

PRODUCTIVITY IN DIE AND MOULD MAKING

Application technology 4
 The choice of holding tools 20
 Die and mould materials 25
 Tailor Made and Engineered solutions 30

MILLING

Overview 32
 CoroMill 200 — Round insert cutter, negative 34
 CoroMill 300 — Round insert cutter, positive 41
 CoroMill Ball Nose endmill 46
 CoroMill 390 — endmill and square shoulder cutter 90° 50
 CoroMill 390 — long edge cutter 90° 59
 CoroMill modular cutting tools 68
 Coromill 790 Al endmill 74
 CoroMill Plura — carbide endmills 79
 CoroMill 245 — facemill 45° 108
 CoroMill 290 — square shoulder cutter 90° 114
 Drilling endmill 120
 Spare parts 122
 Cutting data and technical information 126

SHORT HOLE DRILLING

Overview drilling/boring 144
 CoroDrill Delta-C — solid carbide drills 146
 Coromant Delta — brazed carbide drills 152
 Coromant U — indexable insert drills 158
 Cutting data and graphs 169

DEEP HOLE DRILLING

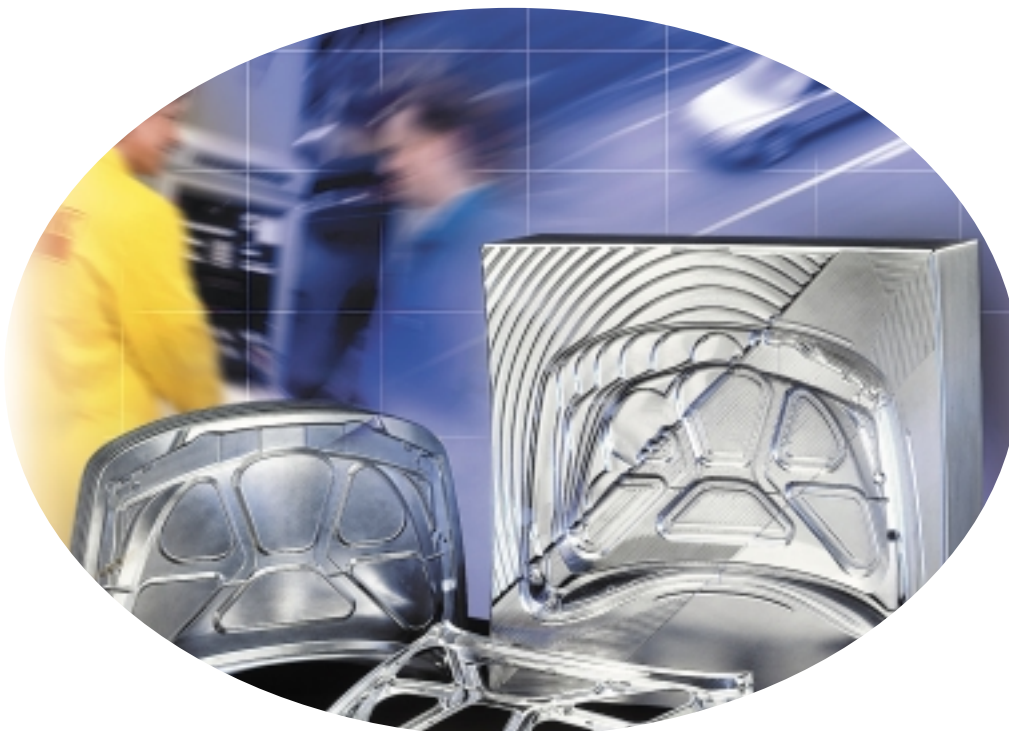
Overview and introduction 178
 Drill heads / STS and Ejector systems 182
 Gun drills 198
 Cutting data and graphs 201

TOOLHOLDING

Basic holders and tool adaptors 210

GENERAL INFORMATION

Practical tips 242
 Material cross reference list 249
 Coromant grades 254



Productivity in die and mould making

Within the die and mould making industry an evolution is taking place. Shorter product life cycles make flexibility and productivity throughout the entire manufacturing process essential to obtain a good total economy. The machine tools are becoming more and more sophisticated in order to perform applications at higher speeds and feeds and accuracy than ever. Pre-hardened and hardened tool steels are more frequently being used, as well as other difficult-to-machine materials.

All these demands contribute to make a careful choice and application of the most capable tooling decisive for a successful production. Many different cutting tools are involved in the manufacture of a wide range of complex tools for pressing, casting, forging, injection and compression moulding — from large deep hole drills to the very smallest ball nose end mills.

The optimum combination of dedicated tooling and advanced application technology is fundamental for obtaining the best manufacturing economy.

This catalogue offers both tools with indexable inserts and solid carbide tools, as well as holding tools, all with the capability required from the die and mould making industry. Cost saving tooling, providing the highest productivity from roughing to finishing operations, are included — as well as tools developed for super-finishing using HSM techniques.

The contents also include basic definitions and formulas for calculation of cutting data as well as brief information about specific application techniques applied within the manufacture of dies and moulds — from the theoretical as well as the practical point of view.

The objective of this catalogue is to provide easy access to more efficient methods and tooling in order to reach a higher level of productivity, and subsequently more competitive die and mould manufacturing.

Production planning

Dedicated tooling operated with modern programming techniques — an opportunity to cut the production cost by half.



Often machining of a die is structured to perform the roughing, rest-milling and semi-finishing operations during the day shift, while attended by an operator. The more time consuming finishing operations are then performed unmanned at night and during weekends.

By careful edge wear analysis, the tool life is established so that tool changes can be automatized and the machine tool even further utilized.

Modern CAD/CAM systems and production equipment can be used in much better ways if old thinking, traditional tooling and production habits are abandoned.

If instead, new ways of thinking and the right approach is applied to applications, high success will be the result.

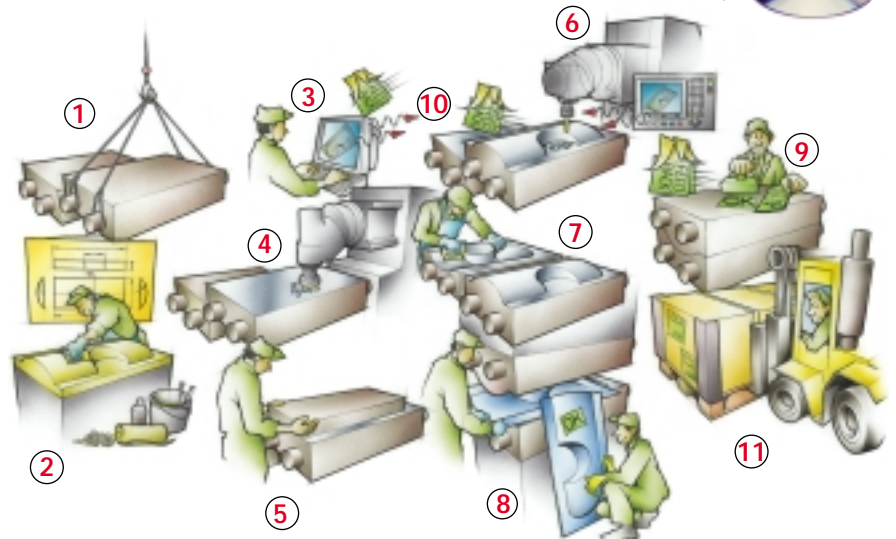
As the cost per hour of an advanced machine tool is often at least 2-3 times that of a work station, the cost of more detailed and optimized programming work is quickly repaid.



Detailed information is found in the Sandvik Coromant application guide Die & Mould Making (C-1120:2), the User's guides CoroMill Plura (C-1129:041), CoroMill Delta-C (C-1229:011) and also our CD Coroguide (C-2948:027-MUL), a guide to the correct cutting data for your application.



1. **Receiving** — standard die parts, steel castings, planning and scheduling
2. **Model shop** — tooling aid and checking fixtures
3. **CAM-room** — schedule, exchange reports DNC/CNC program match, layout
4. **2D-machining** — shoes, pads,
5. **Blocking** — die packs, die design
6. **3D-machining** — sub-assembled dies
7. **Polishing** — standard parts and components
8. **Tryout** — sheet steel material specifications, check fixture.
Inspection — functional build evaluation, stable metal panel product fixture
9. **Die completion** — die design handling devices, production requirements, inspection requirements
10. **Feed back** — die history book, check list base
11. **Shipping**



Simplified, the die construction work flow can be explained in the steps above.

How to apply your tool correctly?

Down milling or up milling?

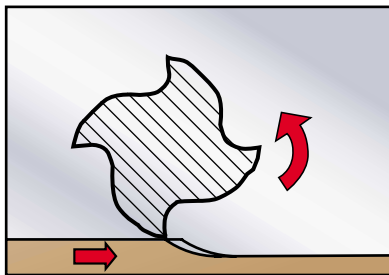
To feed with or against the direction of tool rotation is a decision fundamental for the cutting process. Normally down milling is more favourable than up milling as considerably more heat is generated in up milling. Tool life, therefore, will generally be shorter, often as much as 50%. The radial forces are also considerably higher in up milling, which has a negative affect on the spindle bearings. In down milling the cutting edge is mainly

exposed to compressive stresses, which are much more favourable for the properties of cemented carbide compared with the tensile stresses developed in up milling.

When side milling (finishing) with solid carbide, especially in hardened materials, up milling is first choice. It is then easier to get a better wall straightness tolerance and also a more accurate corner angle. The mismatch between

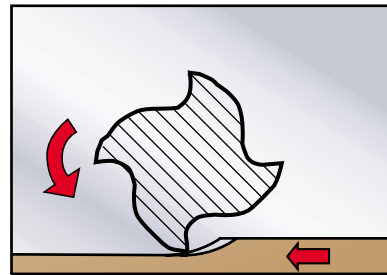
different axial passes will also be less, if none. This is mainly due to the direction of the cutting forces. If using a very sharp cutting edge the cutting forces in up milling tend to pull the cutter towards the material. Up milling can be favourable to older manual milling machines with more play in the lead screw, because a *counter pressure* is created which stabilises machining.

As a general rule – use down milling tool paths.



