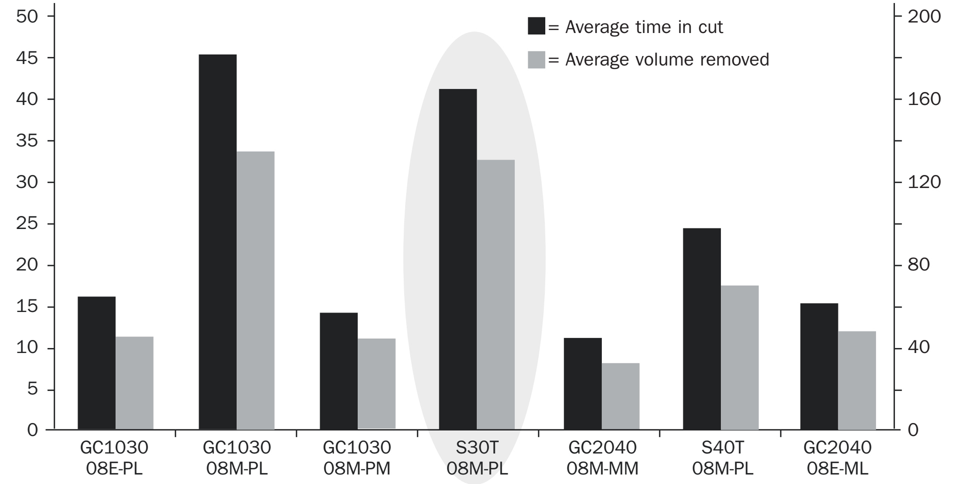
Grade S30T E-PL



Application – edging with low radial immersion

Tool R390-025A25-11H
z_n 4, D_c 25 mm, v_c 35 m/min, a_e 3.1 mm (12.5% of D_c),
a_p 5 mm, h_{ex} 0.07 mm, f_z 0.11 mm/tooth
Material: Inconel 718 (40 HRC)

Minutes in cut **Total metal removed, cm³**

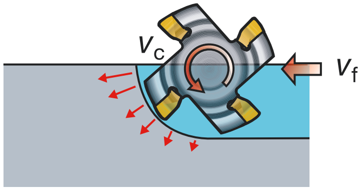


Recommendation – with a low immersion ratio (12.5%) grade and geometry recommendation is S30T M-PL.

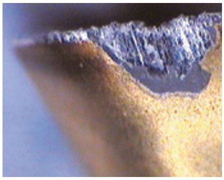
CoroMill® 390

End milling – increased immersion

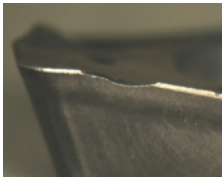
a_e = 25 to 75% of D_c.



Grade GC1030 M-PL

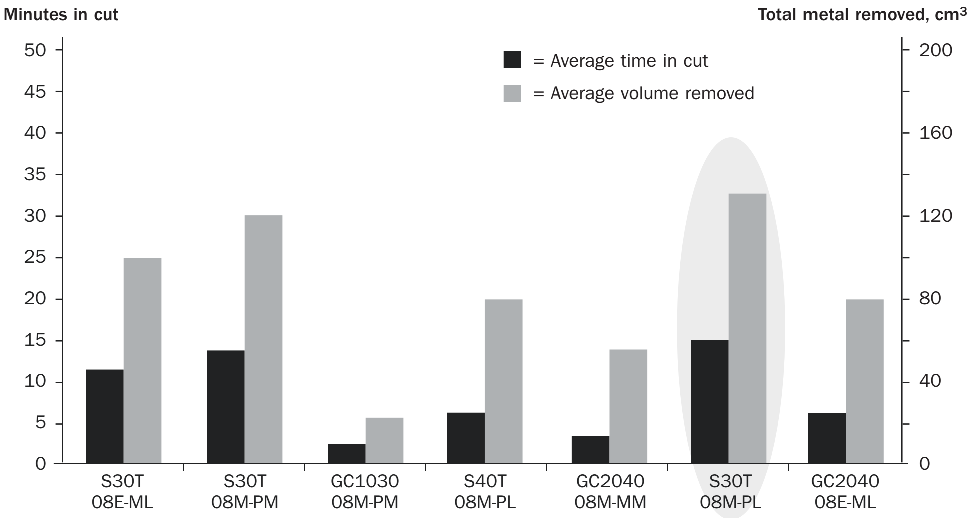


Grade S30T E-PL



Application – increased radial immersion/extreme conditions

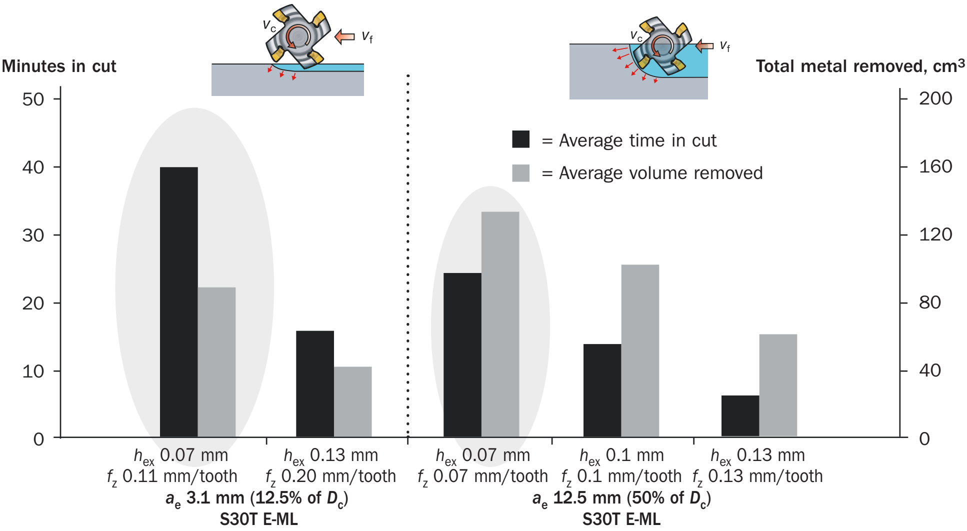
Tool R390-025A25-11H
z_n 4, D_c 25 mm, v_c 35 m/min, a_e 19 mm (75% of D_c),
a_p 5 mm, h_{ex} 0.07 mm, f_z 0.07 mm/tooth
Material: Inconel 718 (40 HRC)



Recommendation – highest tool life and immersion ratio at 75% was achieved using insert S30T M-PL.

Effect of chip thickness, h_{ex}

Tool R390-025A25-11H
z_n 4, D_c 25 mm, v_c 25 m/min, a_p 5 mm,
Material: Inconel 718 (44 HRC)



Recommendation

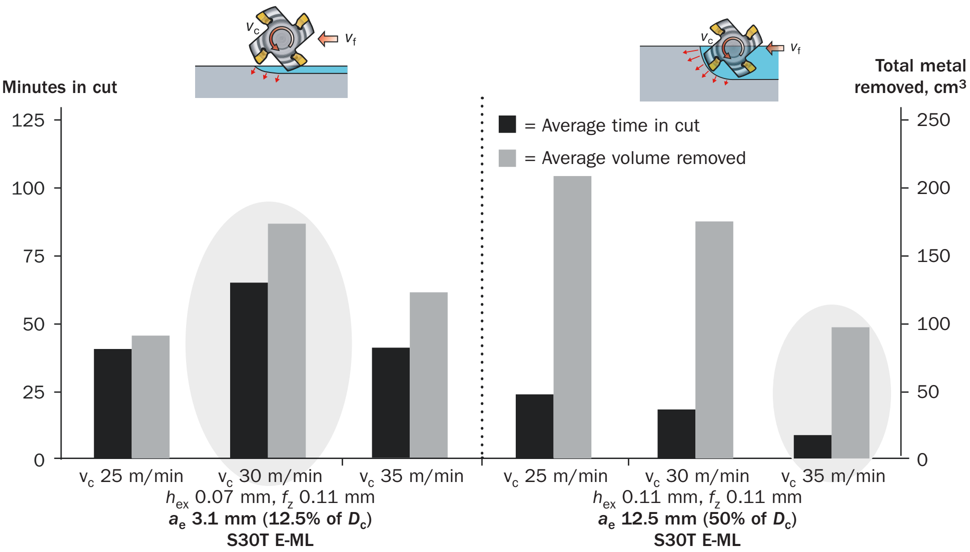
For low radial immmersion, grade/geometry S30T E-ML chip thickness 0.07 mm.

For increased radial immersion, grade/geometry S30T E-ML chip thickness 0.07 mm.

Effect of cutting speed, v_c

Careful consideration must be made when selecting cutting speed for end milling in HRSA materials. Reduce cutting speed if radial emersion is increased.

Tool R390-025A25-11H
z_n 4, D_c 25 mm, a_p 5 mm, Material: Inconel 718 (40 HRC)



Recommendation

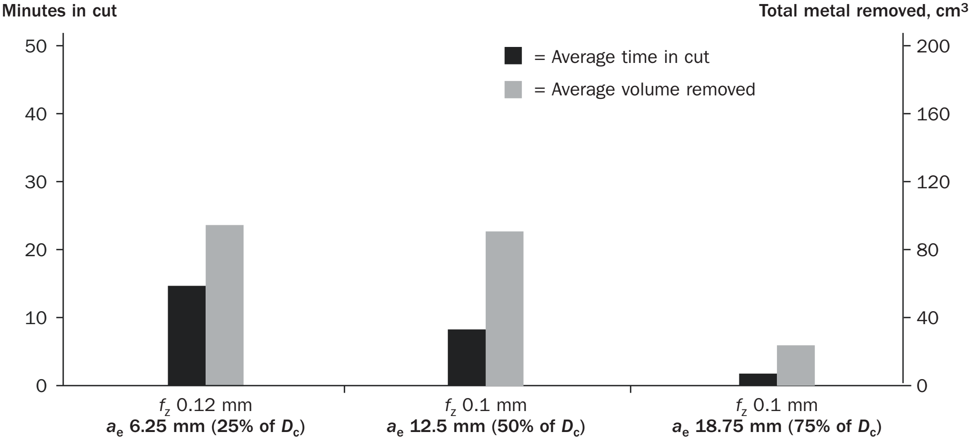
Highest tool life and total metal removed were achieved using S30T E-ML at 25 m/min.

Most productive insert at increased cutting speed S30T E-ML at 35 m/min, 50% immersion ratio and 0.1 mm feed/tooth.

Immersion ratio (a_e/D_c)

The diagram below shows that milling with large percentage radial immersion, has restricted tool life at the aggressive condition of 75% a_e/D_c.

Tool R390-025A25-11H, Insert GC2040 E-ML,
z_n 4, D_c 25 mm, v_c 30 m/min, a_e 12.5 mm (50% of D_c), h_{ex} 0.1 mm, a_p 5 mm,
Material: Inconel 718 (40 HRC)



Recommendation – recommended immersion ratio for insert under 50% giving maximum metal removal rates.

Summary – end milling/90 degree approach in HRSA materials

Edging/circular interpolation of existing hole

- Grade S30T performs better at lower immersion ratios.
- For grade S30T the geometry E-ML performs better on average regardless of the cutting conditions employed.
- a_e = 12.5% of D_c, f_z 0.11 mm (h_{ex} 0.07 mm)

CoroMill® 390

Increased immersion/slotting/ circular ramping

- At higher immersion ratios grade S30T E-ML has the best performance.
- a_e = 50% of D_c, f_z 0.1 mm (h_{ex} 0.1 mm)

Ceramic milling

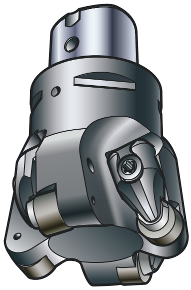
Ceramic cutting tool properties have a much higher resistance to heat compared to carbide tools, making it an excellent option for machining HRSA where high cutting temperatures are present.

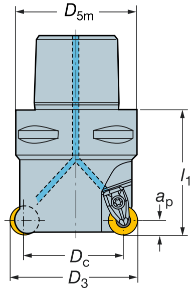
Ceramic milling typically runs at 20 to 30 times the speed of carbide, although at lower feed rates, which results in high productivity gains. They show a high tendency to notching, which is why round inserts are mainly used to ensure a low entering angle. Ceramics have a negative effect on the surface integrity and topography and are therefore not used when machining close to the finished component shape.

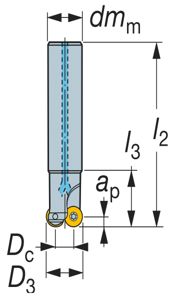
Due to the lack of toughness of ceramic material, round inserts are used to suppress the high notching tendency.

The main application for ceramic milling is machining of engine casings and oil drilling equipment due to the high metal removal rates over carbide inserts, combined with large stock removal.

Cutter programme – Please contact your local Sandvik Coromant representative for ordering.







S-R120R							
	D_3	Pitch, (X)		D_c	Size	D_{5m}	l_1
		Coromant Capto					
12	50	S-R120R-038C5-12X03	3	38	C5	50	70
	63	S-R120R-051C6-12X04	4	51	C6	63	70
	80	S-R120R-068C6-12X05	5	68	C6	63	72

Insert	Clamp	Key	Shim	Shim screw	Key for shim screw
RNGN 120700	5412 034-021	5680 049-01	-	-	-
RNGN 120400	5412 034-021	5680 049-01	176.1-851	3213 010-206	174.1-870

CoroMill 300C							
	D_3	Pitch, (X)		D_c	Size	l_1	l_2
		Coromant Capto					
9	36	R300C-036C3-09M	4	27	C3	40	
9	44	R300C-044C4-09M	4	35	C4	40	
12	36	R300C-036C3-12M	3	24	C3	50	
12	44	R300C-044C4-12M	4	32	C4	50	
12	54	R300C-054C5-12M	4	42	C5	50	
		Cylindrical shank					
6	20	R300C-020A16-06M	3	14			120
6	25	R300C-025A20-06M	4	19			160
6	32	R300C-032A25-06M	4	26			200
9	25	R300C-025A20-09M	3	16			160
9	32	R300C-032A25-09M	3	23			200
9	40	R300C-040A32-09M	4	31			200
12	32	R300C-032A25-12M	3	20			200
12	40	R300C-040A32-12M	4	28			200
12	50	R300C-050A32-12M	4	38			200
		Arbor					
12	40	R300C-040Q16-12M	4	28		50	
12	50	R300C-050Q22-12M	4	38		50	

Insert	Clamp	Key
RPNG 060300E	172.9-825-1	174.1-862
RPNG 090300E	172.9-826-1	174.1-863
RPNG 120400E	172.9-827-1	174.1-864

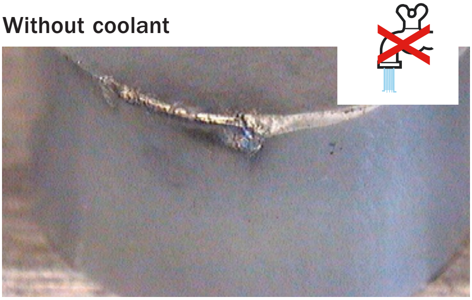
Coolant and ceramic milling

When milling using ceramic inserts, coolant should normally not be used. In most operations the use of coolant has a negative effect on tool life. This is due to the increase in thermal shock with cooling and heating of the cutting zone as the insert enters and then exits the workpiece. This increases the chance of top slice of the ceramic.

However, a small amount of MQL (Minimum Quantity Lubrication) could have a positive effect when reducing heat in the machining process.

Below is an example of the effect on wear/tool life of ceramic with and without coolant. Both inserts have been machined with the same cutting parameters and for the same amount of time.

Without coolant



With coolant

Effect of cutting speed, v_c

When milling in ceramics, due to intermittent cutting, it is a much cooler operation than turning. For this reason speeds of 700-1000 m/min when milling are adopted compared with 200-300 m/min for turning. The high cutting speed used when milling with ceramic inserts increases the temperature of the chip making the chip highly sheared.

The cutting speed should be balanced to create enough heat in the cutting zone but not too high to unbalance the ceramic. Too low cutting speed can result in top slice of the insert, too high cutting speed can result in insert failure.

Top slice

Ceramic cutter, Insert grade CC6060,
z_n 4, D_3 63 mm, v_c 700 m/min, a_e 32 mm,
f_z 0.1 mm/tooth, a_p 1.5 mm, Material: Waspalloy

Even flank wear

Ceramic cutter, Insert grade CC6060,
z_n 4, D_3 63 mm, v_c 1000 m/min, a_e 32 mm,
f_z 0.1 mm/tooth, a_p 1.5 mm, Material: Waspalloy

Recommendation – 1000 m/min cutting speed gives most balanced speed.