Down milling

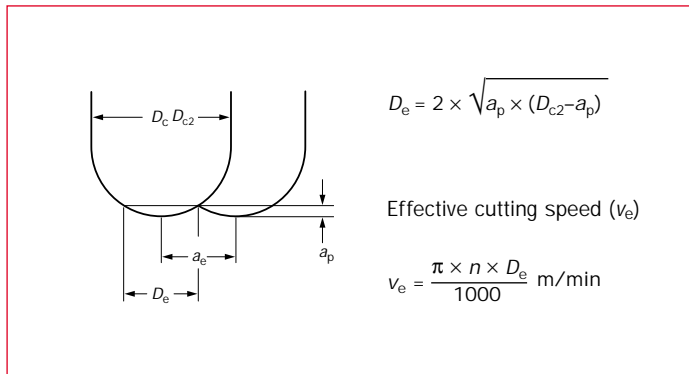
- Less heat generation.
- Compressive stresses favourable for the carbide edge.
- Longer tool life.
- Better surface finish.
- Better geometrical accuracy.
- More friendly to spindle bearings.



Up milling

- Better wall straightness and more accurate corner angle when side milling – finishing.
- Little or no mismatch between subsequent toolpaths.
- Makes somewhat worn manual machine tools perform better.

Effective diameter in cut

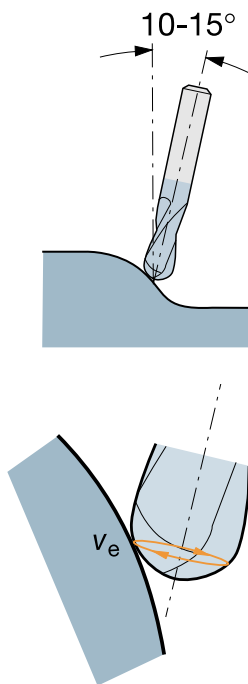


When determining the cutting data for cutting tools with corner radii, ball nose endmills or cutters with round inserts, the effective cutting speed (v_e) should always be calculated, based on the true or effective diameter in cut (D_e) as this varies depending on the actual depth of cut (a_p value).

This variation might be considerable — particularly for cutters with smaller diameters using round inserts or inserts with larger corner radii and ball nose endmills. If this is not taken into account, there will be severe miscalculations of the feed rate as this is dependent on the spindle speed (rpm) and the diameter in cut for a specific cutting speed.

If using the nominal diameter value of the tool, when calculating cutting speed, the effective or true cutting speed will be too low at shallow depths of cut. This is valid for tools such as CoroMill 200 and CoroMill 300 round insert cutters (especially in the smaller diameter range), Coromant ball nose end mills and end mills with big corner radii. The feed rate will of course also be too low and the productivity severely hampered.

Most important is that the cutting conditions for the tools will be way below their capacity and recommended application range. Often this leads to premature chattering and chipping of the cutting edge due to too low cutting speed often in combination with too high feed per tooth.



Tool centre — the most critical cutting area

When using a ball nose endmill the most critical area is at the centre where the cutting speed is zero. This is very unfavourable for the cutting process.

Chip evacuation at the tool centre is also more critical due to the narrow space at the chisel edge. By tilting the spindle or the workpiece 10 to 15 degrees this problem can be avoided, because the cutting zone will be moved away from the tool centre. Sometimes even a shorter or optional type of tool can be used favourable to productivity and the total result.

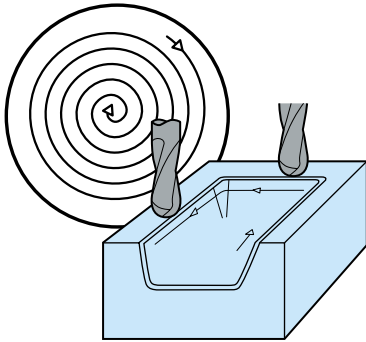
For long tool life in a milling process it is also more favourable to stay in cut continuously and as long as possible. If the "cut" during milling operations is interrupted or intermittent, it exposes the cutting edge to many engagements and retracts in the material. This is of considerable disadvantage for the tool life as the cutting edge will be exposed to thermal stresses and fatigue. It is more favourable to cemented carbide to have an even but high temperature in the cutting zone rather than large fluctuations.

Choice of machining methods

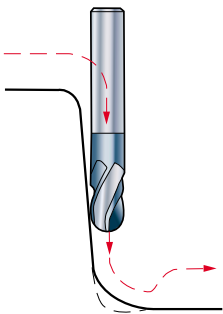
The larger the component and the more complicated, the more important the choice of machining methods, tool paths and the tooling becomes.

The preferred method is **contouring**. This allows the effective cutting speed to be more easily controlled, to utilize higher feed rates and to machine in the preferred down milling mode. The result is increased security, longer tool life, better quality and increased productivity.

The traditional technique — mainly linked to older types of machines and also often the easiest way to programme tool paths for a cavity — is the **conventional copy milling** method. This means many engagements and retracts of the tool in the material.



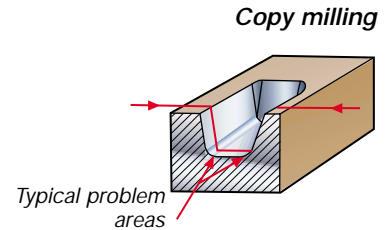
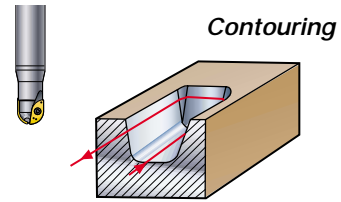
In a milling process it is favourable for a good tool life to stay in cut continuously and as long as possible.



Copy milling along steep walls should be avoided as much as possible.

With modern CAM-systems the more advanced method to "slice off" material can be applied. This technique using a constant Z-value — or true 3-axis machining — and contouring tool paths in combination with down milling, results in:

- a considerably shorter machining time
- better machine and tool utilisation
- improved geometric quality of machined die or mould
- less manual polishing and try-out time



Copy milling often gives errors in the milled surface that affects productivity as well as tool life.

Contouring tool paths

The best way to apply **down milling** in cavity milling is to use contouring tool paths. Contouring with the periphery of the milling cutter often results in higher productivity, as more teeth are effectively in cut on a larger tool diameter.

If the spindle speed is limited in the machine, contouring will help to retain the cutting speed and also

reduce changes in work load and direction. This is of specific importance in HSM applications and hardened materials as the cutting speeds and feeds are high. The cutting edge and process is more vulnerable to variations that can give differences in deflection and create vibrations — and ultimately total tool breakdown.

Copy milling

Copy milling tool paths are often a mix of up and down milling (zig-zag), resulting in extensive in and out of cut. *This is not favourable for any milling cutter and is also harmful to the quality of the die or mould.*

Each engagement means that the tool will deflect and there will be an elevated mark on the surface.

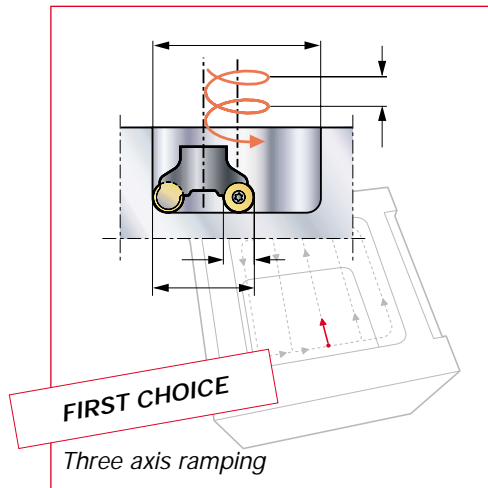
Surface errors appear also at the retract where the cutting forces and the bending of the tool is released,

resulting in a slight undercutting of material.

Particularly suitable for up-copying are the CoroMill Plura endmills of spherical shape. These tools have excellent accessibility and are to some extent also capable of performing undercutting operations.

Methods for opening up a cavity

Two and three axis ramping are efficient ways to approach in cavity milling. Also in many boring operations circular interpolation in helix is much more power efficient and flexible than using a number of boring tools.



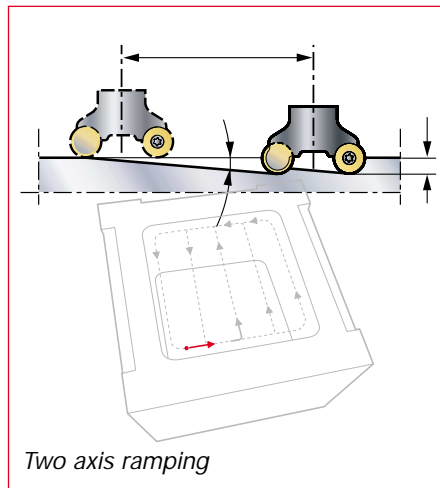
Three axis ramping

Feeding the tool in a helical shaped path in the axial direction of the spindle should be used if possible to reach the axial levels in cavity milling.

This is the most favourable method because it gives a smooth, successive engagement without quick or abrupt changes in the feed direction. Crucial in hard materials and HSM.

The maximum ramping angle for the cutter should always be checked when using circular interpolation in helix.

Pre-drilling of a starting hole is **NOT** recommended as one extra tool is needed. Unproductive time for positioning and tool changing are negative factors, and also tool magazine positions are unnecessarily filled. Chip jamming in the pre-drilled hole(s) very often leads to recutting of chips and tool break down.



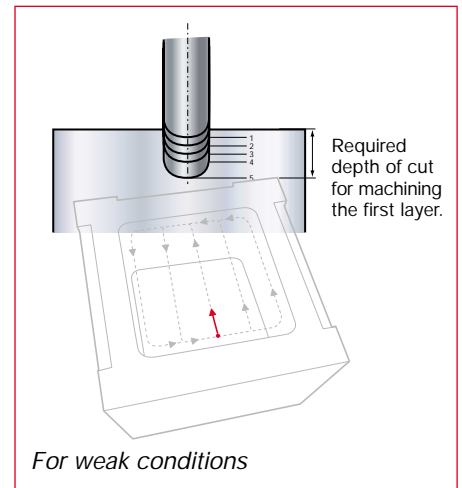
Two axis ramping

Another good method to reach a full axial depth of cut, is linear ramping in the X/Y and Z axis.