Effect of feed per tooth, f_z

As with round carbide inserts the chip thickness varies and depends upon the entering angle. With low a_p/iC ratios the feed can be increased in order to raise the chip thickness to the desired level.

Selecting the correct h_{ex} value when milling with ceramic inserts is critical. Always modify the feed depending upon the entering angle, max chip thickness (h_{ex}).

Top slice/flank wear

Ceramic cutter, Insert grade CC6060, z_n 4, D_3 63 mm, v_c 1000 m/min, a_e 32 mm, f_z 0.12 mm/tooth, a_p 1.5 mm, Material: Waspalloy

Result – feed/tooth too high resulting in excessive heat and top slice.

Even flank wear

Ceramic cutter, Insert grade CC6060, z_n 4, D_3 63 mm, v_c 1000 m/min, a_e 32 mm, f_z 0.1 mm/tooth, a_p 1.5 mm, Material: Waspalloy

Result – feed/tooth correct for application giving an even wear pattern.

Effect of a_p/entering angle

Selecting the correct cutting depth/entering angle when face milling in HRSA materials has an effect on the tool life and productivity. When milling with ceramic inserts the entering angle is critical due to poor notch resistance, performance is best with depth of cut a_p between 1.5 to 2.5 mm (κ_r = 40 to 50°).

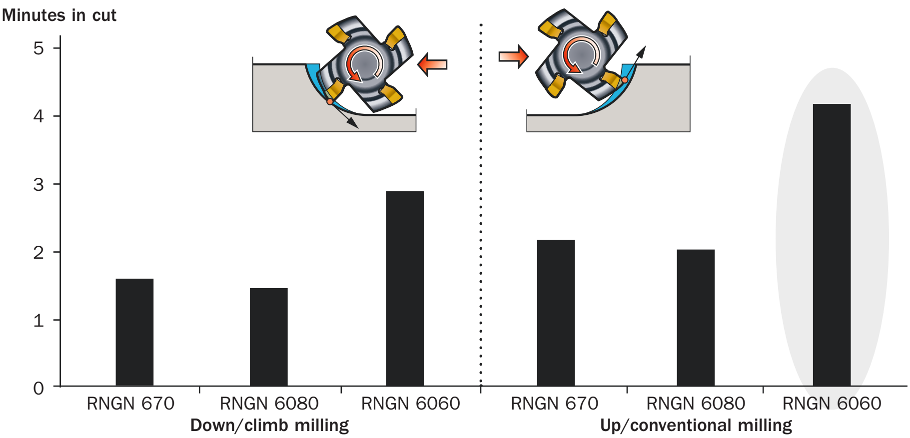
Ceramic versus carbide comparison – milling in Inconel 718

Maintain constant engagement where possible – soft entry into cut.

	Ceramic	Carbide
Application	Conventional/up milling	Climb/down milling
Coolant	Dry	Wet
Insert	RNGN 120700E 6060	R300-1240E-MM 2040
Cutting speed, v_c (m/min)	1000	30
Diameter, D_3 (mm)	63	63
Spindle speed, n (r/min)	5052	152
Feed rate, f_z (mm/tooth)	0.1	0.3
Number of teeth, z_n	4	6
Depth of cut, a_p (mm)	1.5	2
Radial immersion, a_e (mm)	35	35
Metal removal rate, Q (cm^3/min)	106	19
Tool life, (min)	3	25
Total material removed, Q_t (cm^3)	318	477

Grade choice and effect of milling direction

D_3 63 mm, z_n 4, v_c 1000 m/min, a_e 32 mm,
f_z 0.11 mm/tooth, h_{ex} 0.07 mm, a_p 1.5 mm (no coolant)
Material: Inconel 718 (40 HRC)



The diagram shows that:

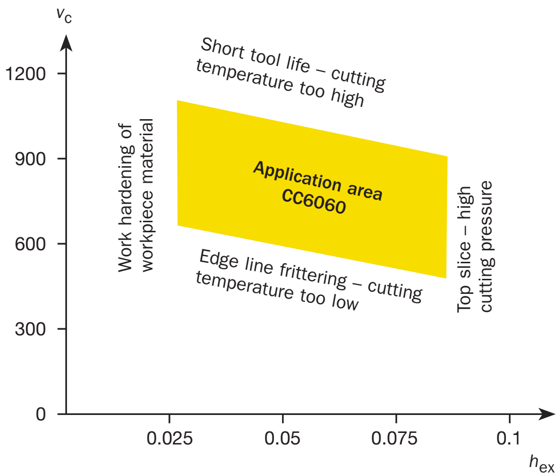
- New Sialon grade CC6060 gives the outstanding performance.
- Up/conventional milling provides a longer tool life and more consistent wear compared to down/climb milling. This is due to the reduced impact force on entering the material better suited to ceramic material.

Cutting data recommendation

The speed should be balanced to create enough heat in the cutting zone to plasticize the chip, but not so high as to unbalance the ceramic.

The feed, f_z, should be selected to provide a chip thickness, h_{ex}, which is high enough so as not to work-harden the material, but not so high as to cause edge frittering.

Higher feeds and depths of cut require a reduction in the cutting speed, v_c.

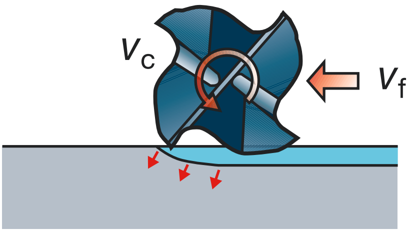


Solid carbide – CoroMill® Plura in HRSA machining

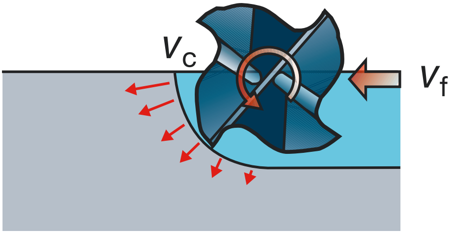
Due to the high hot hardness and toughness of HRSA they are one of the most difficult to machine materials placing great demands on the tool. The outcome is traditionally, low cutting speeds and hence lower productivity/higher machining costs.

High speed machining (HSM) techniques offer an effective way to increase productivity and to mill intricate and thin-walled components. The high feed rates do not allow as much heat to get transferred into the component, due to the short contact time, compared to conventional milling techniques.

However low radial cuts are required to keep the chip thickness small and allow for the higher feed rate.



Feed faster than heat propagation.

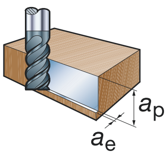


Traditional milling, time for heat propagation.

HSM uses high rpm and axial cut (a_p) but with only small radial engagements (a_e) and feed per tooth (f_z). This is possible due to:

Factor	Effect	Benefit
Thin chip thickness	Lower cutting force/deflection	Deeper axial cuts
Small arc of engagement	Reduced temperature at cutting zone	Higher speeds

This method requires a machine with high spindle speed and high feed dynamics – putting no extra demands upon rigidity.



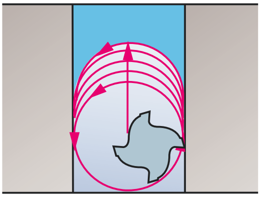
Processes using HSM techniques

Trochoidal milling

A roughing/high material removal method used when in a confined space or slot.

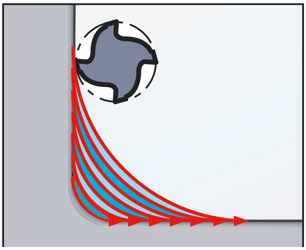
A continuous spiraling path feeding in the radial direction to form a groove or a profile.

It requires specialised programming and machine capabilities.



Slicing

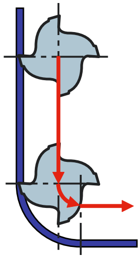
A semi-roughing technique used to produce a profile. Multiple passes to reduce the radial immersion. It requires a machine with high spindle speed and dynamic capability.



Profiling

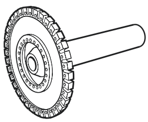
A finishing technique used to produce a finished profile e.g. flank milling which reduces the number of axial passes.

Requires a machine with high spindle speed and specialised programming techniques for simultaneous 5 axis for blisk/impellor machining.

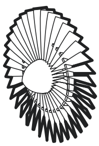


Components/features where HSM can be applied

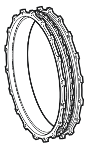
Some of the typical components that can be machined using CoroMill Plura tools and HSM techniques for example are blisks, impellers, turbines (LPT and HPT) and exhaust casings.



Slots – shafts



Blisk/impeller



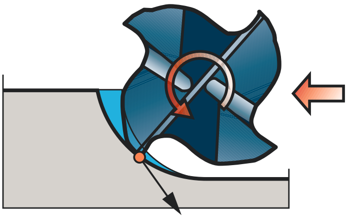
Scallops – casings

Application recommendations

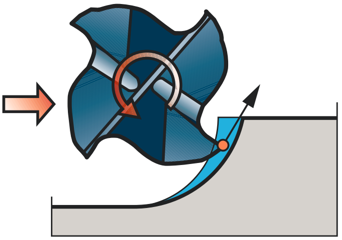
Down milling

It is almost always more favourable to apply down milling rather than up milling. When the cutting edge goes into contact in down milling, the chip thickness has its maximum value, in up milling the chip thickness is zero.

The tool life is generally shorter in up milling due to the fact that there is considerably more heat generated due to the rubbing action that takes place on entry. The radial forces are also considerably higher in up milling.



Down milling (climb milling).



Up milling (conventional milling).

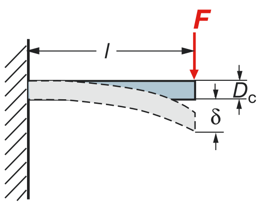
Avoid excessive deflection

Shallow radial cuts (a_e) should be applied to avoid excessive deflection of the cutting tool and to keep a high tolerance level and geometrical accuracies on the machined component.
It is important to use a tool with a maximum core diameter (higher bending stiffness).

l = overhang
D_c = tool diameter
F = radial force
δ = deflection
E = modulus of rigidity of the tool

$$\delta \approx \frac{F \times l^3}{E \times (\pi \times D_c^4)}$$

20% overhang reduction reduces tool deflection by 50%.
20% increased D_c (10 to 12 mm) reduces tool deflection by 50%.



Tool holding

One of the main criteria when deciding both the tool and holding device is to have as small a run out as possible. This keeps a uniform chip thickness on each cutting edge and hence has an even load distribution. The total indicator run out (TIR) should not be more than 10 microns.

A good rule of thumb is that 'For every 10 microns in added run out the tool life reduces by 50%'!

A CoroGrip® power chuck or shrink fit should be used due to:

Minimised run out

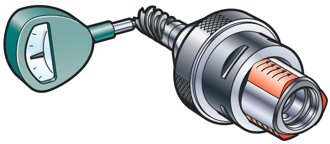
- increased tool life.

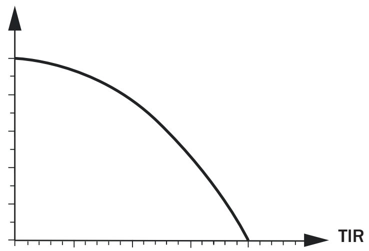
Stability

- reduced vibrations allowing higher depths of cut.

Clamping forces

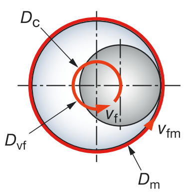
- resists pull out with high helix cutters.



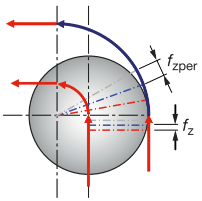


Programming feed rate (peripheral and central feed of the tool)

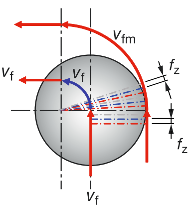
When programming with the feed applied to the tool centre, the feed must be reduced when producing an internal radius or a circular motion (G2 or G3) if not using radius compensation. Due to the fact that the periphery has to travel further than the tool centre for the same angular rotation.



Constant tool centre feed



Reduced feed for radius



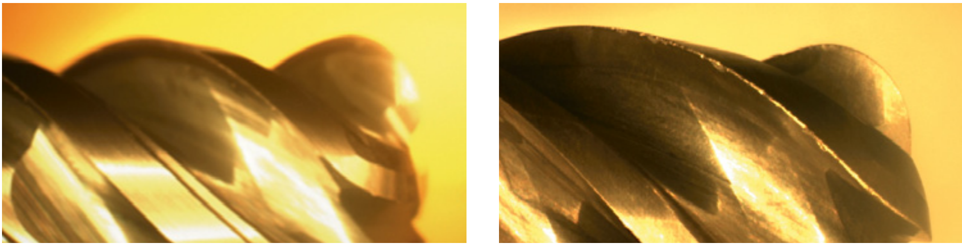
$v_{fm} = n \times f_z \times z_n$

$D_{vf} = D_m - D_c$

$v_f = \dfrac{D_{vf}}{D_m} \times v_{fm}$

Wear mechanisms

Typical wear observed on CoroMill Plura tools in the case of Inconel is micro chipping rather than flank wear. The edge line starts frittering before it leads to total cutting edge failure. The transition from micro chipping to failure is exponential, therefore once this wear is observed the tool should be immediately indexed. This can be monitored with the power/load gauge or by sound.



Tool R216.24-12050-AK26P 1620, v_c 75 m/min, h_{ex} 0.04 mm, a_p 10 mm, a_e 0.5 mm, f_z 0.1 mm/tooth, 15 minutes in cut, Material: Inconel 718 (42 HRC)

Optimised cutter design and cutting parameters

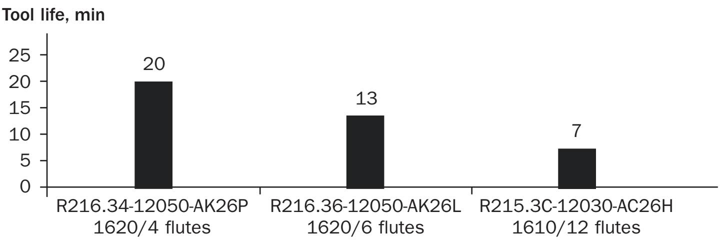
No of flutes (z_n)

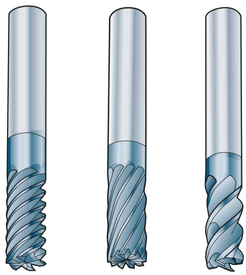
Inconel is a sticky material which can cause problems with clogging of the chip flutes. Therefore even though with HSM, employing low radial engagement with thin chips, extra close multi-fluted end mills (as used in hardened steel) are not recommended.

The thin chip produced can prove catastrophic if it sticks to the cutting edge and can lead to tool breakage. A balance should thus be struck between productivity (multi-flute) and security (lesser no of flutes). Four flutes were seen to be optimum in the dia range from e.g 8 to 12 mm.

The comparison chart for straight end milling is as shown.

Tool life vs number of flutes,
v_c 100 m/min, a_p 10 mm, h_{ex} 0.02 mm,
a_e 0.5 mm, f_z 0.05 mm/tooth
Material: Inconel 718 (42 HRC)





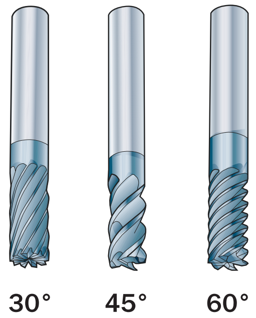
Helix angle

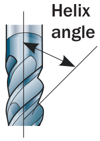
An end mill's helix angle is defined as the angle of the cutting edge relative to the centreline of the tool. The helix influences tool performance mainly by affecting chip flow and cutting forces by determining the length of engagement of the cutting edge for a given depth of cut.