Note that if choosing the right starting point, there will be no need for milling away stock from the ramping section. Ramping can start from in to out or from out to in, depending on the geometry of the die or mould. The main criterion is how to get rid of the chips in the best way. *Down milling* should be performed in a continuous cut. When taking a new radial depth of cut it is important to approach with a ramping movement or, preferably using smooth circular interpolation. In HSM applications this is crucial.

The ramping angle is dependent upon the diameter of the cutter used, clearance to the cutter body, insert size and depth of cut. The clearance also depends upon the diameter of the cutter.

Due to the fact that the total cutting forces are increased in ramping operations it is important to reduce the feed rates, especially in 2-axis ramping. The support in the tip seat for the insert is also less due to the direction of the cutting forces.



For weak conditions

Normally the feed speed should be reduced with 30 - 50 % in 2-axis ramping.

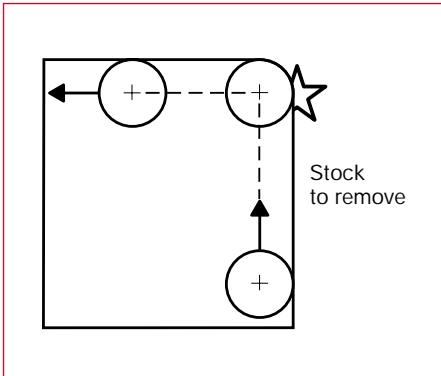
The maximum ramping angle for different sizes of Coromant cutters is given on the ordering pages.

For weak conditions

If using a ball nose end mill it is pretty common to use a peck-drilling cycle to reach a full axial depth of cut and then mill away a layer of the cavity.

This is then repeated until the cavity is finished. One drawback with this approach is that chip evacuation problems arise at the centre of the end mill. A better method is to reach the full axial depth of cut using circular interpolation in helix. It is also important then to facilitate chip evacuation.

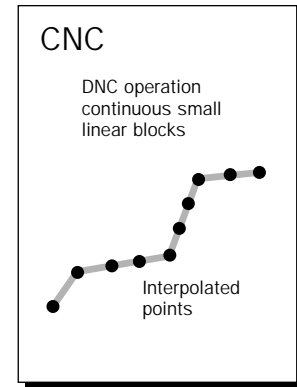
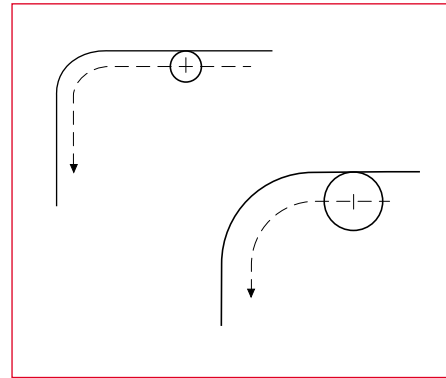
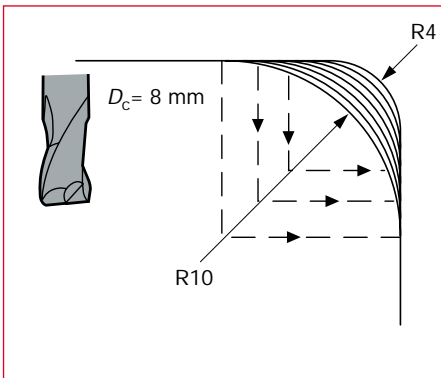
Refined methods for machining of corners



Linear movements

The traditional way of machining a corner is to use linear movements (G1) with non-continuous transitions in the corner. Due to maximum contact between the cutter and the material under the momentaneous stop this will result in vibrations, uneven cutting forces and under-cutting of the corner.

The cutting tool will be exposed to edge frittering and eventually a total break down.



Linear interpolation requires a large volume of interpolated points to define a curve.

Circular interpolation

By using circular interpolation instead, these problems are normally overcome and the corner is produced with smooth continuous transitions and without any vibrations.

The remaining stock in the corner can be machined using circular interpolation — preferably in combination with a cutter radius smaller than the corner radius.

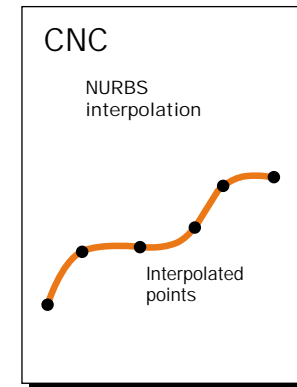
By consequently using a programming technique based on circular interpolation (or NURBS interpolation), it is possible to drive the mechanical functions of a machine tool to much higher levels.

NURBS are built up by three parameters — poles, weights and knots. As NURBS are based on non-linear movements, the tool paths will have continuous transitions and thus make it

possible to keep much higher acceleration, deceleration and interpolation speeds.

A smoother movement of the mechanics also results in better surface finish as well as improved dimensional and geometrical accuracy.

Detailed information about the methods for the machining of corners, NURBS and also machining in segments is given in the Sandvik Coromant application guide Die & Mould Making, (C-1120:2.)



NURBS interpolation requires fewer points to define a curve using control points that note the weight and knot vector of the spline.



Optimize the mechanical functions of the machine tool and gain between 20 – 50 % in productivity.

Advanced methods for high productivity in rough milling

New toolpath strategies appear mainly within HSM today. One method of interest is called **trochoidal milling** where the tool — most often a multi edge solid carbide endmill — is fed in a “spiro-graph-like” toolpath.

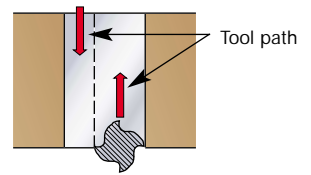
This method is ideal for milling pockets, slots and grooves. The method is well suited for high table feeds (HSM) because the cutting tool always moves in curved patterns, which allows the maximum feed rate to be maintained throughout the entire machining process.

High productivity can be reached by using:

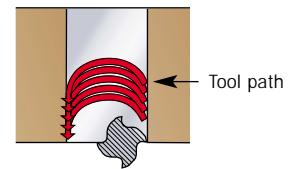
- Small radial engagement
- Large axial engagement
- High cutting speed (made possible by the small radial engagement)
- Large feed per tooth (made possible by the small radial engagement)

Trochoidal milling is specifically suited for highly productive roughing and rest milling in very hard materials (≤ 63 HRC).

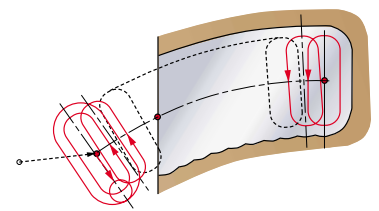
The productivity increase is considerable!



Conventional method

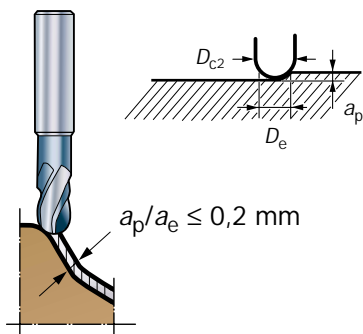


Trochoidal method



Fluting in circular cutting (Trochoidal).

Finishing in hardened tool steels



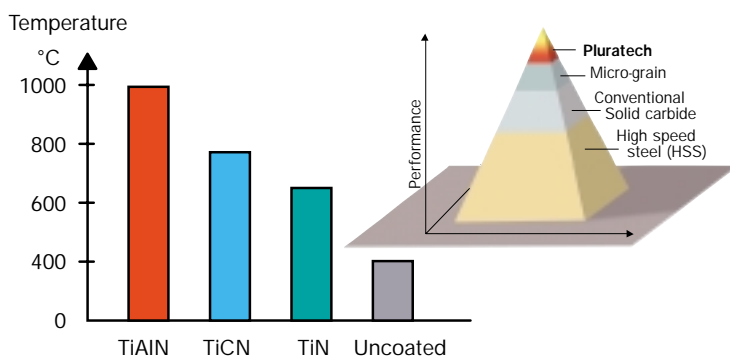
Shallow cuts and constant stock (chip cross section) are fundamental for HSM.

Lower removal rates — but faster!

One main parameter when finishing or super-finishing in hardened tool steel using High Speed Machining (HSM), is to take shallow cuts.

The depth of cut should not exceed 0,2/0,2 mm (a_e/a_p).

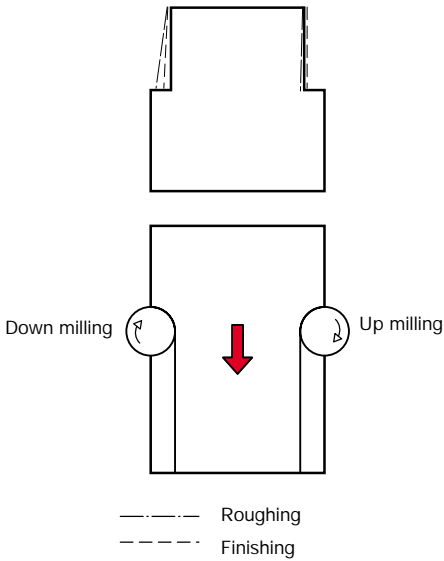
This is to avoid excessive deflection of the tool system and to keep high tolerance levels and geometrical accuracy on the machined die or mould.



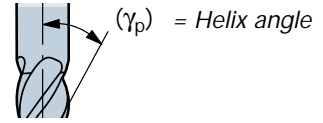
When finishing or super-finishing with high cutting speeds and feeds in hardened tool steels, it is important to choose tools that have a coating with high hot hardness, such as TiAlN.

Pluritech H10P is a new hitech carbide grade — with an extra fine grain carbide substrate and a coating giving a very good thermal barrier — for precision finishing in hard steels.