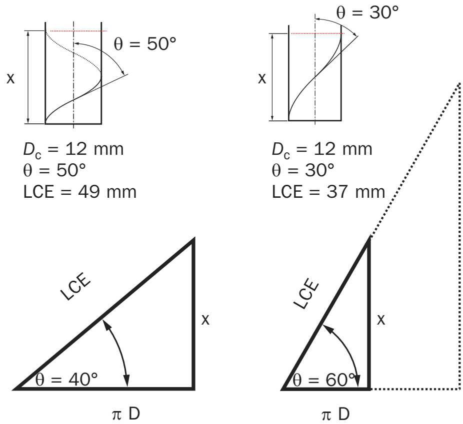
The greater the helix angle, the longer the cutting edge length allowing longer tool life and also giving a more gradual entry and exit into and from the workpiece. This lowers the radial forces that want to push the end mill and workpiece away from each other.

The result is a smoother machining action with less deflection. In most cases, high helix is recommended for finishing operations and a low helix in roughing because of the added strength.

A 50 degree helix is optimally suited for milling in Inconel when radial cuts (a_e) are less than 20% of the cutter diameter D_c, for example trochoidal, slicing and finishing.







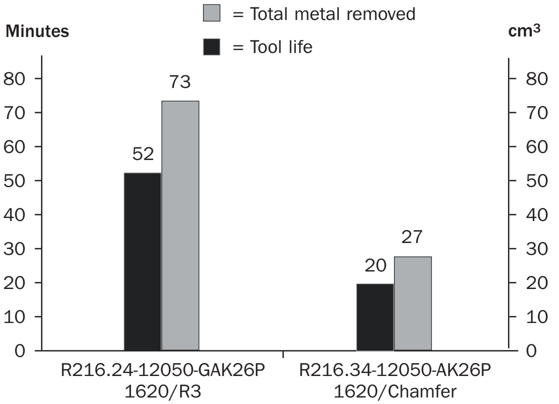
The diagram shows increased length of cutting edge (LCE) as the helix angle increases.

A high helix angle increases the tendency for the cutter to pull itself out of the chuck. A CoroGrip power chuck or shrink fit are required to resist this.

Corner geometry

The main wear observed in the case of HSM in Inconel is micro chipping at the cutting edge. A radius end mill, due to increased strength is always better compared to a chamfer or sharp corner.

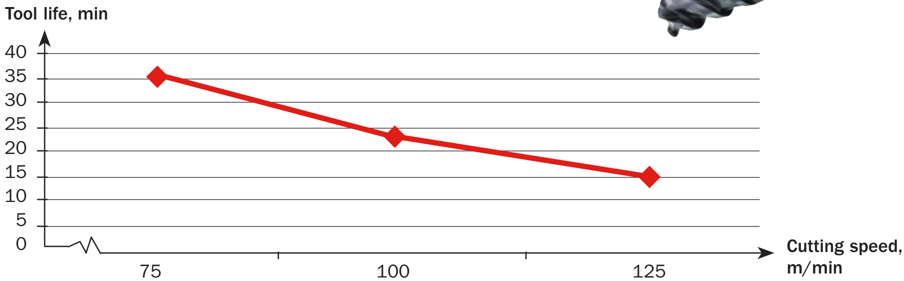
Comparison between chamfer and radius end mill
v_c 100 m/min, a_p 10 mm, h_{ex} 0.02 mm,
a_e 0.5 mm, f_z 0.05 mm/tooth
Material: Inconel 718



Cutting speed (v_c)

Because of the relatively low radial engagements in HSM one can make use of higher than normal cutting speed (v_c), 75 to 100 m/min gives the best balance between productivity and tool life. The chart shows the difference in performance relative to cutting speed.

Tool life vs cutting speed
R216.24-12050-GAK26P 1620, a_p 10 mm,
a_e 0.5 mm, h_{ex} 0.02 mm, f_z 0.05 mm/tooth
Material: Inconel 718



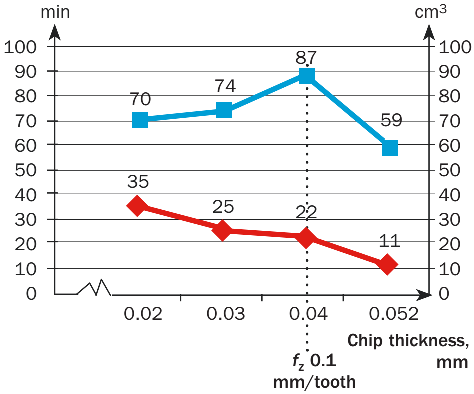
Chip thickness (h_{ex})

The low radial engagement reduces the chip thickness compared to the feed per tooth. Using the optimised chip thickness is pivotal in optimisation of finishing, slicing or trochoidal milling.

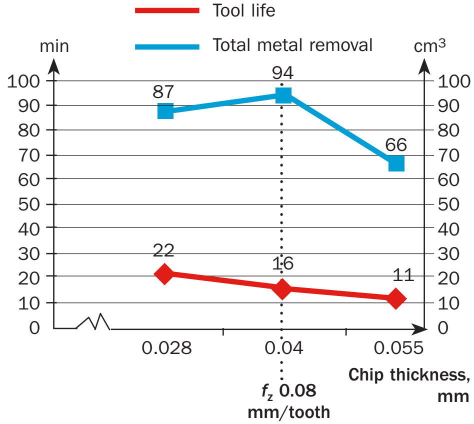
It can be seen in the diagram that a reduction of chip thickness reduces the material removed due to rubbing rather than cutting.

Equally the tool life drops by as much as 50% when the chip thickness is increased from 0.04 to 0.052 mm (25%). The best results are achieved at 0.04 mm thickness.

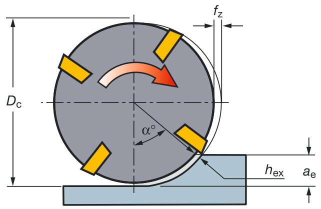
Tool life vs chip thickness – a_e 0.5 mm (4% of D_c)
R216.24-12050-GAK26P 1620
v_c 75 m/min, a_p 10 mm,
Material: Inconel 718



Tool life vs chip thickness – a_e 1.0 mm (8% of D_c)
R216.24-12050-GAK26P 1620
v_c 75 m/min, a_p 10 mm
Material: Inconel 718



The chip thickness is a factor affected by the feed per tooth and the angle of approach (radial engagement and diameter of cutter).



Each cutting edge design has an optimum chip thickness for a particular operation/material (0.04 mm for CoroMill Plura in Inconel). The feed rate selected should be that which gives the optimum feed rate for the relative radial immersion (a_e).

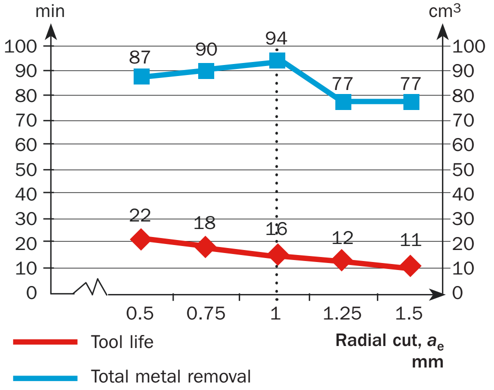
Depth of cut to diameter ratio a_e/D_c	Depth of cut for cutter diameter, mm				Entering angle α	Feed modification	Feed f_z for 0.04 h_{ex}
	8	**10**	**12**	**16**			
20.0%	1.6	2	2.4	3.2	53°	1.3	0.05
17.5%	1.4	1.75	2.1	2.8	49°	1.3	0.05
15.0%	1.2	1.5	1.8	2.4	46°	1.4	0.06
12.5%	1	1.25	1.5	2	41°	1.5	0.06
10.0%	0.8	1	1.2	1.6	37°	1.7	0.07
7.5%	0.6	0.75	0.9	1.2	32°	1.9	0.08
5.0%	0.4	0.5	0.6	0.8	26°	2.3	0.09
2.5%	0.2	0.25	0.3	0.4	18°	3.2	0.13

Radial cut (a_e)

For roughing applications it can be seen in the diagram that the maximum total material removed, when running with a constant speed and chip thickness, can be achieved with a_e = 1.0 mm.

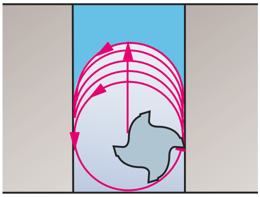
This equates to 8% a_e/D_c and should be used as a base when roughing operations are required.

Total metal removed and tool life vs radial cut
R216.24-12050-GAK26P 1620,
v_c 75 m/min, a_p 10 mm, h_{ex} 0.04 mm
Material: Inconel 718



Trochoidal milling

This is an established process in hardened steels and aluminum. The process puts low demands on stability and can be an extremely productive and secure method. This is advantageous especially where the components are large and costly calling for a secure productive solution.

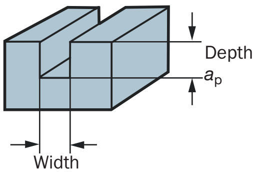


Parameter selection

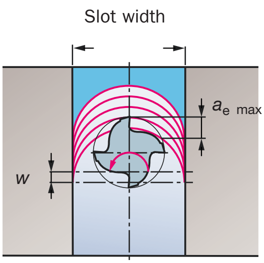
As can be seen, during trochoidal milling, the maximum width of cut – $a_{e\ max}$ is not equal to the programmed step over-'w'.

The maximum radial cut (a_e) max should not exceed 20% of the cutter diameter.

Start recommendations for trochoidal milling:



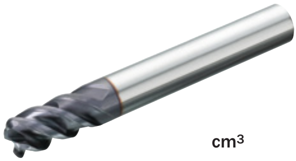
Cutter diameter, mm	D_c = 65% slot width
Step over, mm	w = 8% D_c
Axial cutting depth, mm	a_p = 1 to 1.5 x D_c
Cutting speed, m/min	v_c = 75
Feed/tooth, mm	f_z = 0.05

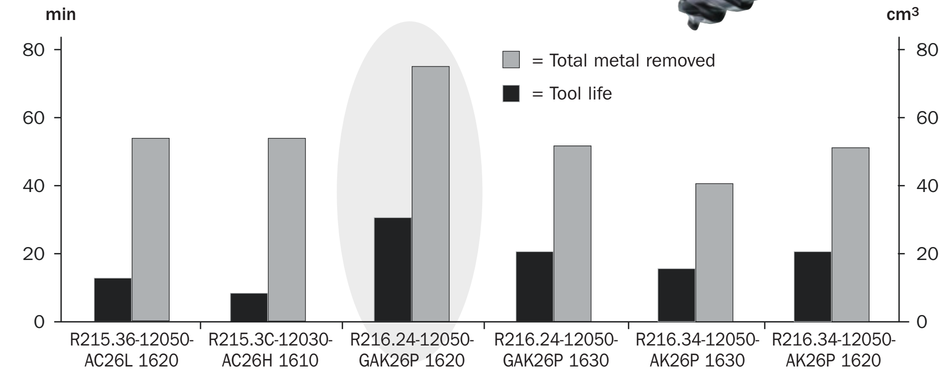


First choice recommendations for HSM

It can be seen in the diagram that R216.24-12050-GAK26P 1620 is the optimised cutter for HSM in Inconel.

Total metal removed and tool life vs radial cut
Material: Inconel 718
v_c 100 m/min, a_e 0.5 mm, a_p 10 mm,
f_z 0.05 mm/tooth, h_{ex} 0.02 mm





Recommended start cutting data for milling HRSA materials with CoroMill Plura

	v_c, m/min	a_p, mm	a_e, mm	h_{ex} mm	f_z mm/tooth	Tool life, min
Finishing	75	1.5 x D_c	0.25 to 0.5	0.04	0.1	22
Roughing	75	1.5 x D_c	8% D_c	0.04	0.08	16

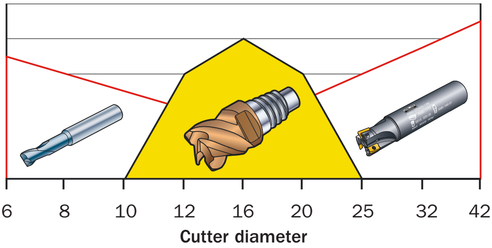
Depth of cut to diameter ratio a_e/D_c	Depth of cut for cutter diameter, mm				Entering angle κ_r	Feed modification	Feed f_z for 0.04 h_{ex}
	8	**10**	**12**	**16**			
20.0%	1.6	2	2.4	3.2	53°	1.3	0.05
17.5%	1.4	1.75	2.1	2.8	49°	1.3	0.05
15.0%	1.2	1.5	1.8	2.4	46°	1.4	0.06
12.5%	1	1.25	1.5	2	41°	1.5	0.06
10.0%	0.8	1	1.2	1.6	37°	1.7	0.07
7.5%	0.6	0.75	0.9	1.2	32°	1.9	0.08
5.0%	0.4	0.5	0.6	0.8	26°	2.3	0.09
2.5%	0.2	0.25	0.3	0.4	18°	3.2	0.13

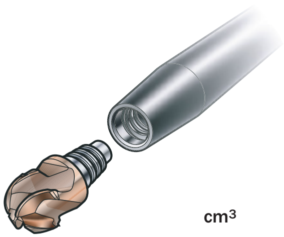
Select feed depending upon percentage radial depth of cut
e.g. roughing:
D_c 12 mm, a_e 0.9 mm, f_z 0.08 mm
e.g. finishing:
D_c 10 mm, a_e 0.25 mm, f_z 0.13 mm

Exchangeable-head – CoroMill® 316 in HRSA machining

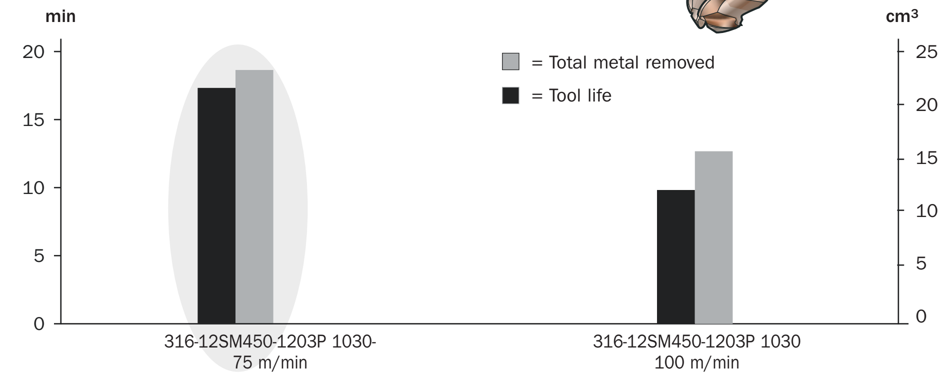
CoroMill 316 is a complement to our existing solid carbide programme. Insert geometry is based on CoroMill Plura tools.