Tool deflection generates vibrations and is harmful for the form accuracy



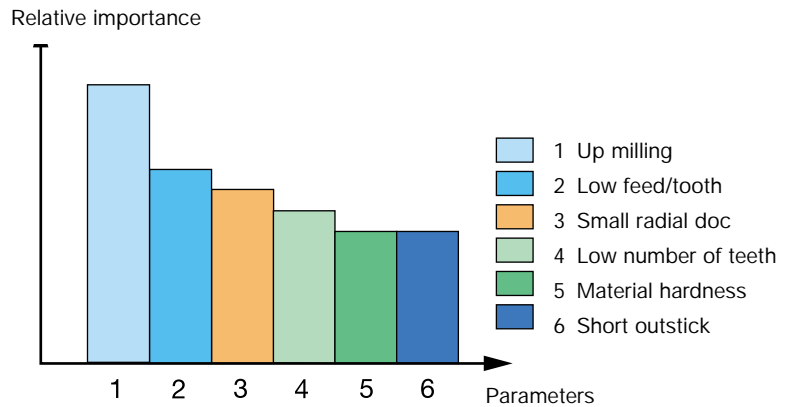
Generally a higher helix angle gives lower radial forces, less tool deflection — but also higher axial forces, which might tend to pull out the endmill from the holder or the work-piece from its clamping.



Example:

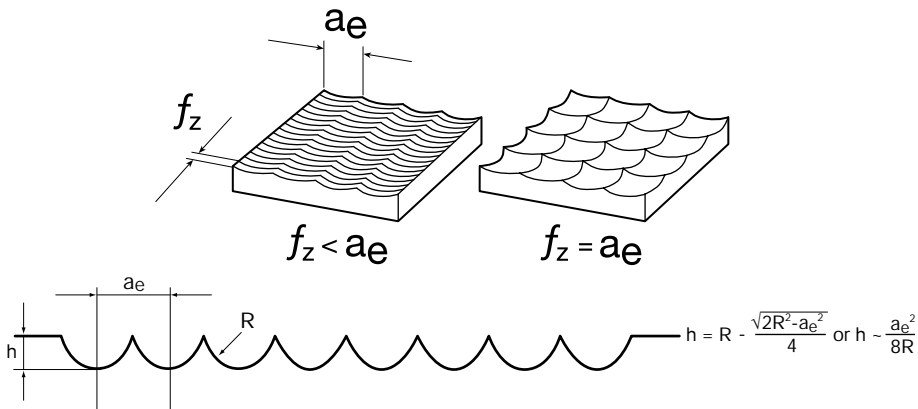
Bending	Roughing	Finishing
Up milling	-0.02 mm	0.00 mm
Down milling	0.06 mm	0.05 mm

Solide Carbide endmills — finishing / deflection



Sculptured surfaces

In finishing and super-finishing, especially in HSM applications, the target is to obtain high geometrical and dimensional accuracy and reduce or even eliminate manual polishing.



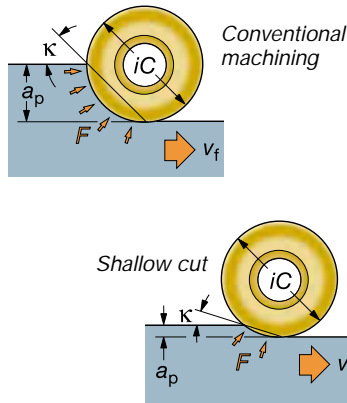
A rough rule of thumb:

A feed per tooth, equal to the radial depth of cut ($f_z = a_e$), is often favourable in finish milling, providing:

- Very competitive, short machining time.
- Increased accuracy and bearing resistance on surface gives longer tool life on die or mould
- Symmetrical surface texture in all directions and very smooth surface finish easy to polish.

The minimum required cusp or scallop height decides the f_z / a_e and R value.

The versatility of round insert cutters



The cutting action of round inserts is very smooth as the entering angle successively alters from nearly zero (very shallow cuts) to 90°.

At maximum depth of cut the entering angle is 45° and when copying with the periphery the angle is 90°. This also explains the strength of round inserts — the workload is built up successively.

Another parameter of importance is that the chip cross section is getting thinner in two planes. This gives very smooth engagements and retracts.

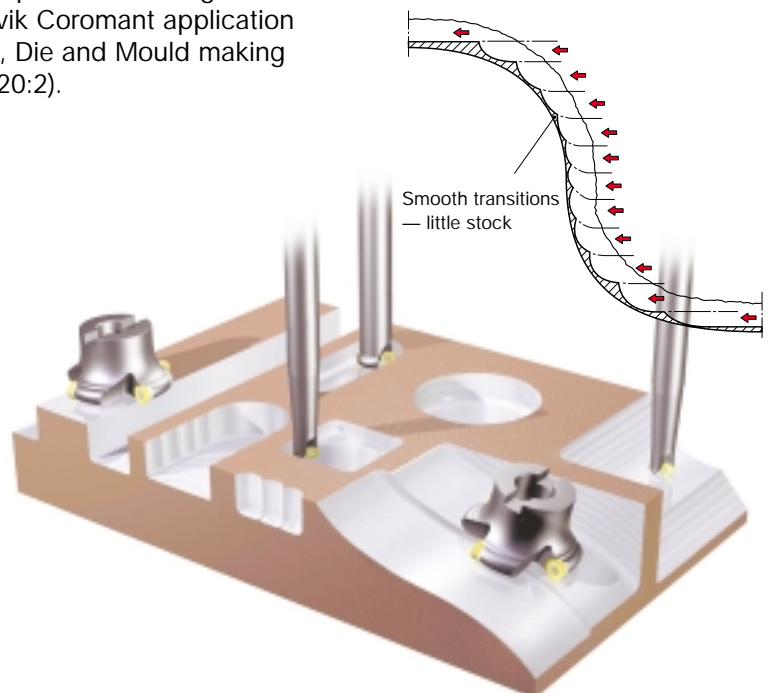
CoroMill 200 cutter is the first choice for roughing to medium roughing of cavities and open multi-curved surfaces. Depth of cut up to 10 mm and feed per tooth up to 0,6 mm.

CoroMill 300 toroid cutter is a high precision tool for light roughing and semi-finishing of dies and moulds and complicated profiles as well as finish milling. The long reach make them ideal for deep closed pockets and narrow cavities. Designed for small depth of cut, 0,2 - 1 mm and the feed per tooth up to 0,5 mm.

These cutters can, to a big extent, replace ball nose endmills with considerable productivity improvements.

When the machining is split up into segments it is important that each segment can be machined with one set of edges before tool change.

Detailed information about methods for machining in segments and other special cases is given in the Sandvik Coromant application guide, Die and Mould making (C:1120:2).



Applications with CoroMill 200 and CoroMill 300 insert cutters.



The term High Speed Machining (HSM) commonly refers to end milling at high rotational speeds and high surface feeds.

High Speed or High Performance Machining ...

... the fastest way to higher accuracy in die and mould manufacturing.

The HSM technique (High Speed Machining) can deliver — with outstanding productivity — the accuracy required for good economy in high precision die and mould making. The life cycle of the products produced by dies and moulds also tend to be shorter, making a faster manufacturing process even more important for the future.

Reduced machining times, higher precision, reduced lead times, reduced number of process stoppages, none or very little need for manual finishing work and reduced try-out time are only some of the opportunities in HSM.

What is today's definition of HSM?

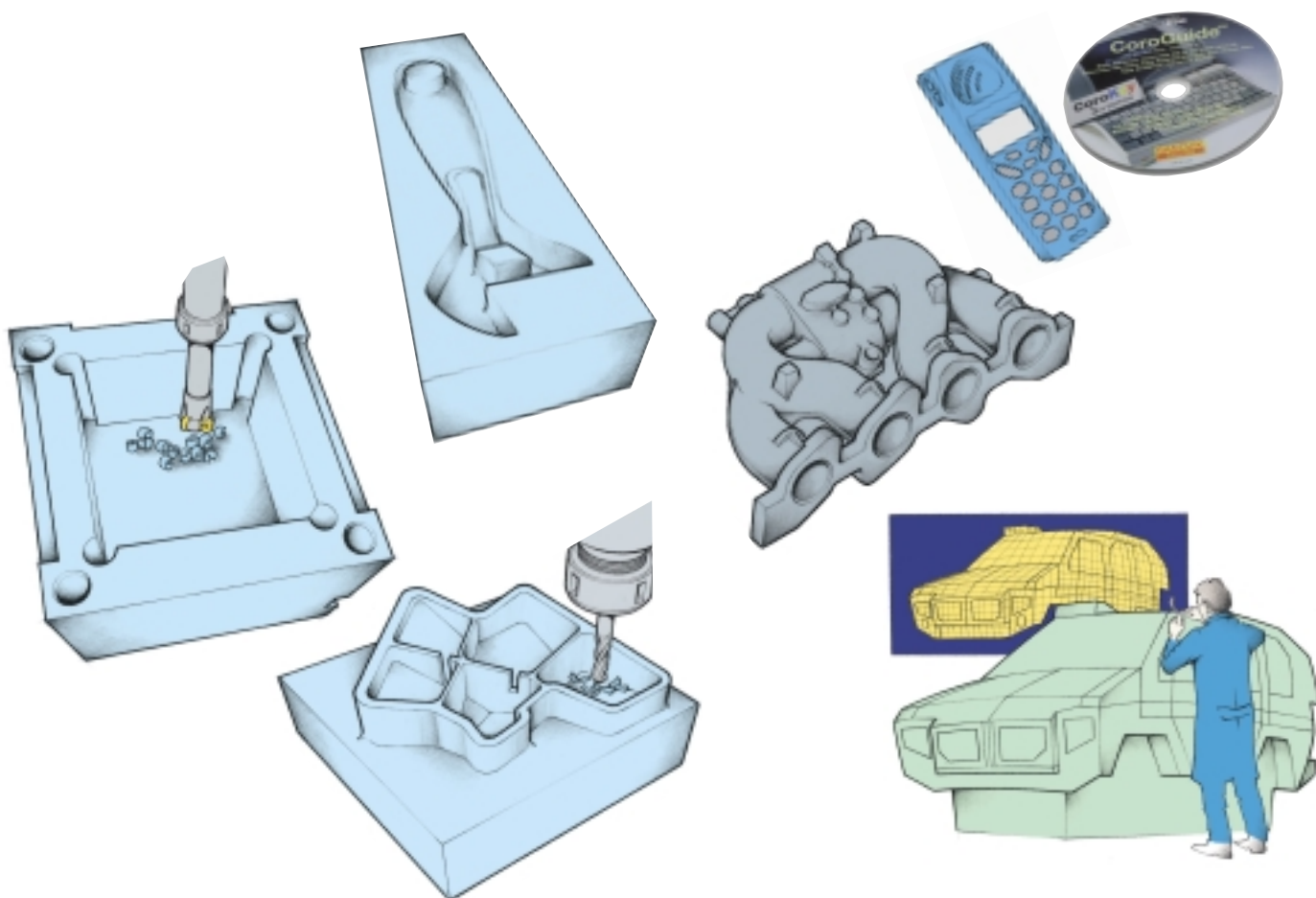
- HSM is, however, not just high cutting speed machining. It should be regarded as a process performed with very specific methods and production equipment.
- HSM is not necessarily high spindle speed machining. In many HSM applications large sized cutters are operating with lower spindle speeds.
- HSM is often applied to finishing in hardened steel with speeds and feeds as high as 4-6 times compared with conventional cutting data.
- HSM is high productive rough to finish machining of small sized components and finishing and super-finishing of components of all sizes.
- HSM will grow in importance the closer blanks become to the near net shape.
- HSM today, is mainly performed on taper 40 machines.