- Roughing, semi finishing and finishing applications in general machining.
- Cost effective solution compared to CoroMill Plura for larger tool diameters
- Several different geometries and shanks
- Suitable for operations that demand long reach, for example deep pockets
- High accuracy in finishing operations with long overhangs





Total metal removed and tool life vs radial cut
Material: Inconel 718 (43 HRC)
v_c 75/100 m/min, a_e 0.5 mm, a_p 6.5 mm,
f_z 0.05 mm/tooth, h_{ex} 0.02 mm



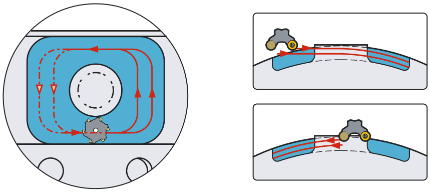
Recommended start cutting data for milling HRSA materials with CoroMill 316

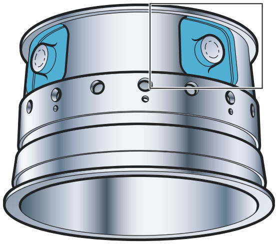
	v_c, m/min	a_p, mm	a_e, mm	h_{ex} mm	f_z mm/tooth	Tool life, min
Finishing	75	>6.5	0.25 to 0.5	0.02	0.05	17

Component/Feature based solutions

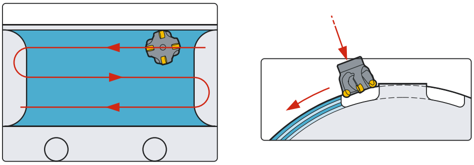
Engine casing

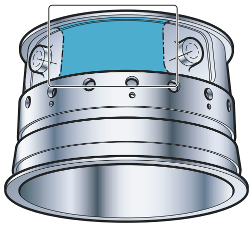
Feature	Rough mill sole plate
Material	Inconel 718
Cutter	R300-050Q22-12H
Cutter diameter, D_3	50 mm
Insert	R300-1204E-PL 1030
Number of teeth, z_n	5
Cutting speed, v_c	35 m/min
Spindle speed, n	223 r/min
Table feed, v_f	311 mm/min
Feed per tooth, f_z	0.25 mm
Depth of axial cut, a_p	2.5 mm
Depth of radial cut, a_e	38 mm
Metal removal rate, Q	27.7 cm^3/min



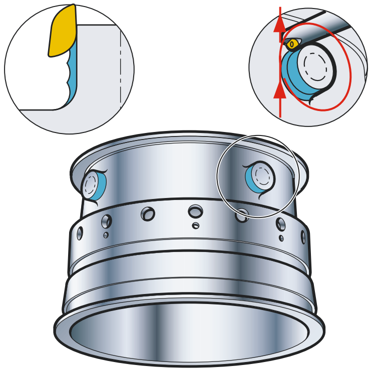


Feature	Rough mill sole plate band
Material	Inconel 718
Cutter	S-R210R-068C6-12X05
Cutter diameter, D_3	80 mm
Insert	RNGN 120700E 6060
Number of teeth, z_n	5
Cutting speed, v_c	1000 m/min
Spindle speed, n	4136 r/min
Table feed, v_f	2068 mm/min
Feed per tooth, f_z	0.1 mm
Depth of axial cut, a_p	2 mm
Depth of radial cut, a_e	50 mm
Metal removal rate, Q	207 cm^3/min

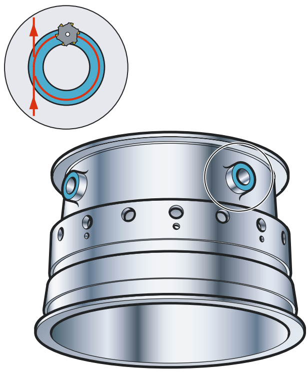




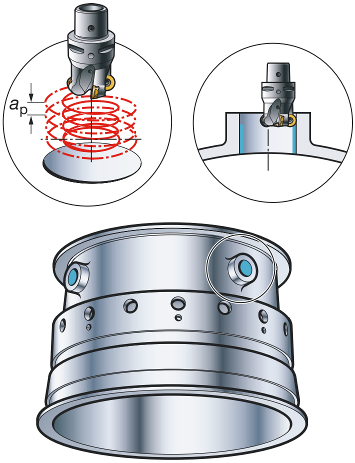
Feature	Machine radius
Material	Inconel 718
Cutter	R216-20B25-050
Cutter diameter, D_c	20 mm
Insert	R216-20T3E-M 2040
Number of teeth, z_n	2
Cutting speed, v_c	50 m/min
Spindle speed, n	447 r/min
Table feed, v_f	143 mm/min
Feed per tooth, f_z	0.15 mm
Depth of axial cut, a_p	3 mm
Depth of radial cut, a_e	3–5 mm
Metal removal rate, Q	5 cm^3/min



Feature	Finish mill ignitor face
Material	Inconel 718
Cutter	R300-050Q22-08H
Cutter diameter, D_3	50 mm
Insert	R300-0828E-PL 1030
Number of teeth, z_n	8
Cutting speed, v_c	45 m/min
Spindle speed, n	259 r/min
Table feed, v_f	622 mm/min
Feed per tooth, f_z	0.26 mm
Depth of axial cut, a_p	0.7 mm
Depth of radial cut, a_e	38 mm
Metal removal rate, Q	17 cm^3/min

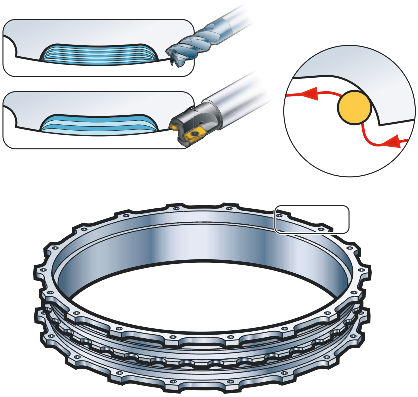


Feature	**Sole plate hole – Ø 59 mm – 20 mm deep**
Method	Circular ramping from solid
Material	Inconel 718
Cutter	CoroMill 300 R300-035C3-12H
Cutter diameter, D_3	35 mm
Insert	R300-1240E-MM 2040
Number of teeth, z_n	4
Cutting speed, v_c	25 m/min
Spindle speed, n	227 r/min
Table feed, v_f	76 mm/min
Feed per tooth, f_z	0.2 mm
Depth of axial cut, a_p	2 mm
Depth of radial cut, a_e	Full
Time	11 min
Metal removal rate, Q	5 cm³/min

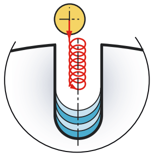


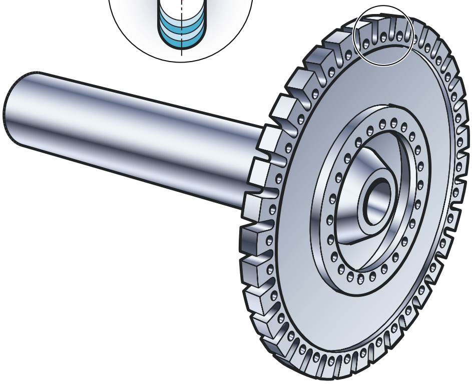
Rings – scallops

Feature	**Mill ring scallops**	
Material	Inconel 718	
Cutter	CoroMill 390	CoroMill Plura
Cutter diameter, D_c	16 mm	12 mm
Insert	R390-11T308M-PL 1030	R216.24-12050DAK26P 1620
Number of teeth, z_n	2	4
Cutting speed, v_c	30 m/min	75 m/min
Spindle speed, n	600 r/min	2000 r/min
Table feed, v_f	120 mm/min	637 mm/min
Feed per tooth, f_z	0.1 mm	0.08 mm
Depth of axial cut, a_p	5 mm	5 mm
Depth of radial cut, a_e	2 mm	1 mm
Metal removal rate, Q	1.2 cm³/min	3.2 cm³/min



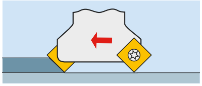
Feature	Slotting
Method	Trochoidal milling
Material	Inconel 718
Cutter	R216.24-08050 EAK19P 1620
Cutter diameter, D_c	8 mm
Number of teeth, z_n	4
Cutting speed, v_c	75 m/min
Spindle speed, n	3000 r/min
Table feed, v_f	200 mm/min
Feed per tooth, f_z	0.05 mm
Depth of axial cut, a_p	8 mm
Depth of radial pitch, w	0.67 mm
Time	5.64 min
Metal removal rate, Q	1.0 cm^3/min





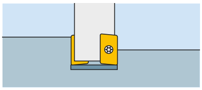
Feature	Blisk machining Roughing	Finishing
Method	Plunge milling	Point milling
Material	Inconel 718	Inconel 718
Cutter	R230.24-16000-AP096H1 Xceed (Gannet cutter)	Special conical ball nose end mill
Cutter diameter, D_c	16 mm	7 mm
Number of teeth, z_n	4	4
Cutting speed, v_c	50 m/min	75 m/min
Spindle speed, n	995 r/min	3410 r/min
Table feed, v_f	60 mm/min	546 mm/min
Feed per tooth, f_z	0.015 mm	0.04 mm
Depth of axial cut, a_p	4.8 mm	0.23 mm
Depth of radial pitch, w	16 mm	0.5 mm
Metal removal rate, Q	28 cm^3/min	–

Recommended start cutting data



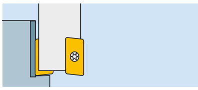
Face milling

Type	Application	Tool choice	Max. depth of cut a_p mm	Radial cut a_e mm	Cutting speed v_c m/min	Feed f_z mm/tooth
Round	Low to medium a_p	**CoroMill® 300** R300-0828E-PL GC1030 R300-1240E-PL G1030 R300-1240E-MM GC2040 R300-1648E-MM GC2040	1 2.5 2.5 4	70% of D_c	30 30 25 25	0.18 0.25 0.25 0.30
κ_r 45°	Medium to large a_p	**CoroMill® 245** R245-12T3E-ML GC2040	3		30	0.20
κ_r 90°	Against shoulder	**CoroMill® 390** R390-11T308E-ML GC2040	5		30	0.10



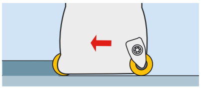
Closed slot milling

Cutter dia. D_c mm	Tool choice	Max. depth of cut a_p mm	Cutting speed v_c m/min	Feed f_z mm/tooth
16–40 25–40	**CoroMill® 390** R390-11T308E-ML GC2040 R390-170408E-ML GC2040	8 14	25–35 25–35	0.10 0.10
6–20	**CoroMill® Plura** R216.24-xx050-AKxxP GC1620	0.5 x D_c	75	0.05



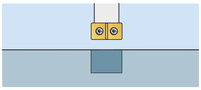
Side milling

Cutter dia. D_c mm	Tool choice	Max. depth of cut a_p mm	Radial cut a_e/D_c	Cutting speed v_c m/min	Max. chip thickness h_{ex} mm
16–40 16–40 25–40 25–40	**CoroMill® 390 end mill** R390-11T308M-PL GC1030 R390-11T308E-ML GC2040 R390-17408M-PL GC1030 R390-17408E-ML GC2040	8 8 14 14	12.50% 50–75% 12.50% 50–75%	35 35 35 35	0.10 0.07 0.10 0.07
6–20 6–20	**CoroMill® Plura** R216.24-xx050-AKxxP GC1620 R216.24-xx050-AKxxP GC1620	2 x D_c 2 x D_c	8%	75	0.04



Ceramic milling

Type	Application	Tool choice	Max. depth of cut a_p mm	Radial cut a_e mm	Cutting speed v_c m/min	Feed f_z mm/tooth
Round	Low to medium a_p	**Ceramic milling** RNGN 120700E CC6060	2.0	70% of D_c	1000	0.10



Open slot milling

Cutter		**CoroMill® 331** - side and face cutters		Cutting speed v_c m/min	Max. chip thickness h_{ex} mm
Width mm	Diameter mm	N331 insert size			
6–8	80–200	N/L/R331.1A-**04**-WL GC1030		35	0.07
8–10	80–200	N/L/R331.1A-**05**-WL GC1030		35	(N/L/R331.1A)
10–15	80–200	N/L/R331.1A-**08**-WL GC1030	RCHT 10T3M0-PL GC1030	35	
15–20.5	100–315	N/L/R331.1A-**11**-WL GC1030	RCHT 1204M0-PL GC1030	35	0.12 (RCHT)
20.5–26.5	160–315	N/L/R331.1A-**14**-WL GC1030	RCHT 1606M0-PL GC1030	35	

Hole making in HRSA alloys

Hole types and hole making methods

Hole making in HRSA can be split up into 5 distinct areas:

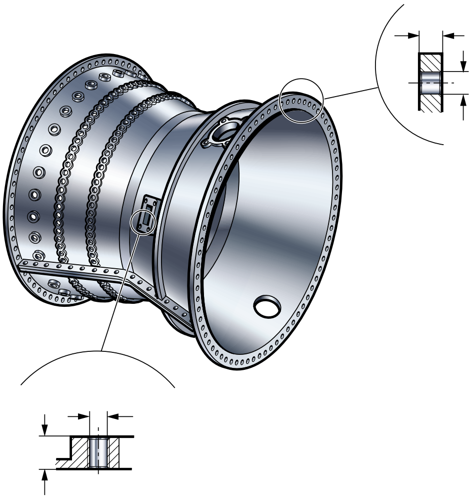
> Ø 16 mm through holes for assembly

In aerospace engines: casings, flanges, rings, discs, shafts, etc usually have identical, relatively small holes drilled in large numbers. These are often in diameters and with limitations suited to solid carbide drills.

For critical parts the hole is often made in up to 5 steps to guarantee quality and consistency.

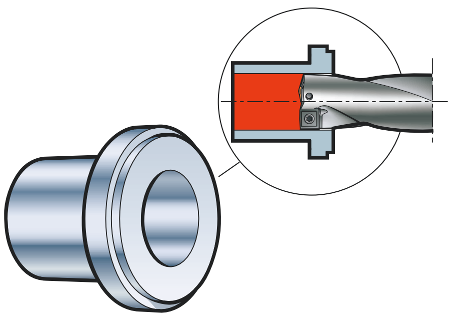
1. drilled
2. bored (using a sized milling tool) to make concentric
3. finish bored (using a sized milling tool)
4. chamfer front
5. chamfer back

This is one of the final machining operations so reliability is paramount, and as there can be hundreds of holes on just one component this can naturally be a lengthy process.



Ø 12 to Ø 60 mm through holes in stable components

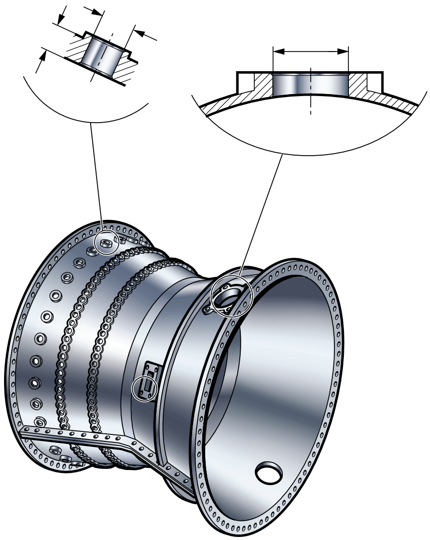
Oil and gas, bearing industry and mechanical engineering where the components are stable, indexable insert drills are used as the first roughing operation on either lathes or machining centres. Subsequent operations can include boring and turning.



Ø 20 to Ø 80 mm through holes in thin wall or unstable components

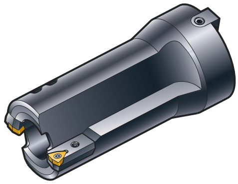
Aerospace casings, which are large thin wall components with large diameter boss holes, use circular ramping to produce the hole.

This method, although not as quick as drilling, has the advantages of producing low axial cutting forces providing the most reliable process.



Ø 60 to Ø 110 mm through holes up to 4 x diameter deep

For oil and gas and bearing industry, where tubes and rings are required, trepanning on a lathe is used as a method to save the core for other components due to the high material cost. This method also reduces the power required and production time. For deeper holes the bar is trepanned from both sides.

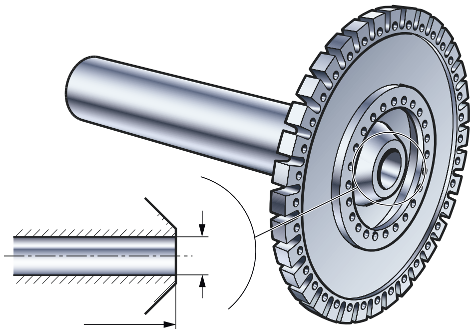


Deep holes – >10 x diameter

Oil and gas and aerospace engine shafts have deep holes. Indexable deep-hole drills are used which require a bushing or pre-machined hole to start.

After drilling, either damped silent boring bars or special boring heads with support pads are used to finish the bore.

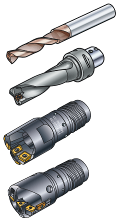
Traditionally this operation has used special deep-hole drilling/boring machines, however, these are now being transferred onto mill-turn or multi-task machines.