Blow mould.

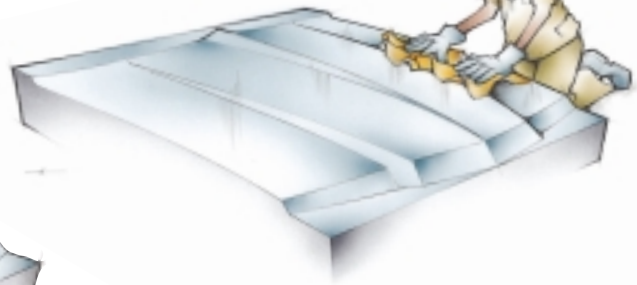
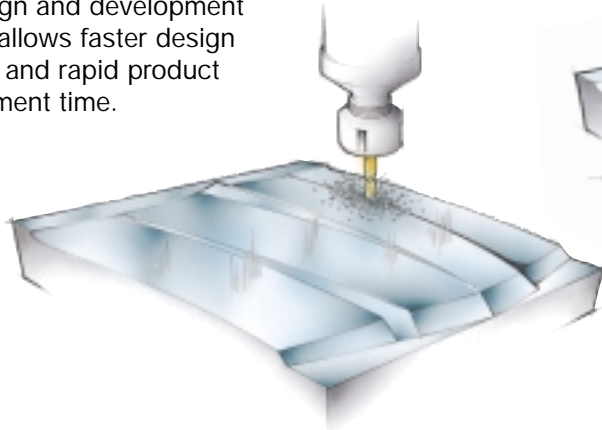
The application of high speed machining

- Milling of cavities in qualified, high-alloy tool steels up to 63 HRC and demanding tool steels.
- Injection and blow moulds of small sizes and often with deep cavities.
- Forging dies, ideal due to their often shallow geometries.
- Modelling and prototyping of dies and moulds made of easy-to-machine materials, with cutting speeds as high as 1500 – 5000 m per min. and consequently very high feeds.



Many manufacturing factors benefit from the use of the HSM technique

- The material — roughing to finishing in high hardened tool steels and other difficult to machine materials.
- The product quality — HSM offers a more accurate and economical substitute for manual finishing
- The process — shorter through put times of the product by fewer set-ups and simplified flow.
- The design and development work — allows faster design changes and rapid product development time.
- Product complicity — extremely thin wall sections of sophisticated design.
- Advanced production equipment — the strong development of cutting materials, holding tools, machine tools, controls and especially CAD/CAM features and equipment, has opened possibilities that can now be met by new production methods and techniques.



Some advantages with HSM

Shallow cuts generate low forces on the tool, resulting in only small but consistent tool deflection providing longer tool life and less machine tool maintenance.

Low operating temperature of cutting tool and workpiece due to extremely short contact time.

High productivity in general finishing and the possibility to achieve extremely good surface finish — often down to Ra ~ 0,2 microns.

Economy in roughing, semi-finishing and finishing provided the total material stock is kept low.

The geometrical accuracy produced dramatically reduces time consuming try-out and manual polishing.



Machining of extremely thin wall sections

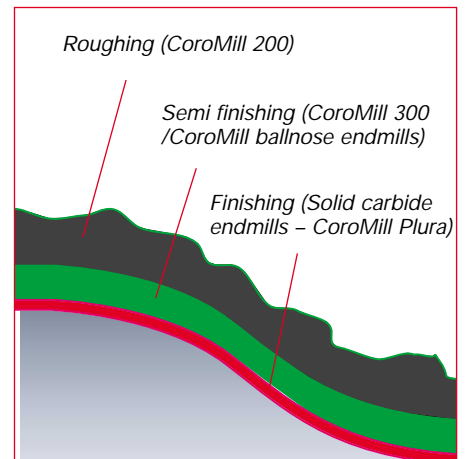
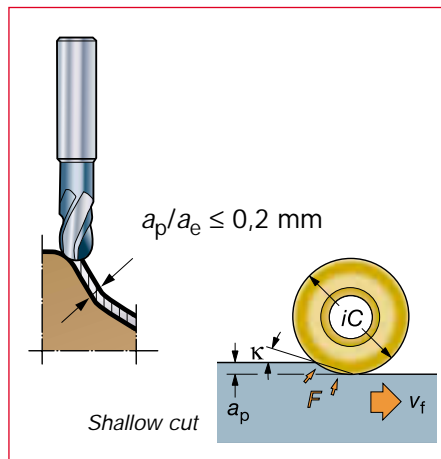
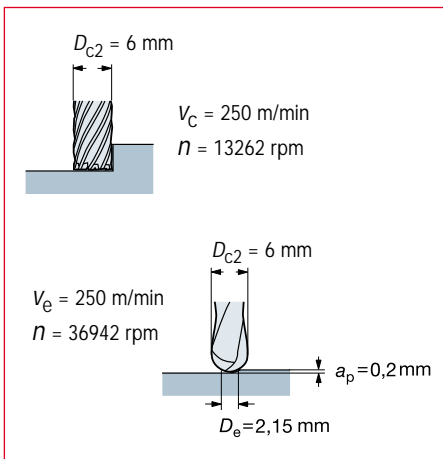
The picture shows a Coromant solid carbide endmill performing finish milling of a 0,2 mm thick aluminium wall with a height of 20 mm.

Dedicated tooling — a necessity for HSM conditions

To perform HSM applications it is necessary to use rigid and dedicated machine tools and controls with specific design features and options.

All production equipment — including the cutting and holding tools — has to be designed for the specific process of HSM.

CoroGrip high precision power chuck ensures high clamping force and torque.



True cutting speed

As cutting speed is dependant on both spindle speed and the diameter of the tool, HSM can be defined as true cutting speed above a certain level.

Particularly at smaller depths of cut and when using cutters with an entering angle smaller than 90 degrees, round inserts, ball nose endmills or a large corner radius, it is important to base calculations of the cutting speed on the effective diameter in cut (D_e).

The linear dependence between cutting speed (v_c) and feed rate results in "high feeds with high speeds". To compensate for a smaller diameter the rpm must be increased to keep the cutting speed constant – also resulting in a higher feed speed.

Shallow cuts

Fundamental for HSM applications is that the depths of cut — a_e and a_p , and the average chip thickness, h_m , - are kept much lower compared to conventional machining. The cutting forces (F) are consequently considerably smaller than in conventional machining.

Exceptions are machining in aluminium and other non-ferrous materials.

Constant stock

It is also necessary to use an advanced programming method, providing constant stock for each operation and tool — a basic criterion for high productivity, process security and good quality.

HSM in hardened tool steel

Roughing, semi-finishing, finishing and in many cases super-finishing operations are possible to perform in hardened steel using HSM, generally with CoroMill Plura solid carbide cutting tools.

Restmilling of corners and radii should always be done to create constant stock for the following operation and tool.

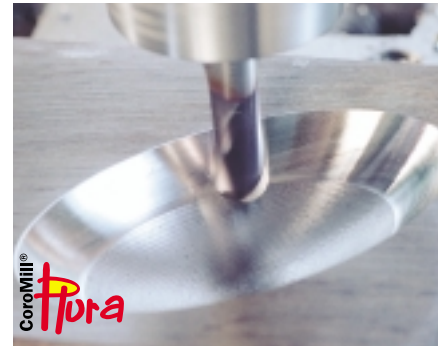
In many cases 3—4 different tool types are used. Some versions of the CoroMill Plura endmill GC1010, are specially designed for HSM.

These tools have reinforced cutting edges and neutral or negative rakes like CoroMill Plura H10P (mainly for materials above 54 HRC).

One typical and important design feature is the thick core for maximum bending stiffness. It is favourable to use a ball nose endmill with a short cutting edge and contact length.

Another design feature of importance is an undercutting capability, which is necessary when machining along steep walls with small clearances.

It is also possible to use smaller sized Sandvik Coromant tools with indexable inserts — especially for roughing and semi-finishing.



Coromant ball nose endmills, round insert cutters like the CoroMill 200 /300 and CoroMill Plura carbide endmills with large corner radii are suitable for hard steels.

HSM cutting data recommendations

Typical cutting data for solid carbide endmills with a TiC,N or TiAlN-coating in hardened steel (54-58 HRC):

The recommendations are dependent of the usual variations in operation conditions, tool diameter, overhang etc., and should be regarded as *typical values only*. The v_e value is calculated on the effective diameter in cut (D_e).

Roughing

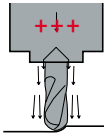
v_e 100 m/min,
 a_p 6-8% of the cutter diameter,
 a_e 35-40% of the cutter diameter,
 f_z 0,05-0,1 mm/z

Semi-finishing

v_e 150-200 m/min,
 a_p 3-4% of the cutter diameter,
 a_e 20-40% of the cutter diameter,
 f_z 0,05-0,15 mm/z

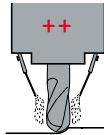
Finishing and super-finishing

v_e 200-250 m/min,
 a_p 0,1-0,2 mm,
 a_e 0,1-0,2 mm,
 f_z 0,02-0,2 mm/z



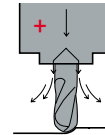
Compressed air

The best way to ensure a perfect chip evacuation. It should be well directed to the cutting zone.



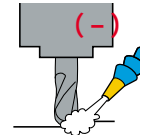
Oil mist

The second best is to have oil mist under high pressure directed to the cutting zone.



Cutting fluid

Third comes coolant with high pressure (approx. 70 bar or more) and good flow.



The worst case is ordinary, external coolant supply, with low pressure and flow.

Dry milling extends the life of the cutting edge

The Coromant cemented carbides, especially the *coated GC*, perform better as regards to tool life and reliability when used in a dry milling environment.

The effect of cutting fluid in milling emphasises the temperature variations that take place with the cutting edge going in and out of cut.

Adding cutting fluid will increase variations by cooling the cutting edge while being out of cut. These variations or thermal shocks lead to cyclic stresses and thermal cracking.

This of course will result in a premature end of tool life. The hotter the machining zone is, the more unsuitable it is to use cutting fluid.

Coromant carbide grades, cermets, ceramics and CBN tools are developed to withstand constant, high cutting speeds and temperatures and should always be applied in dry conditions.

When using coated milling grades the thickness of the coating layer plays an important role.

A thin coating leads to less thermal tension and stress.

Tool life differences of up to 40%, and in some specific cases even more, are not unusual, to the advantage of dry milling.

Cutting fluid unfavourable in HSM

The temperature variations generated in milling are even further accentuated in HSM than in conventional operations, making the use of cutting fluid even more unfavourable.

One of the main factors for success in HSM applications is the total evacuation of chips from the cutting zone. This must be done instead by compressed air.

When working in hardened steel it is absolutely essential to ensure predictable tool life of the cutting edges and for good process security to avoid recutting of chips.

High pressure oil mist is preferred for lubrication and also for evacuation of the chips.

If using cutting fluid, it should be applied copiously and a cemented carbide grade suitable for both wet and dry conditions should be selected.

The advantages and benefits of dry milling — and also some disadvantages — are dealt with in the Sandvik Coromant application guide Die and Mould Making (C-1120:2).

Exceptions

There are some exceptions when the use of cutting fluid can be justified to some extent in **conventional machining**:

- In machining of heat resistant alloys with low cutting speeds.
- In deep slotting operations.
- To keep the workpiece temperature low.
- To prevent heat generation into the cutter, holding tool and spindle.
- In finishing of stainless steel and aluminium, to prevent smearing of small particles into the surface.
- In machining of thin walled components to prevent geometrical distortion.
- To reduce dust in cast iron machining.
- To prevent corrosion of components and vital machine parts.

The choice of holding tools

One of the main criteria when choosing both holding and cutting tools is to have as small run-out as possible. The smaller the run-out is, the more even the workload will be on each insert in a milling cutter

In HSM applications the size of run-out is specifically crucial. The TIR (Total Indicator Readout) should be maximum 10 microns at the cutting edge. A good rule of thumb is: For each 10 microns in added run-out — 50% less tool life!

Coromant Capto® improves productivity

The Coromant Capto coupling has the best stability characteristics on the market and its modularity increases the flexibility and the number of possible tool combinations.

Extended tools for machining centres are frequently required to be able to reach the surfaces to be machined.

With Coromant Capto it is possible to build an assembly with long and short basic holders, extensions and reductions, so the right length can be achieved. It is very important to use shortest possible tool overhang in milling operations. Many times a small difference in length can be the difference in working with a good (or medium productivity) or not working at all.

If solid tools are used, they are often too short or too long. In many cases special solid tools have to be used, which is expensive and also requires long delivery times.

Modular tools are built together in minutes!

A special tool with Coromant Capto coupling can also be used in other machines with different spindles. Only a new standard basic holder is required.



With Coromant Capto® it is always possible to use optimal tool assemblies and cutting data for best productivity.



Two face contact

Press fit and torque loads spread symmetrically around the coupling, without load peaks, are the main reasons for the outstanding performance of the Coromant Capto system.

Tight press fit guarantees that there is no play in the coupling. It can transmit torque in both directions.

Due to the combination of press fit and high clamping force the coupling gets **a two-face contact**. A large surface contact area around the polygon and on the flange transmits the cutting force (F). No load peaks. Axial positioning of the cutting edge will remain constant despite high axial cutting forces.

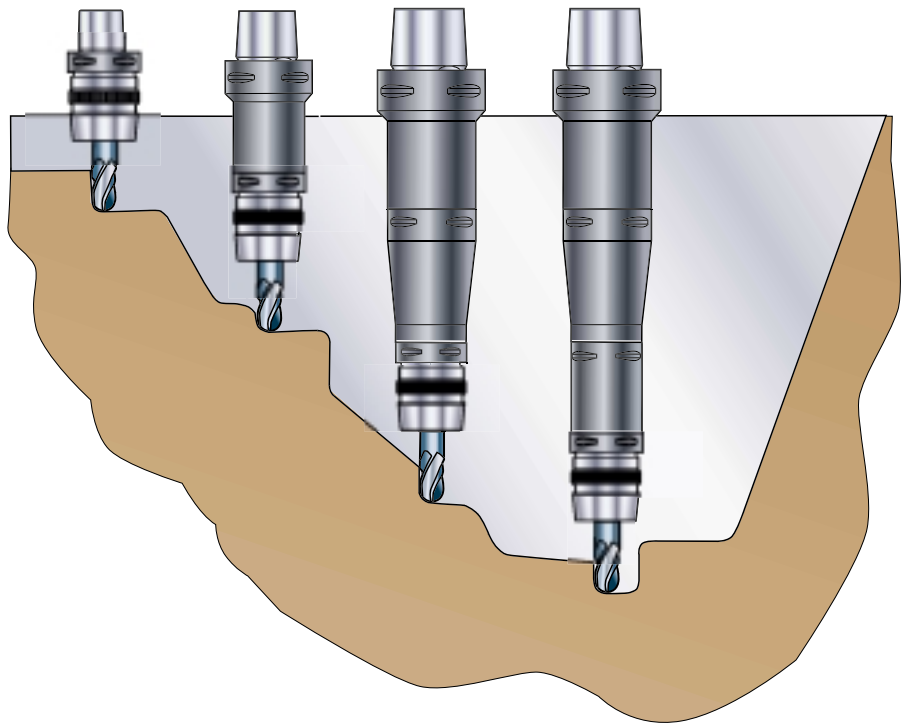
To maintain maximum productivity when roughing a cavity it is important to choose a series of extensions for the cutter. It is a very bad compromise to start with the longest extension, as productivity will be very low.

The recommendation is to change extension at pre-determined positions in the program. The geometry of the die or mould decides these positions.

Cutting data should also be adapted to each tool length to keep up maximum productivity.

When the total tool length, from the gauge line to the lowest point on the cutting edge, exceeds 4-5 times diameter at the gauge line, tuned, tapered bars should be used.

When using extended tools it is important to choose the largest possible diameter on extensions and adaptors relative to the cutter diameter. Every millimeter is important for maximum rigidity, stiffness and productivity.



Extend the tools gradually when roughing a cavity to maintain maximum productivity.

It is most often sufficient to have 1 mm radial difference between the holding and cutting tool. The easiest way to achieve this is to use oversized cutters.

A unique Coromant Capto solution for elimination of vibrations in milling or boring operations is the damped adaptors — but also a rigid fixturing of the work piece is of utmost importance.

A rigid clamping a necessity

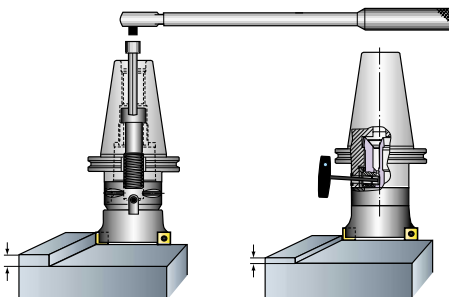
To avoid vibration in a milling or boring operation a rigid clamping is required. It is mainly the bending moment that is critical, and the most important factor for taking up high bending moment in the clamping force.

Coromant Capto uses centre bolt clamping, which is the strongest and cheapest way to clamp.

Normally the clamping force is double compared with any side locking (front clamping) mechanisms.

Normally more than 100% productivity increase can be achieved with centre bolt clamping.

Unbalance vs vibration, see the Sandvik Coromant application guide Die & Mould Making (C-1120:2).



Maximum stability and minimum run-out — from the spindle to the chip!

CoroGrip is a holding system that covers all applications from super-finishing to heavy roughing. One holder can clamp all types of tools, from facemills to drills with both cylindrical, Whistle Notch or Weldon shanks. Standard collets such as collets for HydroGrip, BIG, Nikken, NT etc., can also be used in the CoroGrip holders.

The bore and shank (taper) on CoroGrip holders are ground to very close tolerances, which enables very small run-out, only 0.002 — 0.006 mm at 4 x D.

The symmetrical clamping force in the system keeps the low run-out for a very long time. The rigid design of the holders also keeps the low run-out in roughing operations.

It is possible to clamp Weldon/Whistle notch shanks directly in the holder and also clamp the shanks half way into the holder. The clamping force in the CoroGrip holders is extremely high and its balanced design makes it

perfect for high speed machining, which makes the CoroGrip a good alternative to shrink fit holders.

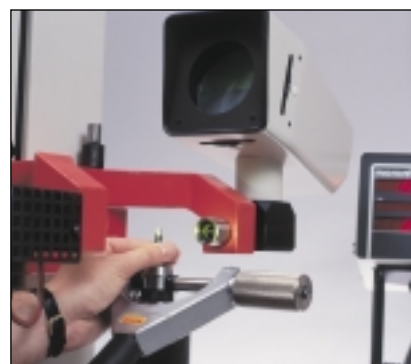
To obtain the high clamping force an external hydraulic pump has to be used. With a pressure of 700 bar, the outer sleeve is pushed up on the taper, when the tool is clamped, and down on the taper when it is unclamped. When the tool is clamped the clamping mechanism is self-locking and there is no hydraulic pressure in the holder during machining.