Tools for hole making

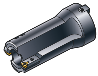
Drilling

Solid carbide		
CoroDrill Delta-C R846		hole dia 3.0 to 20.0 mm
Indexable insert		
CoroDrill 880		hole dia 14.0 to 63.5 mm
Deep hole		
CoroDrill 800		hole dia 25 to 65 mm
T-Max 424.10		hole dia 63.5 to 130* mm



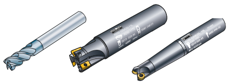
Trepanning

T-Max U 416.7		hole dia 60 to 110* mm



Circular ramping from solid

CoroMill Plura R216.34		
CoroMill 390-11	tool dia 16 to 40 mm	hole dia >28 mm
CoroMill 300-12	tool dia 25 to 80 mm	hole dia >38 mm

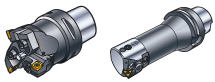


Circular milling – pre-drilled hole

Finish diameter		
CoroMill Plura	tool dia 4 to 20 mm	hole dia >5 mm
CoroMill 390	tool dia 16 to 40 mm	hole dia >20 mm
Back chamfering		
CoroTurn XS +		
Coromant Capto		hole dia >7 mm
CoroMill 327		hole dia > 11.7 mm
U-Max chamfering		hole dia >27 mm
Thread milling		
CoroMill Plura	tool dia 6 to 20 mm	hole dia ≥M4

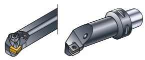
Boring (tool rotating)

CoroBore 820 – roughing		hole dia 35 to 260 mm
CoroBore 825 – finishing		hole dia 23 to 167 mm



Boring (component rotating)

CoroTurn RC	bar dia 25 to 50 mm	hole dia >32 mm
CoroTurn 107	bar dia 6 to 40 mm	hole dia >8.5 mm
CoroTurn SL	bar dia 16 to 60 mm	hole dia >20 mm



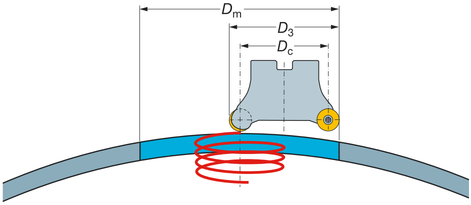
Silent Tools dampened boring bars

CoroTurn SL	bar dia 16 to 60 mm	hole dia >20 mm
CoroTurn SL-QC	bar dia 80 to 300 mm	hole dia >100 mm

*Larger diameter on request.

Circular ramping from solid

Milling of holes is a flexible process able to produce a range of hole sizes with each cutter. It produces low axial cutting forces and copes well with interrupted entrances and exits which are a problem when machining into curved surfaces such as casings.



Cutter and hole diameter

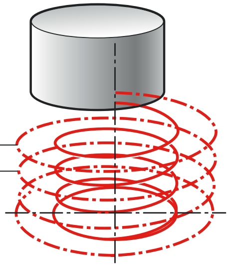
The ramping process requires a cutter which is capable of axial cutting. The diameter selection is very important when using cutters which are not centre cutting. The diameter of the cutter, minus the insert radius, should not exceed half the hole diameter. This is to ensure that there is no 'pip' remaining.

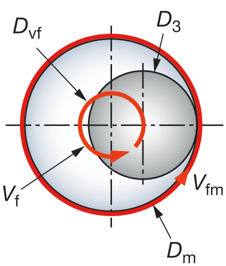
$$\text{Max } D_3 = \frac{D_m}{2} + 0.5iC$$

$$\text{Min } D_m = (D_3 - 0.5iC) \times 2$$

Pitch

The axial pitch (a_p) per revolution is determined by the max depth of cut limitation for the cutter concept (0.15 x *iC* for round inserts).





Tool centre feed diameter – D_{vf}

$D_{vf} = D_m - D_3$

Example

D_m	= 58	a_p	= 2
D_3	= 35	z_n	= 4
v_c	= 25	n	= 227
f_z	= 0.2	$\mathbf{v_{fm}}$	**= 182**
D_{vf}	= 18	$\mathbf{v_f}$	**= 76**

Feed rate

The feed rate must be reduced in internal applications due to the periphery of the tool moving faster than the centre line of the tool.

Programming of the feed rate (mm/min) on most milling machines/CAM systems is based on the centre line of the spindle requiring a manual recalculation.

Programmed feed rates

- v_{fm} – when using radius compensation
- v_f – when using the tool centre feed

$$v_{fm} = n \times f_z \times z_n$$

$$v_f = \frac{D_{vf}}{D_m} \times v_{fm}$$

Circular milling of existing hole

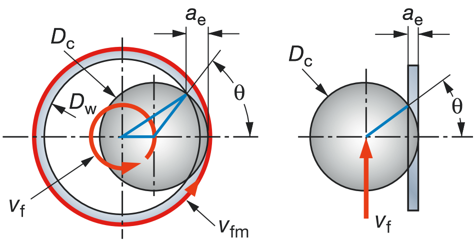
Hole diameters can be finish machined using circular milling as an alternative to boring depending on the surface finish requirement. The feed rate (v_f) must be reduced compared to straight line milling due to that in internal applications.

- The periphery of the tool will be moving faster than the centre line of the tool. Programming of the feed rate (mm/min) on most milling machines/CAM systems is based on the centre line of the spindle.
- The radial engagement (a_e) increases compared to that of a straight cut increasing the chip thickness, h_{ex}.
- The effect of both of these factors is increased with the increase in cutter diameter relative to the hole size.
- The correct feed reduction compared to straight cutting can be selected from the diagram.
- Allowing for stability use $D_c = 0.4 \times D_m$ and reduce feed by 50% of normal.

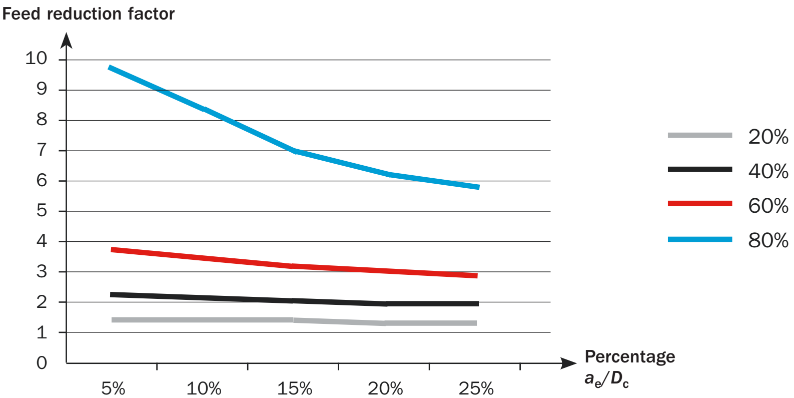
$$f_z = \frac{f_z}{\sin\theta}$$

$$v_{fm} = n \times f_z \times z_n$$

$$v_f = \frac{D_{vf}}{D_m} \times v_{fm}$$



Tool centre feed reduction factor for given cutter dia. hole dia. ratio (D_c/D_m) and constant chip thickness.

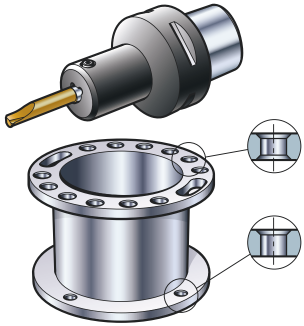


Back chamfering/deburring

When machining aerospace engine components, one tricky but often neglected operation is deburring holes.

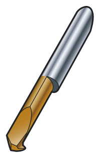
By using a CoroMill 327 or CoroTurn XS back chamfering tool it is possible to increase productivity while providing a repeatable, mechanised, safe process – eliminating the need for manual deburring and special tools.

The CoroTurn XS tool must be used with a Coromant Capto adaptor to provide the correct clearance when used in rotating applications.



CoroTurn® XS

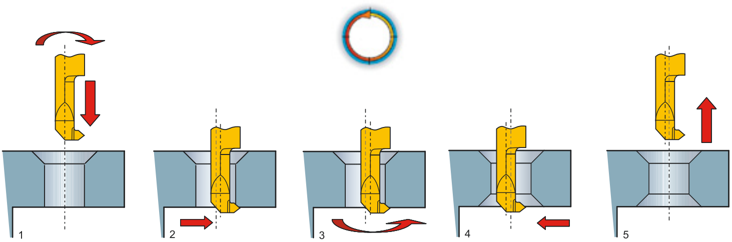
Adaptor	Insert	Min hole size	Max hole depth
C4-CXS-47-05	CXS-05T045-20-5215R CXS-05T045-20-5220R	7	15 20
C4-CXS-47-06	CXS-06T045-20-6220R CXS-06T045-20-6225R	8	20 25
C4CXS-47-07	CXS-07T045-20-7220R CXS-07T045-20-7240R	9	20 40



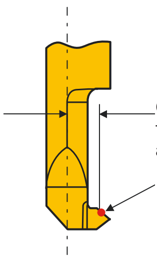
CoroMill® 327

Size	Insert	Min hole size	Max hole depth
06	327R06-12045-CH	11.7	40
12	327R12-20045-CH	21.7	85

Programming sequence method



1. Position the cutter centrally over the drilled hole with the cutter rotating (v_c 75) and move axially to flange depth (Z = flange height – chamfer size).
2. Feed cutter to engage with radius compensation (Y = hole radius).
3. Interpolate 360° (f_z 0.1 mm).
4. Feed back to hole centre.
5. Retract cutter.

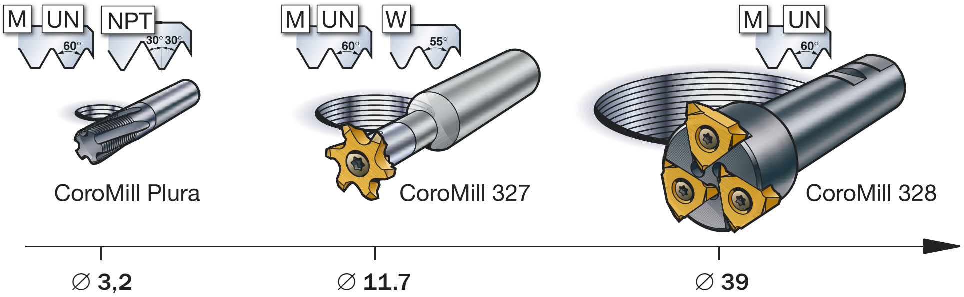


Thread milling

Many holes in a typical aerospace engine casing require threads. Machining the thread in smaller holes can prove to be a difficult operation. This can be performed using a tap to machine the thread. However in HRSA materials this can cause problems with chip jamming resulting in tap breakage and ultimately scrapping of an expensive component.

The main options for thread milling using Sandvik Coromant tools are single-point threading with CoroMill 327 and CoroMill 328, and multi-point threading with CoroMill® Plura.

All tools produce different threads of the same pitch with one tool.



	CoroMill® Plura	CoroMill® 327	CoroMill® 328
Pitch	0.7 – 3 mm 28 – 10 t.p.i.	1 – 4.5 mm 24 – 5 t.p.i.	1.5 – 6 mm 16 – 4 t.p.i.
Cutter dia (D_c), mm	3.2 – 19	11.7 – 21.7	39 – 80

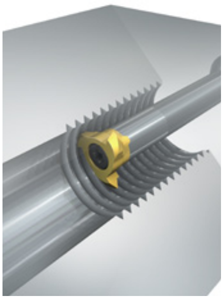
Single-point threading

CoroMill® 327

Designed for holes over 12 mm, CoroMill 327 offers inserts for metric, UN and Whitworth threads.

The front-mounted inserts are positioned in grooves for secure mounting, and through-tool coolant aids chip evacuation, giving secure and continuous performance. CoroMill 327 is available in versatile grade GC1025, for all material types.

CoroMill 327 has Weldon shanks, steel or solid carbide.



CoroMill® 328

For larger holes over 39 mm, CoroMill 328 offers inserts for metric and UN threads. Inserts are pocket-mounted for safe and stable positioning, with 3 cutting edges per insert and high-pitch cutter bodies. CoroMill 328 is available in versatile grade GC1025, for all material types.

CoroMill 328 has Weldon, arbor and bore with keyway mounting.

Multi-point threading

CoroMill® Plura

Threads are milled in one pass and this multi-point tool gives a true full-profile thread form, with 60° metric, UNC/UNF and NPT/NPTF options available.

Designed for smaller thread sizes and in two optimized grade choices, with or without through coolant. Ideal tool for mass production.

Main considerations

To achieve the best results in a thread milling operation, always consider the following points.

Choice of cutting diameter:

- A smaller cutting diameter will help to achieve higher quality threads.

Tool path is important:

- Tool path will give right or left hand threads, using down- or upmilling.
- Always engage and retract the thread mill in a smooth path, i.e. roll in and out of cut.

Be aware of feed per tooth:

- Always work with small feed per tooth values (very small hex) to achieve best quality.

Always calculate the correct feed required by the machine software:

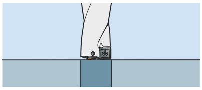
- To ensure the correct insert load.

Several infeed passes may be needed:

- In difficult applications, it may be necessary to separate the operation into several infeed passes to achieve higher quality threads.

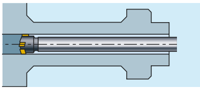
Note. Detailed information about thread milling is provided in the Threading application guide, C-2920:031. Contact your local Sandvik Coromant representative for more information.

Recommended start cutting data



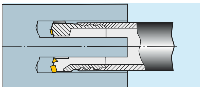
Short hole drilling

Drill dia. D_c mm	Tool choice	Cutting speed v_c m/min	Feed f_n mm/r
3–16	**CoroDrill® Delta-C R846** Grade GC1220	20-30	0.06–0.12
12–63.5	**CoroDrill® 880** Peripheral insert= -LM H13A Central insert = -LM 1044	20–30	0.04–0.10



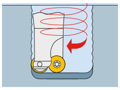
Deep hole drilling

Drill dia. D_c mm	Tool choice	Cutting speed v_c m/min	Feed f_n mm/r
25–43 43–65	**CoroDrill® 800** Central insert – GC1025 Intermediate insert – GC1025 Peripheral insert – GC1025 Wear support pad – PM1	15–25	0.09–0.25 0.20–0.30
63.5–184	**T-Max 424.10** Central insert, -23 GC1025 Intermediate insert, -23 GC1025/H13A Peripheral insert, -23 H13A Wear support pad – S2	20–30	0.15–0.30



Trepanning

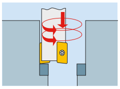
Drill dia. D_c mm	Tool choice	Cutting speed v_c m/min	Feed f_n mm/r
60–110 (larger on request)	**T-Max U 416.7** WCMX 06T308R-53 1020	25	0.10



Circular ramping from solid

Cutter concept	Cutter diameter	Min. hole dia. $D_{m\ min}$ mm	Insert choice	Max. depth of cut/pitch a_p mm	Cutting speed v_c m/min	Feed* f_z mm/tooth
CoroMill® 390 end mill	16 20 25 32 40	26 34 44 58 74	R390-11T331E-ML 2040	2	30	0.10
CoroMill® 300	25 32 34 35 40 42 50 52 63 66 80	38 52 56 58 68 72 88 92 114 120 148	R300-1240E-MM 2040	2	30	0.20

*Reduce feed when programming tool centre feed.



Circular milling from predrilled hole

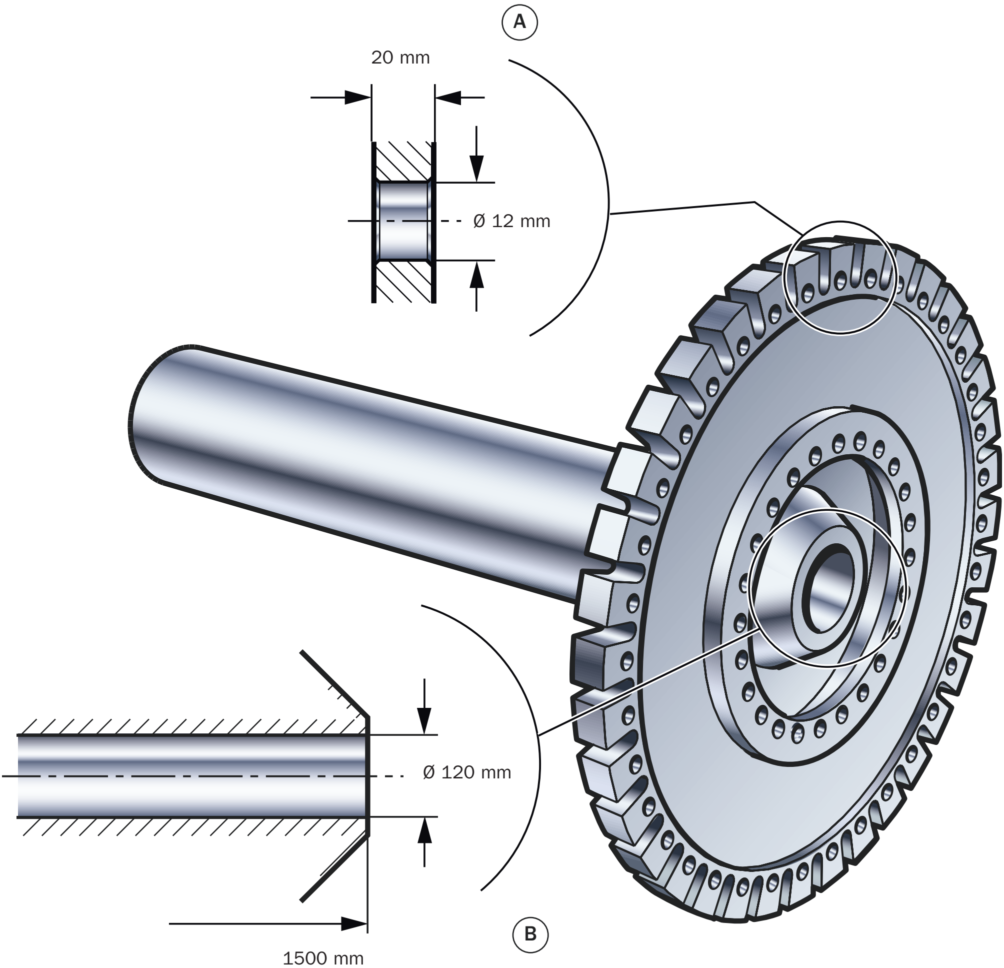
Cutter dia. D_c mm	Min. hole dia. D_c = 80% D_m D_m mm	Tool choice	Max. depth of cut/pitch a_p mm	Cutting speed v_c m/min	Optimal radial cut a_e mm	Feed* f_z mm/ tooth	Feed reduction factor when using tool centre feed: cutter dia. relative to hole D_c/D_m			
							80%	60%	40%	20%
4 5 6 8 10 12	5 6.25 7.5 10 12.5 15	**CoroMill® Plura** R216.23-04050CAK11P 1620 R216.23-05050CAK13P 1620 R216.24-06050CAK13P 1620 R216.24-08050EAK19P 1620 R216.24-10050EAK22P 1620 R216.24-12050GAK26P 1620	4 5 6 8 10 12	75	0.3 0.4 0.5 0.6 0.8 1.0	0.08	7.6	3.4	2	1.4
16 20 25 32 40	20 25 31 40 50	**CoroMill® 390** R390-11T308M-PM 1030	10	30	2.0 2.5 3.1 4.0 5.0	0.1				

*Reduce feed when programming tool centre feed.