All CoroGrip power chucks are individually balanced to a quality of G 2.5 at 25000 rpm for small sizes (Taper 40, HSK 32-40-50-63, Coromant Capto C3 — C6) and G 2.5 at 14000 rpm for larger sizes (Taper 50, HSK 80-100, Coromant Capto C8).

CoroGrip high precision power chuck and CoroMill Plura endmill is the perfect combination for HSM.



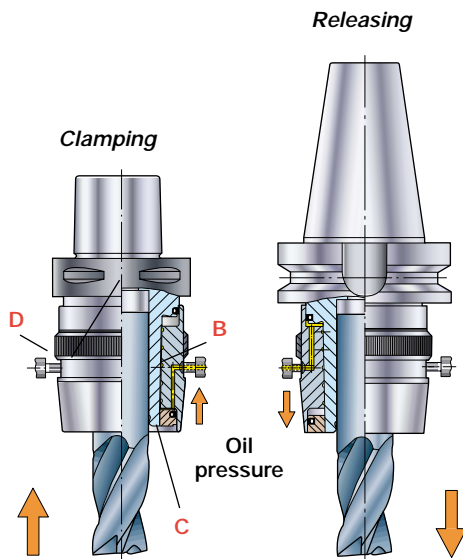
It takes less than 20 seconds to change a tool!



Three different items of equipment are supplied for changing tools in the CoroGrip holder — a portable manual hydraulic pump and a stationary pump, foot and table designs, using the regular pneumatic system available in the work shop (min. 5 bar required).

By using either of these systems tool changing is ergonomically and easily performed - and the same gripping force is always applied to the tool shank.

The tool length can be set within +/- 3-5 microns. The tool will maintain its axial position stability absolutely during the clamping process.



CoroGrip high precision chuck

- A. Inner body with a small angle taper (around 2 degrees angle)
- B. Movable sleeve.
- C. Locking ring
- D. Plastic cover ring

A safe grip in all applications — from finishing to heavy roughing

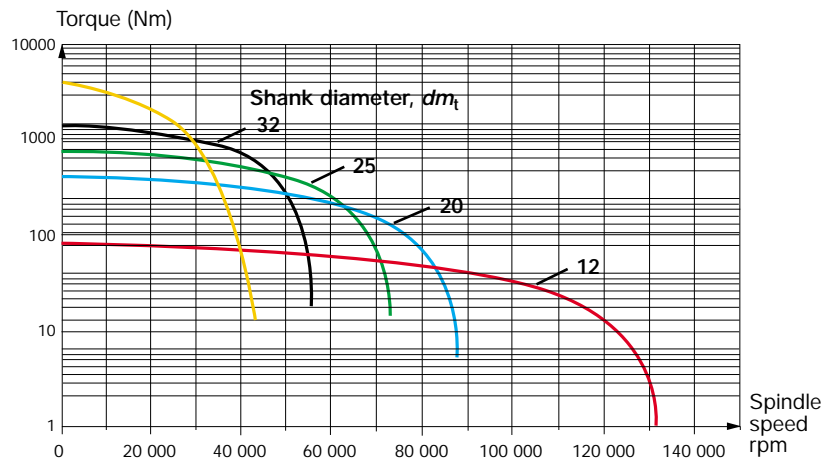
Extremely high torque transmission capability prevents the tool from slipping.

Also in rough operations and — as the mechanism is self-locking, no hydraulic pressure is retained in the mechanism during machining.

The clamping force is consistent during the entire operation and maintained during long time in use.

CoroGrip — the chuck for HSM

Torque transmission by spindle speed (Tool shank tolerance h6).



The high and reliable clamping force generated, remains stable on a safe level also at extremely high spindle speeds.

This together with the individually balanced holder makes CoroGrip a most suitable chuck for High Speed Machining.

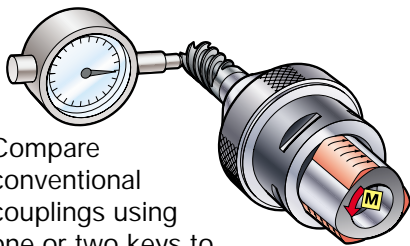
Transmission torque comparison

Shank diameter	Torque, Nm	Holder
12 mm	93	CoroGrip
	72	Shrink fit
	50	Hydraulic chuck
20 mm	440	CoroGrip
	243	Shrink fit
	181	Hydraulic chuck
25 mm	804	CoroGrip
	421	Shrink fit
	365	Hydraulic chuck
32 mm	1512	CoroGrip
	---	Shrink fit
	651	Hydraulic chuck

CoroGrip provides a clamping force considerably higher than shrink fit holders and hydraulic chucks. Its precision is on level with shrink fit holders.

Comparison between holders for clamping of shaft tools

	Weldon/ Whistle Notch holder	Collet chuck DIN 6499	Power chuck	HydroGrip Hydraulic chuck	Shrink fit holder	CoroGrip Hydro-mechanical
Type of operation	Heavy roughing — Semi-finishing	Roughing — Semi-finishing	Heavy roughing — Finishing	Finishing	Heavy roughing — Finishing	Heavy roughing — Finishing
Transmission torque	+++	++	++	+	+++	+++
Accuracy TIR 4 x D (mm)	0,01 – 0,02	0,01 – 0,03	0,003 – 0,010	0,003 – 0,008	0,003 – 0,006	0,003 – 0,006
Suitable for high speeds	+	+	++	++	+++	+++
Maintenance	None required	Cleaning and changing collets	Cleaning and changing spare parts	None required	None required	None required
Possibility to use collets	No	Yes	Yes	Yes	No	Yes

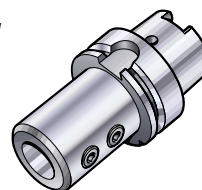


Compare conventional couplings using one or two keys to transfer torque. There will always be a certain amount of play in the coupling which can cause the following problem:

- Asymmetric load situation with high load peak on the key. Causes high surface pressure and shorter tool-life, as well as higher run-out.

HSK is a spindle interface and not a modular coupling and should not be mistaken for a modular tool system. The clamping force is not as great as that produced by Coromant Capto with centre bolt clamping, which is mainly required for milling and boring operations.

HSK extensions can, however, be used for light drilling and reaming operations, where there are low radial forces.



Tool material and its machinability

When the alloy and carbon content go up the machinability goes down!

The higher alloy steel has, the more difficult it is to machine with cutting tools. The same relationship prevails to hardness and machinability. When the hardness goes up the machinability is reduced.

To increase the machinability for materials the sulphur content can be increased in the material with higher machinability as a result. However, sulphur reduces the mechanical properties in the steel, e.g. the toughness. Another drawback with sulphur is that the possibility to get a good surface finish decreases as the sulphur content is increased. Therefore the sulphur content is very low in a high quality tool steel.

Cold work steels are often alloyed with carbon, chromium and vanadium to get hard and abrasive resistant carbides in the steel. The hard carbides will reduce the machinability in the steel by giving a high wear on the cutting tool.

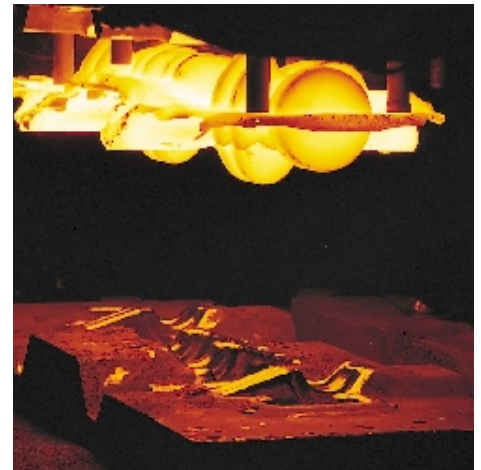
Choosing steel grade is not a simple task!

The choice of steel grade is often made at the design stage of a mould in order to have the material in-house and ready to be machined when the design is finished. In many cases this choice is a compromise between the wishes of the mouldmaker and the moulder.

Typical in-service hardness is in the range of 32 - 58 HRC for die and mould materials. The softer annealed ferritic steels with an approximate Brinell hardness of 180 are characterised by a very uniform machinability. The size of the die or mould often decides the hardness of it.

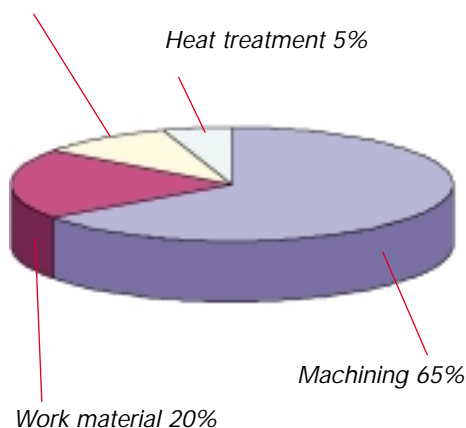
In the chart on the next page you can see factors that the three different parties steel manufacturer, mouldmaker and enduser of the mould wish to get out of the steel.

A substantial proportion of production costs in the die and mould industry is involved in the machining process, as generally large volumes of metal are removed. The finished tool is also subjected to strict geometrical and surface tolerances.



This means that the machinability of the tool material, its polishability and heat treatment properties all have to be carefully considered in order to achieve maximum production economy.