Shaft

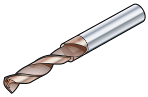
Inconel 718

S 20.22

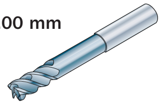


OPERATION A
Ø 12 mm, hole 20 mm deep

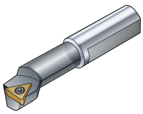
1. Drilling
CoroDrill Delta-C R846 – Ø 11.00 mm
R846-1100-30-A1A 1220
v_c 25 m/min
f_n 0.1 mm/r



2. Finish boring
Alternative 1
CoroMill Plura R216.24 – Ø 12.00 mm
R216.24-12050CAK26P 1620
v_c 40 m/min
f_n 0.2 mm/r



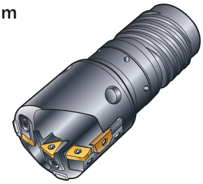
2. Finish boring
Alternative 2
Boring bar R429
Insert TCEX 06T1 02L-F 1105
v_c 35 m/min
f_n 0.08 mm/r



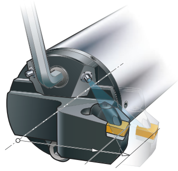
3. Chamfering
Insert 327R06-1212045-CH 1025
Body 327-12B30EC-06
v_c 75 m/min
f_z 0.1 mm/tooth

OPERATION B
Ø 120 mm, hole 1500 mm deep

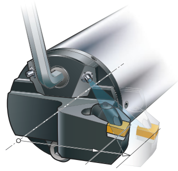
1. Drilling
T-Max 424.10 DHD, Ø 110 mm
Insert grade B2D1
v_c 43 m/min
f_n 0.23 mm/r



2. Rough boring – hole Ø 119.5 mm
CoroTurn SL – Silent Tools boring bar, Ø 80 mm
DNMG 150608-23 1105
v_c 40 m/min
f_n 0.2 mm/r
a_p 2.0 mm



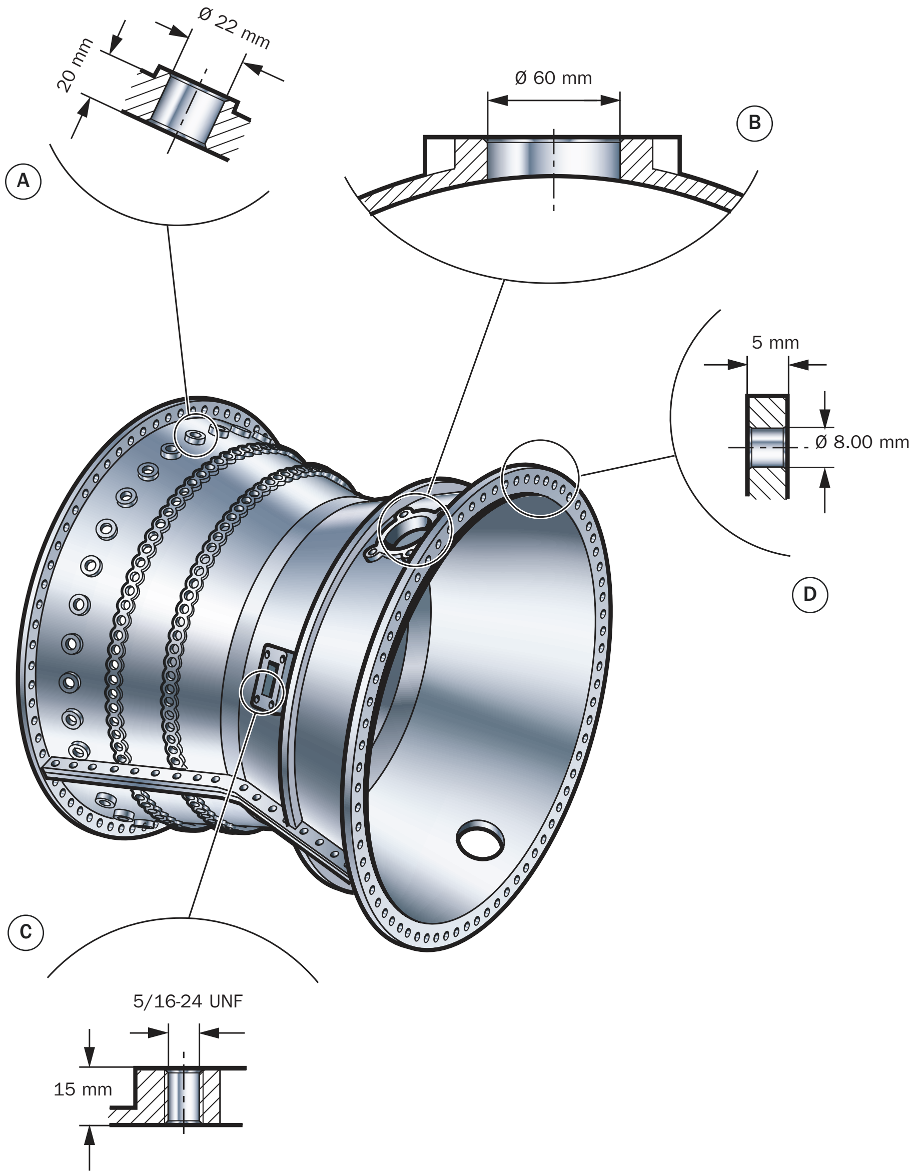
3. Finish boring – hole Ø 120.0 mm
CoroTurn SL – Silent Tools boring bar, Ø 80 mm
DNMG 150608-SM 1105
v_c 40 m/min
f_n 0.2 mm/r
a_p 0.25 mm



Casing

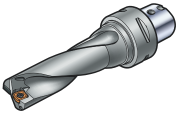
Waspalloy

S CMC 20.22

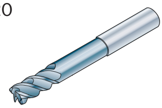


OPERATION A
Ø 22 mm, hole 20 mm deep

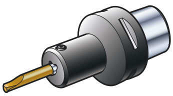
1 Drilling
CoroDrill 880 – Ø 21.00 mm
Central insert – LM 1044
Peripheral insert – LM 1044
v_c 30 m/min
f_n 0.05 mm/r



2. Circular milling – hole Ø 22 mm
CoroMill Plura R216.24 – Ø 12.00 mm
R216.24-12050CAK26P 1620
v_c 75 m/min
a_e 0.5 mm
f_z 0.1 mm/tooth
a_p 10 mm

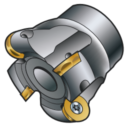


3. Chamfering
Coromant Capto
CoroTurn XS
C4-CXS-47-06
CXS-06T045-20-6225R 1025
v_c 75 m/min
f_z 0.1 mm/tooth

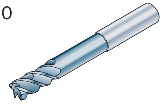


OPERATION B
Ø 60 mm, hole 20 mm deep

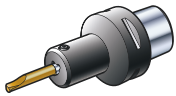
1. Circular ramping – hole Ø 59 mm
CoroMill 300 – Ø 35 mm
R300-12400E-MM 2040
v_c 25 m/min
f_z 0.2 mm/tooth
a_p 2.0 mm



2. Circular milling – hole Ø 60 mm
CoroMill Plura R216.24 – Ø 12.00 mm
R216.24-12050CAK26P 1620
v_c 75 m/min
f_z 0.1 mm/tooth
a_p 10 mm

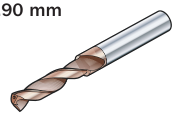


3. Chamfering
Coromant Capto
CoroTurn XS
C4-CXS-47-06
CXS-06T045-20-6225R 1025
v_c 75 m/min
f_z 0.1 mm/tooth

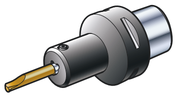


OPERATION C
5/16-24 UNF, hole 15 mm deep

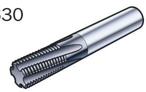
1. Drilling
CoroDrill Delta-C R846 – Ø 6.90 mm
R846-0690-30-A1A 1220
v_c 25 m/min
f_n 0.1 mm/r



2. Chamfering
Coromant Capto
CoroTurn XS
C4-CXS-47-05
CXS-05T045-20-5220R 1025
v_c 75 m/min
f_z 0.1 mm/tooth

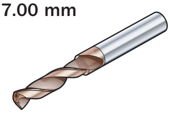


3. Thread milling
CoroMill Plura
R217.33C060240AC13N 1630
v_c 75 m/min
f_z 0.07 mm/tooth

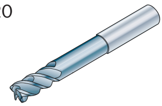


OPERATION D
Ø 8 mm, hole 5 mm deep

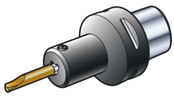
1. Drilling
CoroDrill Delta-C R846 – Ø 7.00 mm
R846-0700-30-A1A 1220
v_c 25 m/min
f_n 0.1 mm/r



2. Finish boring
CoroMill Plura R216.24 – Ø 8.00 mm
R216.24-12050CAK26P 1620
v_c 40 m/min
f_n 0.2 mm/r



3. Chamfering
Coromant Capto
CoroTurn XS
C4-CXS-47-05
CXS-05T045-20-5220R 1025
v_c 75 m/min
f_z 0.1 mm/tooth



Technical data

Turning

Terminology and units

D_m	= Machined diameter	mm
v_c	= Cutting speed	m/min
n	= Spindle speed	r/min
T_c	= Machining time	min
Q	= Metal removal rate	cm^3/min
l_m	= Machined length	mm
P_c	= Net power	kW
$k_{c\,0,4}$	= Specific cutting force for chip thickness 0.4 mm	N/mm^2
f_n	= Feed per revolution	mm/r
κ_r	= Entering angle	degree
R_{max}	= Profile depth	µm
r_ε	= Insert nose radius	mm
a_p	= Depth of cut	mm
h_{ex}	= Max chip thickness	mm
SCL	= Spiral Cutting Length	m

Formulas

Cutting speed (m/min)

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

Spindle speed (r/min)

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

Metal removal rate (cm^3/min)

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

Power requirement (kW)

$$P_c = \frac{v_c \times a_p \times f_n \times k_{c\,0.4}}{60 \times 10^3} \left[\frac{0.4}{f_n \times \sin \kappa_r}\right]^{0.29}$$

Machining time (min)

Constant n

$$T_c = \frac{l_m}{f_n \times n}$$

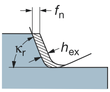
Profile depth (µm)

$$R_{max} = \frac{f_n^2 \times 125}{r_\varepsilon}$$

Max chip thickness

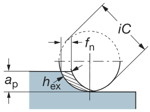
Insert shapes: C, D, S, T, V, W

$$h_{ex} = f_n \times \sin \kappa_r$$



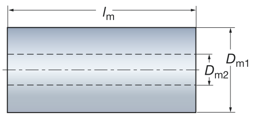
Round inserts

$$h_{ex} = f_n \times \sqrt{\frac{4a_p}{iC} - \left(\frac{2a_p}{iC}\right)^2}$$



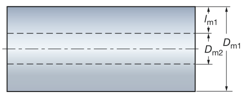
SCL (Spiral Cutting Length) – input info in mm – result in m

External or internal turning



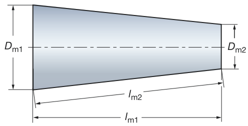
$$SCL = \frac{D_m \times \pi}{1000} \times \frac{l_m}{f_n}$$

Facing



$$SCL = \left(\frac{D_{m1}+D_{m2}}{2} \times \frac{\pi}{1000}\right) \times \frac{l_{m1}}{f_n}$$

Taper cutting



$$SCL = \left(\frac{D_{m1}+D_{m2}}{2} \times \frac{\pi}{1000}\right) \times \frac{l_{m2}}{f_n}$$

How to calculate: l_{m2}

$$l_{m2} = \sqrt{(l_{m1})^2 + \left(\frac{D_{m1} - D_{m2}}{2}\right)^2}$$

Machining time (min)

$$T_c = \frac{SCL}{v_c}$$

Milling

Terminology and units

D_{cap}	= Cutting diameter at actual depth of cut, a_p	mm	
l_m	= Machined length	mm	
a_p	= Cutting depth	mm	
a_e	= Working engagement	mm	
v_c	= Cutting speed	m/min	
Q	= Metal removal rate	cm^3/min	
T_c	= Machining time	min	
z_n	= Total number of teeth in cutter	piece	
f_z	= Feed per tooth	mm/r	
f_n	= Feed per revolution	mm	
v_f	= Table feed (feed speed)	mm/min	
h_{ex}	= Max chip thickness	mm	
h_m	= Average chip thickness	mm	
z_c	= Effective number of teeth	piece	
k_{c1}	= Specific cutting force (for h_{ex} = 1 mm)	N/mm^2	
n	= Spindle speed	r/min	
P_c	= Cutting power net	kW	
κ_r	= Entering angle	degrees	
m_c	= Rise in specific cutting force (k_c) as a function of chip thickness		
iC	= Inscribed circle		
γ_0	= Chip rake angle		

Formulas

Cutting speed (m/min)

$$v_c = \frac{D_{cap} \times \pi \times n}{1000}$$

Spindle speed (r/min)

$$n = \frac{v_c \times 1000}{\pi \times D_{cap}}$$

Table feed (feed speed) (mm/min)

$$v_f = f_z \times n \times z_n$$

Feed per tooth (mm)

$$f_z = \frac{v_f}{n \times z_c}$$

Feed per revolution (mm/r)

$$f_n = \frac{v_f}{n}$$

Metal removal rate (cm^3)

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

Specific cutting force (N/mm^2)

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

Average chip thickness (mm) (Side and facemilling) when $a_e/D_c \leq 0.1$

$$h_m \approx f_z \sqrt{\frac{a_e}{D_{cap}}}$$

Average chip thickness (mm) when $a_e/D_c \geq 0.1$

$$h_m = \frac{\sin \kappa_r \times 180 \times a_e \times f_z}{\pi \times D_{cap} \times \arcsin\left(\frac{a_e}{D_{cap}}\right)}$$

Machining time (min)

$$T_c = \frac{l_m}{v_f}$$

Net power (kW)

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

HRSA – Material cross-reference list

Material condition	Commercial designation	Hardness Brinell HB Ann.	Hardness Brinell HB Aged	Code	Ni	Cr	Co	Fe	Mo	C	Mn
	Haynes 75	-	-	MC S2.0.Z.AN CMC 20.2	76.0[1)]	20.0	-	5.0	-	0.11	1.0[3)]
	Haynes 263	-	-		52.0[1)]	20.0	20.0	0.7[2)]	6.0	0.06	0.6[3)]
	Haynes 625	-	-		62.0[1)]	21.0	1.0[2)]	5.0[2)]	9.0	0.1	0.5[3)]
	Haynes X-750	-	-		70.0[1)]	16.0	1.0[2)]	8.0	-	0.08	0.35[2)]
	Haynes 718	-	-		52.0[1)]	18.0	1.0[3)]	19.0	3.0	0.05	0.35[2)]
	Incoloy 864	-	-		30.0-38.0	20.0-25.0	-	Balance	4.0-4.8	0.08[2)]	1.0[3)]
	Nimocast PE10	-	-	MC S2.0.Z.AN CMC 20.2	56.4	20.0	-	-	6.0	-	-
	Nimocast PD16	-	-		43.8	16.5	-	34.0	-	0.06	-
	Nimocast PK24	-	-		61.1	9.5	15.0	-	9.0	0.17	-
Ni-based	Nimocast 842	-	-		57.7	22.0	10.0	-	-	0.3	-
	Nimocast 713	-	-		72.6	13.4	-	-	-	0.12	-
	Refractaloy 26	-	-		36.1	17.21	18.51	Balance	2.62	0.03	0.55
	Rene 63	-	-	MC S2.0.Z.AG CMC 20.2	Balance	14.0	15.0	3.5	6.0	0.05	0.1
	Rene 77	-	-		Balance	15.0	15.0	0.4	4.2	0.07	0.1
	Rene 80	-	-		Balance	14.0	9.5	-	4.0	0.17	-
	Rene 95	500	300		64.5	14.0	8.0	-	3.5	0.15	-
	Rene 100	-	-		Balance	10.0	15.0	-	3.0	0.18	-
	Rene 125	-	-		Balance	8.9	10.0	-	2.0	0.1	-
	TRW 1800	-	-		70.0	13.0	-	-	-	0.1	-
	TRW VIA	-	-		Balance	6.0	7.5	-	2.0	0.13	-
	Hastelloy B*	140	-		67.0[2)]	1.0[3)]	2.5	5.0	28.0	0.05[2)]	1.0[2)]
	Hastelloy S*	200	-	MC S2.0.Z.AN CMC 20.21	67.0[1)]	16.0	2.0[2)]	3.0[2)]	15.0	0.02[2)]	0.5
	Hastelloy W*	-	-		63.0[1)]	5.0	2.5	6.0	24.0	0.12[2)]	1.0[2)]
	Hastelloy X*	160	-		47.0[1)]	22.0	1.5	18.0	9.0	0.1	1.0[2)]
	Haynes HR-120				37.0	25.0	3.0[2)]	33[1)]	2.5[2)]	0.05	0.7
	Haynes HR-160	-	-		37.0[1)]	28.0	29.0	2.0[2)]	1.0[2)]	0.05	0.5
	Haynes 214*	-	-		75.0[1)]	16.0	-	3.0	-	0.05	0.5[2)]
	Haynes 230	-	-		57.0[1)]	22.0	5.0[2)]	3.0[2)]	2.0	0.1	0.5
	Haynes 242*				65.0[1)]	8.0	2.5[2)]	2.0[2)]	25.0	0.03[2)]	0.8[2)]
	Incoloy 825*	180	-	MC S2.0.Z.AN CMC 20.21	38.0-46.0	19.5-23.5	-	min 22.0	2.5-3.5	0.05[2)]	1.0[2)]
	Incoloy 890	-	-		42.5	25.0	-	Balance	1.5	0.1	1.0[2)]
Annealed or solution treated	Incoloy 909	-	-		35.0-40.0	-	12-16.0	Balance	-	0.06[2)]	-
	Incoloy 330	-	-		34.0-37.0	17-20.0	-	Balance	-	0.08[2)]	2.0[2)]
	Inconel 600*	170	-		72.0	14.0-17.0	-	6-10	-	0.15[2)]	1.0[2)]
	Inconel 601*	150	-		58.0-63.0	21.0-25.0	-	3)	-	0.1[2)]	1.0[2)]
	Inconel 603 XL	-	-		Balance	15.0-23.0	-	-	4.0[2)]	0.3[2)]	0.3[2)]
	Inconel 617*	-	-		min 44.5	20.0-24.0	10-15	3.0[2)]	8-10	0.05-1.5	1.0[2)]
	Inconel 625*	180	-	MC S2.0.Z.AN CMC 20.21	min 58.0	20.0-23.0	1.0[2)]	5.0[3)]	8-10	0.1[2)]	0.5[2)]
	Inconel 690	-	-		min 58.0	27.0-31.0	-	7.0-11.0	-	0.05[2)]	0.5[2)]
	Inconel 693	-	-		3)	27.0-31.0	-	2.5-6.0	-	0.15[2)]	1.0[2)]
	Nimonic 75*	170	-		Balance	18.0-21.0	-	5.0[2)]	-	0.08-0.15	1.0[2)]
	Udimet 520	-	-		Balance	18.0-20.0	11.014.0	-	5.5-7.0	0.02-0.06	-
	Udimet 720	-	-		Balance	15.5-16.5	14.0-15.5	-	2.75-3.25	0.01-0.02	-
	Udimet D-979	-	-		42.0-48.0	14.0-16.0	-	Balance	3.0-4.5	0.08[2)]	0.75[2)]
	Udimet R-41	-	-		Balance	18.0-20.0	10.0-12.0	5.0[2)]	9.0-10.5	0.12[2)]	-
	Nimonic 86	-	-	MC S2.0.Z.AG CMC 20.22	Balance	25.0	-	-	10.0	0 05	0.015
	Astroloy*	-	370		Balance	15.0	17.0	-	5.0	0.04	-
	Hastelloy R-235*		310		61.0	15.0	2.5	10.0	5.5	0.15	0.25
	Haynes R-41	-	-		52.0[1)]	19.0	11.0	5.0	10.0	0.09	0.1[2)]
	Incoloy 901*	180	300		44.3	12.5	-	34.0	6.0	0.05	0.24
	Incoloy 903*	-	380		36.0-40.0	-	13.0-17.0	Balance	-	-	-
	Incoloy 907	-	-		35.0-40.0	-	12.0-16.0	Balance	-	-	-
Aged or solution treated	Incoloy 908	-	-		47.0-51.0	3.75-4.5	0.5[2)]	Balance	-	0.03[2)]	1.0[2)]
	Inconel 706*	-	-	MC S2.0.Z.AG CMC 20.22	39.0-44.0	14.5-17.5	1.0[2)]	Balance	-	0.06[2)]	0.35[2)]
	Inconel 718*	180	380		50.0-55.0	17.0-21.0	1.0[2)]	Balance	2.8-3.3	0.08[2)]	0.35[2)]
	Inconel 722*	-	380		Balance	15.0	-	6.5	-	0.04	0.55
	Inconel X-750*	-	390		min 70.0	14.0-17.0	1.0[2)]	5.0-9.0	-	0.06[2)]	1.0[2)]
	Inconel 751**	-	-		min 70.0	14.0-17.0	-	5.0-9.0	-	0.1[2)]	0.5[2)]
	Inconel 783*	-	-		26.0-30.0	2.5-3.5	3)	24-27.0	-	0.03[2)]	0.5[2)]
Continued...	Inconel HX	-	-		Balance	20.5-23.0	0.5-2.5	17.0-20	8.0-10	0.05-1.15	1.0[2)]

1) Balance
2) Maximum
3) Remainder
* These alloys can be hardened by an aging process.

Si	Al	Ti	Others	USA SAE	USA AMS	UK BS	France ANFOR	Germany Werkst. Nr	Germany DIN 1706	Others
1.0[2)]	-	0.4	0.5[2)]	-	-	-	-	-	-	-
0.4[2)]	0.6[2)]	2.4[2)]	0.2[2)]	-	5886	-	-	-	-	N07263
0.5[2)]	0.4[2)]	0.4[2)]	3.7	-	5666	-	-	-	-	N06625
0.35[2)]	0.8	2.5	1.5	-	5542	-	NC15TNbA	-	-	N07750
0.35[2)]	0.5	0.9	5.109	-	5662/5664		-	-	-	N07718
0.6-1.0	-	0.4-1.0	-	-	-	-	-	-	-	S35135
-	-	-	9.0	-	-	HC202	NC20N13	-	-	-
-	1.2	1.2	-	-	5397	HC204	NK15CAT	LW2.4674	NiFe33Cr17Mo	-
-	5.5	4.7	1.0	-	-	3146	-	-	S-NiCr13Al16MoNb	-
-	-	-	-	5391A	-	HC203	NC13AD	2.4670	S-NiCr13Al16MoNb	SS071712
-	6.2	1.0	2.3	5931A	-	HC203	NC13AD	-	G-NiCr13A16MoNb	-
0.23	0.21	2.66	0.019	-	-	-	Z6NKCDT38	-	-	-
0.2	3.8	2.5	3.5	-	-	-	-	-	-	-
0.1	4.3	3.3	-	-	-	-	-	-	-	-
-	3.0	5.0	-	-	-	-	-	-	-	-
-	-	2.5	3.5	-	-	-	NC14K8	-	-	-
-	5.5	4.7	-	-	-	-	-	-	NiCo15Cr10MoAlTi	-
-	4.7	2.5	4.05	-	-	-	-	-	-	-
-	6.0	0.06	10.5	-	-	-	-	-	-	-
-	5.4	1.0	6.3	-	-	-	-	-	NiTa9Co8W6CrAl	-
1.0[2)]	-	-	0.35[2)]	5396A	5396	-	ND37FeV	2.4800	S-NiMo30	N10001
0.4	0.25	-	1.035[2)]	-	-	-	-	-	-	-
1.0[2)]	-	-	0.6*	-	5755	-	-	-	-	N10004
1.0	-	-	0.6080[2)]	5390A	5390	-	NC22FeD	2.4603	-	N06002
0.6	0.1	-	3.404	-	-	-	-	-	-	N08120
2.75	-	0.5	2.0[2)]	-	-	-	-	-	-	N12160
0.2[2)]	4.5	-	0.12[2)]	-	-	-	-	-	DIN 177444 No 2.4646	N07214
0.4	0.3	-	14.035	-	5891	-	-	-	DIN 177444 No 2.4733	N06230
0.8[2)]	0.5[2)]	-	-	-	-	-	-	-	-	-
0.5[2)]	0.2[2)]	0.6-1.2	3.0	-	-	30072-76	NC21FeDU	2.4858	NiCr21Mo	N08825
1.8	0.1	1.0[2)]	1.0	-	-	-	-	-	-	N08890
0.25-0.5	0.15[2)]	1.3-1.8	5.0	-	-	-	-	-	-	N19909
0.75-1.5	-	-	0,06*	-	-	-	-	1.4886	-	N08330
0.5[2)]	-	-	0.515[2)]	5540	5580	3072-76	NC15Fe	2.4816	NiCr15Fe	N06600
0.5[2)]	1-1.7	-	1.015[2)]	-	5715	-	-	2.4851	NiCr23Fe	N06601
2.0[2)]	0.5[2)]	0.5[2)]	0.1	-	-	-	-	-	-	-
1.0[2)]	0.8-1.5	0.6[2)]	1.0[2)]	-	-	-	-	2.4663a	-	N06617
0.5[2)]	0.4[2)]	0.4[2)]	4.0	-	5666	-	NC22FeDNB	2.4856	NiCr22Mo9Nd	N06625
0.5[2)]	-	-	0.515[2)]	-	-	-	-	2.4642	-	N06690
0.5[2)]	2.4-4	1.0[2)]	1.0	-	-	-	-	-	-	N06693
1.0[2)]	-	0.2-0.6	0.5[2)]	-	-	HR5, 203,-4	NC20T	2.4630	NiCr20Ti	N06075
-	1.8-2.3	2.9-3.25	1.0	-	-	-	-	-	-	-
-	2.25-2.75	4.75-5.25	1.5	-	-	-	-	-	-	-
0.75[2)]	0.75-1.3	2.7-3.3	4.0	-	-	-	-	-	-	N09979
-	1.4-1.8	3.0-3.3	0.01[2)]							N07041
-	-	-	0.03	-	-	-	-	-	-	-
-	4.0	3.5	0.025	-	-	-	-	-	-	N13017
0.6	3.0	2.0	-							AISI 686
0.5*	1.5	3.1	0.006	-	-	-	-	-	-	-
0.12	0.15	2.7	0.15	-	5660	-	ZSNCDT42	LW2.4662	NiFe33Cr14MoTi	N09901
-	0.3-1.15	1.0-1.85	3.0	-	-	-	-	-	-	N19903
0.07-0.35	0.2[2)]	1.3-1.8	5.0	-	-	-	-	-	-	N19907
0.5[2)]	0.75-1.25	1.2-1.8	3.5	-	-	-	-	-	-	N09908
0.35[2)]	0.4[2)]	1.5-2	3.4	-	-	-	-	-	-	N09706
0.35[2)]	0.2-0.8	0.65-1.15	5.3	5383	5589	HR8	NC19FeNB	LW 2.4668	NiCr19Fe19NbMo	N07718
0.2	0.6	2.4	30.0	-	5541	-	NC16FeTi	-	NiCr16FeTi	N07722
0.5[2)]	0.4-1.0	2.25-2.75	1.5	5542G	5582	-	NC16FeTNb	2.4669	NiCr16FeTi	N07750
0.9-1.5	2-2.6	1.0	-	-	-	-	-	-	-	N07751
0.5[2)]	5.0-6.0	0.1-0.4	3.5	-	-	-	-	-	-	R30783
1.0[2)]	-	-	1.0	-	-	-	-	2.4665	-	N06002

Material condition	Commercial designation	Hardness Brinell HB Ann.	Hardness Brinell HB Aged	Code	Nominal composition approximate content in % Ni	Cr	Co	Fe	Mo	C	Mn
Aged or solution treated	Jethete M-252*	-	320	MC S2.0.Z.AG CMC 20.22	55.3	19.0	10.0	2.5	-	0.15	-
	MAR-M 246*	-	270		59.5	9.0	10.0	-	2.5	0.15	-
	MAR-M 421*	-	-		62.3	15.5	10.0	-	1.7	0.15	-
	MAR-M 432*	-	-		52.3	15.5	20.0	-	-	0.15	-
	Nimocast 80*	-	-		69.9	20.0	2.0	5.0	-	0.1	-
	Nimocast 90*	-	-		52.9	20.0	16.5	5.0	-	0.1	-
	Nimonic 80A*	-	350		Balance	18.0-21.0	2.0[1)]	3.0[1)]	-	0.1	1.0[1)]
	Nimonic 81*	200	-		Balance	30.0	2.0	1.0	0.3	0.05	0.5
	Nimonic 90*	-	346		Balance	18.0-21.0	15.0-21.0	1.5[1)]	-	0.13[1)]	1.0[1)]
	Nimonic 91	-	-			27.0-30.0	19.0-21.0	1.0[1)]	-	0.1[1)]	1.0[1)]
	Nimonic 105*	-	320		Balance	14.0-15.7	18.0-22.0	1.0[1)]	4.5-5.5	0.12[1)]	1.0[1)]
	Nimonic 115*	-	350		Balance	14.0-16.0	13.0-15.5	1.0[1)]	3.0-5.0	0.12-0.2	1.0[1)]
	Nimonic 901*	-	350	MC S2.0.Z.AG CMC 20.22	42.5	12.5	1.0[1)]	Balance	5.75	0.1[1)]	0.5[1)]
	Nimonic 263*	-	275		Balance	19.0-21.0	19.0-21.0	0.7[1)]	5.6-6.1	0.04-0.08	0.6[1)]
	Nimonic PE16*	-	250		42.0-43.0	15.5-17.5	2.0[1)]	Balance	2.8-3.8	0.04-0.06	0.2[1)]
	Nimonic PK33*	-	350		Balance	16.0-20.0	12.0-16.0	1.0[1)]	5.0-9.0	0.07[1)]	0.5[1)]
	Rene 41	-	-		Balance	19.0	11.0	3.0	9.75	0.05	-
	Waspaloy	-	-		Balance	18.0-21.0	12.0-15.0	2.0[1)]	3.5-5.0	0.02-0.1[1)]	1.0[1)]
	Waspaloy*	-	-		58.0b	19.0	13.5	2.0[1)]	4.3	0.08	0.1[1)]
Cast or cast and aged	GMR 235*	-	310	MC S2.0.C.NS CMC 20.24	63.3	15.5	-	10.0	5.2	0.15	0.25
	GMR 235D*	-	-		4.5	15.5	-	4.5	5.0	0.15	-
	IN-100*	-	350		61.6	12.5	18.5	-	3.2	0.07	1.2
	Jessop G39*	130	-		67.5	19.5	-	5.0	3.0	0.5	-
	Jessop G64*	220	-		60.7	11.0	-	2.0	3.0	0.15	-
	Jessop G81*	-	300		79.3	20.0	13.0	-	-	0.05	-
	MAR-M 200*	-	-		69.4	9.0	10.0	-	-	0.15	-
Co-based	Air Resist 13	-	-	MC S3.0.Z.AG CMC 20.3	-	21.0	Balance	-	-	0.45	-
	Air Resist 213	-	-		-	19.0	Balance	-	-	0.18	-
	Altemp S 816	-	-		20.0	20.0	47.6	-	4.0	4.0	-
	FSX 414	-	-		10.0	29.0	Balance	-	-	0.25	-
	HS 25	-	-		10.0	20.0	Balance	-	-	0.10	-
	HS 30	-	-	MC S3.0.Z.AG CMC 20.3	16.0	24.0	51.4	1.0	6.0	-	0.6
	HS31	-	-		10.0	25.0	Balance	1.5	-	0.50	-
	HS36	-	-		10.0	18.0	53.1	2.0	-	15.0	1.5
	Jessop 832	-	-		12.0	19.0	44.0	17.0	2.0	-	0.8
	Jessop 834	-	-		12.0	19.0	42.0	20.0	2.0	-	-
	Jessop 875	-	-		-	21.0	66.0	-	-	11.0	-
	Jetalloy 209	-	-	MC S3.0.Z.AG CMC 20.3	10.0	20.0	52.0	1.0	-	15.0	-
	L-251	-	-		10.0	19.0	Balance	1.0	-	-	-
	M 203	-	-		24.5	19.5	Balance	1.0	-	2.15	0.8
	M 204	-	-		24.5	18.5	Balance	1.6	-	-	-
	M 205	-	-		24.5	18.5	Balance	1.6	-	2.75	-
	MAR-M 302	-	-	MC S3.0.Z.AG CMC 20.3	-	21.5	Balance	-	-	0.85	-
	MAR-M 322	-	-		-	21.5	Balance	0.75	-	-	0.1
	MAR-M 509	-	-		10.0	23.5	Balance	-	-	-	0.1
	MAR-M 905	-	-		20.0	20.0	Balance	-	-	-	-
	MAR-M 918	-	-		20.0	20.0	Balance	-	-	0.05	-
	Refreactaloy 70	-	-		20.0	21.0	46.0	0.5	8.0	4.0	-
	V-36	-	-		20.0	25.0	43.2	2.4	4.0	2.0	0.6
	WI-52	-	-	MC S3.0.Z.AG CMC 20.3	0.5	21.0	62.6	2.0	-	11.0	0.25
	Jessop X-40	-	-		10.0	25.0	Balance	1.5	-	0.50	0.5
	Jessop X-45	-	-		10.5	25.5	Balance	2.0	-	0.25	-
	Jessop X-50	-	-		20.5	25.5	40.3	4.0	-	12.0	-
	Jessop X-63	-	-		10.0	25.0	57.6	1.0	6.0	-	-

Continued...

1) Maximum

* These alloys can be hardened by an aging process.

Si	Al	Ti	Others	USA SAE	USA AMS	UK BS	France ANFOR	Germany Werkst. Nr	Germany DIN 1706	Others
-	1.0	2.5	-	-	5551	-	-	2.4916	G-NiCr19Co	N07252
-	5.5	1.5	1.5	-	-	-	-	2.4675	NiCo10W10Cr9AlTi	-
-	4.25	1.75	1.75	-	-	-	-	-	NiCr16Co10WAlTi	-
-	2.5	4.3	2.0	-	-	-	-	-	NiCo20Cr16WAlTi	-
-	1.2	2.5	-	-	-	3146	NC 20 TA	2.4631	NiCr20TiAl	-
-	1.3	2.4	-	-	-	-	NC 20 K17 TA	2.4632	NiCr20Co18Ti	-
1.0[1)]	1.0-1.8	1.8-2.7	0.17	-	-	Hr 410,601	NC20TA	2.4631	NiCr20TiAk	N07080
0.5	0.9	1.8	0.26	-	-	-	-	-	-	-
1.0[1)]	1.0-2.0	2.0-3.0	0.39[1)]	-	-	Hr 2, 202	Nc20ATV	2.4632	NiCr20Co18Ti	N07090
1.0[1)]	0.9-1.5	1.9-2.7	1.6	-	-	-	-	-	-	-
1.0[1)]	4.5-4.9	0.9-1.5	0.4[2)]	-	-	HR 3	NCKD20ATV	2.4634	NiCo20C15MoAlTi	-
1.0[1)]	4.5-5.5	3.5-4.5	0.4[2)]	-	-	HR4	NCK15ATD	2.4636	NiCo15Cr15MoAlTi	-
0.4[1)]	0.35[1)]	2.9	0.6[2)]	5660C	5661A	-	ZSNCDT42	2.4662	NiCr15MoTi	N09901
0.4[1)]	0.6[2)]	1.9-2.4	0.3[2)]	-	-	HR10	NCK20D	2.4650	NiCr15Co19MoTi	N07263
0.5[1)]	1.1-1.3	1.1-1.3	0.6[2)]	-	-	HR207	NW11AC	-	NiFe33Cr17Mo	-
0.5[1)]	1.7-2.5	1.5-3.0	0.27[1)]	-	-	-	NC19DUV	-	NiCr20Co16MoTi	-
-	1.6	3.5	0.007	-	5399	-	NC19KDT	2.4973	NiCr19Co11MoTi	N07041
0.75[1)]	1.2-1.6	2.75-3.25	0.6	-	-	-	-	2.4654	-	N07001
0.15[1)]	1.5	3.0	0.16[1)]	-	5544	-	NC20K14	LW 2.4668	NiCr19Fe19NbMo	
0.6	3.0	2.0	0.06	-	-	-	-	-	-	AISI: 686
-	3.5	2.5	0.05	-	-	-	-	-	NiCr16MoAl	-
0.5	5.0	4.75	-	-	5397	HC204	NK15CAT	LW 2.4674	NiCo15Cr10MoAlTi	N13100
-	-	-	4.5	-	-	-	-	-	NiCr20MoW	-
-	6.0	-	4.0	-	-	-	-	-	NiCr11AlWNb	-
-	1.3	2.3	-	-	-	-	-	-	NiCr20Co18Ti	-
-	5.0	2.0	1.0	-	-	-	NKW10CATaHf	-	NiW13Co10Cr9AlTi	-
-	3.5	-	0.1	-	-	-	-	-	-	-
-	3.5	-	0.1	-	5537C	-	KC20WN	-	CoCr20W15Ni	-
-	-	-	0.4	-	-5534	-	-	LW 2.4989	CoCr20Ni20W	-
-	-	-	-	-	-	-	-	-	-	-
-	-	-	-	5537C	5759	-	KC20WN	LW 2.4964	CoCr20W15Ni	AISI:670
0.6	-	-	0.4	5380	-	-	-	-	CoCr26Ni14Mo	R30030
-	-	-	0.5	5382	ASTM A567	3146	KC25NW	LW 2.4670	CoCr25NiW	R30031
-	-	-	0.4	-	-	-	-	-	CoCr19W14NiB	-
0.3	-	-	3.5	-	-	-	-	-	CoCr19Fe16NiMoVNb	-
-	-	-	6.5	-	-	-	-	-	CoCr19Fe20NiMoVNb	-
-	-	-	2.45	-	-	-	-	-	CoCr21W11Nb	-
-	-	2.0	0.02	-	5772	-	KC22WN	-	CoCr22W14Ni	-
-	-	0.4	-	-	-	-	-	-	-	-
1.0	24.5	0.07	-	-	-	-	-	-	-	-
1.0	-	0.07	-	-	-	-	-	-	-	-
-	-	0.07	1.67	-	-	-	-	-	-	-
-	-	-	9.0						CoCrW10TaZrB	
0.1	0.75	1.0	6.75						CoCr22W9TaZrNb	
0.1	0.2	0.6	4.0			3146-3			CoCr24Ni10WtaZrB	
-	0.5	0.05	7.6							
-	-	-	-						CoCr20Ni20Ta	
-	-	-	0.08							
0.5	-	-	2.29						CoCr25Ni20MoWNb	
0.25	-	-	2.45	-	-	-	-	-	CoCr21Mo11W	-
0.5	-	-	0.5	-	5382	3156-2	-	LW 2.4670	CoCr25NiW	ASTM: A567
-	-	-	8.6	-	-	-	-	-	-	-
-	-	-	0.75	-	-	-	-	-	-	-
-	-	-	0.45	-	-	-	-	-	-	-

Material condition	Commercial designation	Hardness Brinell HB Ann.	Aged	Code	Nominal composition approximate content in % Ni	Cr	Co	Fe	Mo	C	Mn
Annealed or solution treated	J1650	-	-	MC S3.0.Z.AG CMC 20.3	27.0	20.0	Balance	-	-	0.2	-
	Haynes 25*	-	-		10.0	20.0	51.0[1]	3.0[2]	-	0.1	1.5
	Haynes 188*	-	-		22.0	22.0	39.0[1]	3.0[2]	-	0.1	1.25[2]
	Undimet 188*	-	-		20.0-24.0	20.0-24.0	Balance	3.0[2]	-	0.05-0.15	1.25[2]
	Undimet L-605*	-	-		9.0-11.0	19.0-21.0	Balance	3.0[2]	-	0.05-0.15	1.0-2.0
In aged condition	HS6*	-	-	MC S3.0.Z.AG CMC 20.32	2.5	28.0	60.5	3.0	-	5.0	-
	HS21*	-	-		3.0	27.0	Balance	1.0	5.0	-	0.6
	J1570*	-	350		28.0	20.0	43.0	2.0	-	7.0	-
Fe-based quenched and tempered martensitic > 0.12%C	Greek Ascoloy	-	300	MC SN.N.N.NN CMC 05.3	2.0	12.0	-	Balance	-	0.19	-
	Jethete M 152**	-	300		2.5	12.0	-	Balance	1.7	0.15	-
Fe-based precipi-tation hardened steels	Crucible A286*	-	250	MC M1.0.Z.PH CMC 05.4	25.0	14.0	-	Balance	1.3	0.05	1.3
	Discaloy 24*	-	280		26.0	13.5	-	Balance	2.7	0.04	0.9
	Discaloy 16-25-6*	-	290		25.0	16.0	-	Balance	6.0	0.12	1.35
	Unitemp 212*	-	280		25.0	16.0	-	Balance	-	0.08	0.05
	Incoloy A-286	-	280		25.5	15.0	-	Balance	1.25	-	-
	665B*	-	280		26.0	13.5	-	Balance	1.5	0.08	-
	19-9-DL*	-	250		9.0	18.5	-	Balance	1.4	0.3	1.5
	17-4-PH*	-	250	MC P5.0.Z.PH CMC 05.4	4.0	16.5	-	Balance	-	0.7	1.0
	Udimar 250*	-	-	MC P3.3.Z.AN CMC 05.4	17.0-19.0	-	7.0-8.5	Balance	4.6-5.1	0.03[2]	0.1[2]
	Udimar 300*	-	-		18.0-19.0	-	8.0-9.5	Balance	4.6-5.2	0.03[2]	0.1[2]
Annealed or solution treated	Incoloy 800**	184	-	MC S1.0.U.AN CMC 20.11	30-35.0	19-23.0	-	min 39.5	-	0.1[2]	-
	Incoloy 803	-	-		32-37.0	25-29.0	-	Balance	-	0.06-0.1	1.5[2]
	Incoloy DS**	180	-		34.5-41	17-19.0		Balance	-	0.01	0.8-1.5
	Sanicro 30**	150	-		34.0	22.0	-	Balance	-	0.03	0.55
	Nilo 36	-	-		36.0	0.25	-	Balance	-	0.05	0.50
	Nilo 42	-	-		42.0	0.25	1.0	Balance	-	0.005	0.8
Aged or solution treated and aged	Haynes 556*	-	260	MC S1.0.U.AG CMC 20.12	20.0	22.0	18.0	31.0[1]	3.0	0.1	1.0
	Multimet 155*	-	260		20.0	21.0	20.0	30.0[1]	3.0	0.12	1.5
	N155*	-	260		20.0	21.0	20.0	Balance	3.0	0.15	1.5
	N156*	-	260		33.0	17.0	24.0	Balance	3.0	0.33	-
	S 590*	-	270		20.0	21.0	20.0	Balance	4.0	0.43	-

1) Balance
2) Maximum
* These alloys can be hardened by an aging process.
** These alloys cannot be hardened by an aging process.

Si	Al	Ti	Others	USA SAE	USA AMS	UK BS	France ANFOR	Germany Werkst. Nr	Germany DIN 1706	Others
-	-	3.8	-	-	-	-	-	-	-	-
0.4[2]	-	-	15.0	5537C	5759	-	KC20WN	LW 2.4964	CoCr20W15Ni	R30605
0.35	-	-	14.03	-	5772	-	KC22WN	-	CoCr22W14Ni	R30188
0.2-0.5	-	-	15.0	-	-	-	-	-	-	R30188
0.4[2]	-	-	16.0	-	-	-	-	-	-	R30605
-	-	-	1.0	-	5373	-	-	-	-	R30006
0.6	-	0.25	-	-	5385	3531	-	-	CoCr28Mo	R30021
-	-	-	-	-	-	-	-	-	-	-
-	-	-	3.15	-	5508	-	-	-	-	S41880
-	-	-	0.3	-	5718	-	Z12 CND12	LW 1.4939	-	-
0.5	0.2	2.1	-	J467	5525	HR5152	Z06 NCT25	LW 1.4980	-	ASTM: 368
0.8	0.1	1.7	-	(J467)	5733	-	Z3 NCT25	LW 1.4943	-	S66220
0.7	-	0.3	0.4	-	5725	-	Z3 NCT25	-	-	-
0.15	0.15	4.0	0.5	-	-	-	-	-	-	-
-	-	2.1	-	5525	5726	HR 51-2	Z 3 NCT 25	1.4980	X5NiCrTi2615	S66286
-	-	2.85	-	J467	5543	-	-	1.4943	-	S66545
0.6	-	0.25	1.75	-	5526	-	-	LW 1.4984	-	S63198
1.0	-	-	3.57	J467	5604	-	-	1.4542	XCrNiCuNb174	S17400
0.1[2]	0.05-0.15	0.3-0.5	0.02[2]	-	-	-	-	-	-	K92890
0.1[2]	0.05-0.15	0.55-0.8	0.02[2]	-	-	-	-	-	-	K93120
-	0.15-0.6	0.15-0.6	-	-	-	-	-	1.4876	X10NiCrAlTi3220	N08800
1.0[2]	0.15-0.6	0.15-0.6	-	-	-	-	-	-	-	S35045
1.9-2.6	-	0.2	0.5	-	-	-	-	1.4864	X12NiCrSi3616	-
0.55	0.3	0.5	0.1	-	-	-	-	1.4558	X2NiCrAlTi3220	-
0.25	0.10	0.10	0.14	-	-	-	-	1.3912	-	K93600
0.03	0.15	-	-	-	-	-	A54-301	1.3917		K94100
0.4	0.2	-	3.34	-	5877	-	-	-	-	R30556
1.0[2]	-	-	3.65	-	5769	-	-	1.4971	X12CrCoNi2120	R30155
0.5	-	-	1.0	-	5768	-	-	1.4971	X12CrCoNi2120	R30155
-	-	-	-	-	-	-	-	-	-	-
-	-	-	-	-	5770	-	-	1.4977	X40CoCrNi2020	R30590

Notes

Notes

Notes

Acknowledgement
Some sections of this article present the findings of technical investigations carried out by the AMRC* which are sponsored by Sandvik Coromant.

*The Advanced Manufacturing Research Centre (AMRC) is a partnership which builds on the shared scientific excellence, expertise and technological innovation of industrial partners and world-class research within the University of Sheffield's Faculty of Engineering.

Head office:

AB Sandvik Coromant
SE-811 81 Sandviken, Sweden
www.sandvik.coromant.com
E-mail: info.coromant@sandvik.com

C-2920:034 ENG/01
This folder is printed on recycleable paper. Printed in Sweden at Sandvikens Tryckeri

SANDVIK Coromant

Your success in focus