Assembly & adjustment 10%



Machinability

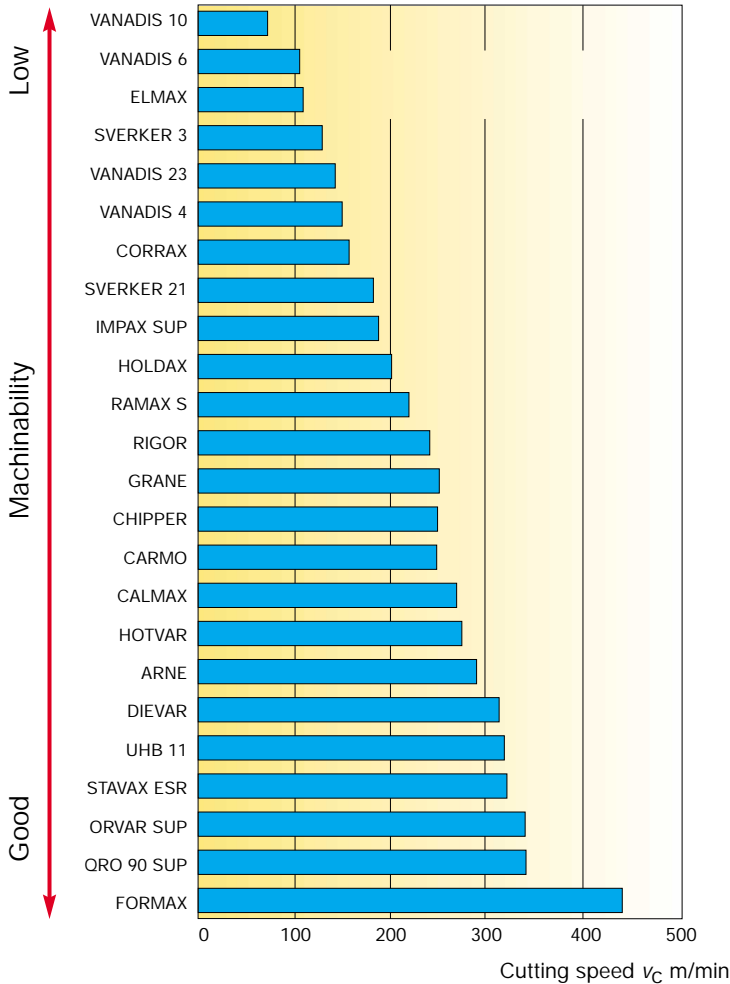
Percentage wise the machining cost is a large part of the total production cost for a die or mould.

This means that the machinability is of utmost importance for an economical production of dies and moulds.

There are five main factors that influence the machinability of a material:

- Chemical composition
- Structure
- Hardness
- Non-metallic inclusions
- Residual stresses

Machinability in different tool steels, facemilling with carbide inserts



Example of material and hardness for different lengths of production series of a plastic mould:

- For long runs > 1 000 000 shots: High hardness steel should be used 48 — 65 HRC; Calmax, Grane, Orvar Supreme, Stavax ESR, Corax, Rigor, Vanadis 4, Elmax
- For medium runs 100 000— 1 000 000 shots: Pre-hardened steel should be used 30 — 45 HRC; Impax Supreme, Ramax
- For short runs < 100 000 shots: Soft annealed steel or aluminium should be used 160 — 250 HB; Calmax, Grane, Alumec

Requirement	Material properties	Influence
Favourable impact strength and good polishability	High purity	Poor chip breaking
Good wear resistance	Hard carbides in steel matrix	Considerable tool wear
Avoid subsequent heat treatment	High delivery hardness	High cutting edge temperature, high tool wear

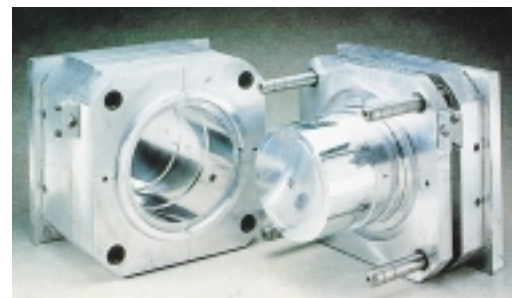
Material for prototypes and short series

ALUMEC has an excellent machinability also with high cutting speeds, that leads to lower mould cost and shorter delivery time. This makes it suitable for production of prototypes as well.

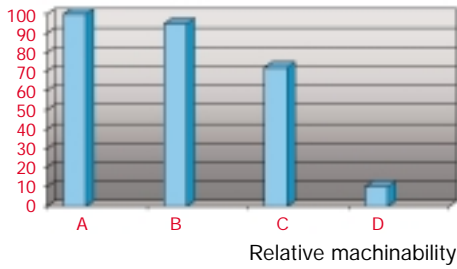
ALUMEC is a high-tensile aluminium alloy that is produced in the form of hot-rolled, heat-treated plate. It is subjected to a special cold-stretching process for maximum stress relief. This material has achieved a widespread use within the mechanical engineering industry because of its high strength and good stability.

ALUMOLD is another material with characteristics and advantages similar to ALUMEC. These materials are ideal for short and medium-long production runs, which are not subjected to high pressure or abrasive plastics.

KIRKZITE is another common material mostly used for prototypes.



The machinability of cast iron



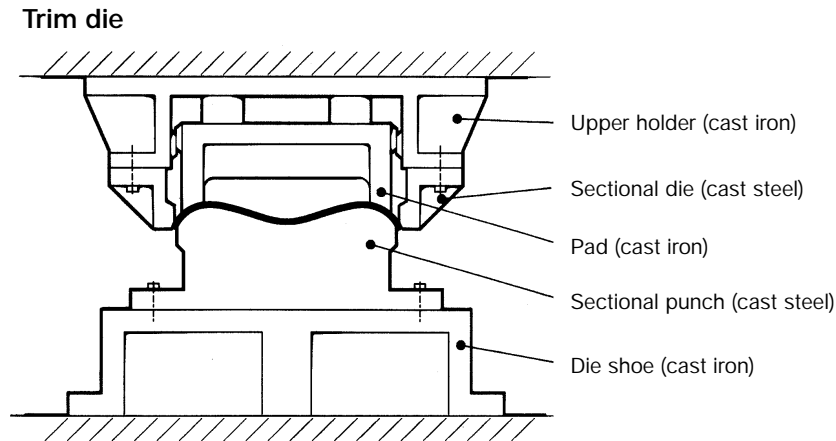
The machinability of cast iron is also indicated by the hardness — often measured in Brinell — which deteriorates with increasing value. The hardness is, however, an unreliable measurement of machinability as there are two factors that the value does not show.

Abrasive hardness, due to sand inclusions and free carbides, is very negative for machinability.

A cast-iron of 200 HB and with a number of free carbides is more difficult to machine than a cast-iron of 200 HB and a 100% pearlitic structure with no free carbides. The general relative machinability (M) of the four main kinds of cast-iron is indicated in the bar chart where (A) is grey cast-iron, (B) malleable, (C) S.G. iron and (D) chilled, white cast-iron.

In general, the higher hardness and strength the cast iron has, the lower the machinability will be and the shorter the tool-life.

Typical tool properties generally required to machine cast-iron are high hot-hardness and chemical stability, but depending upon the operation, workpiece geometry, machining conditions and toughness. Thermal shock resistance and mechanical strength are also needed from the cutting edge.

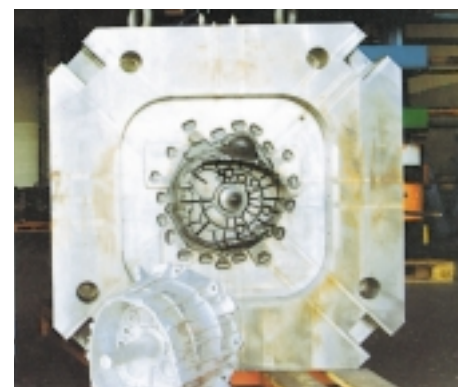
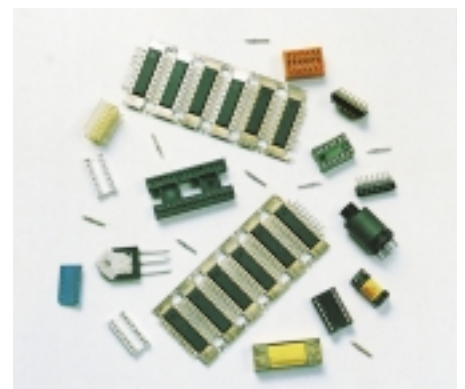
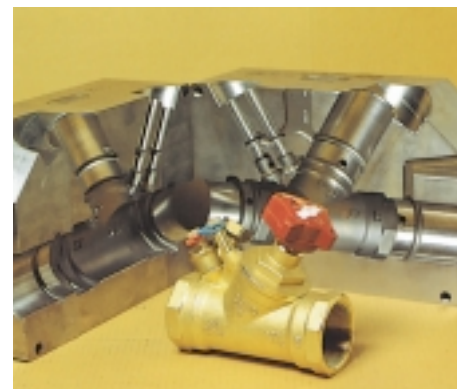
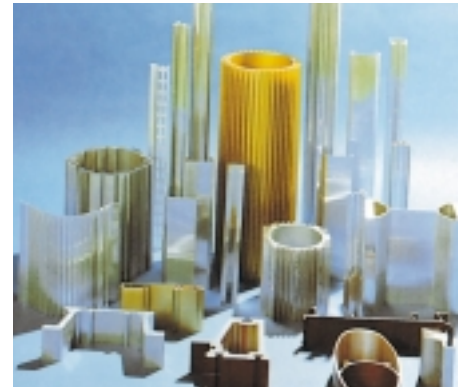


Materiel type	Hardness HB	Area of use	CMC Coromant	Sweden SS	Germany DIN	USA AISA	UK BS	Japan JIS	France ANFOR	Italy UNI	Brazil
Grey Cast-iron	150-200	Frames	08.1/2	0125	GG25	A48 Class 40B	BS1452 G150	FC250 FC300 FC350	Ft25	G25	GG25
Alloyed grey-iron	220-260	Dies	07.2	0852	GG26	—	G250 + Cr % Mo	Not available	Mn450	GNM45	GG26
Alloyed grey-iron	210-240	Dies	07.2	0852	GG26	—	BS1452 G250	Not available	Mn450	GNM45	GG26
Nodular cast-iron	200-260	Dies & stamps	09.1	0717-12	GGG60	—	BS2989 600/3	FCD450 FCD550	FGS400	GS370-17	GGG60
Nodular cast-iron	230-300	Dies & stamps	09.2	0732-3	GGG60	A536 Grade 80-55-06	BS2989 700/2	FCD600 FCD800	FGS600	GS600	GGG60

PRODUCTIVITY IN DIE & MOULD MAKING

Uddeholm grades	ASSAB	Sandvik Coromant CMC	Sweden SS	Germany W.-No	USA AISI/ASTM	UK BS	Japan JIS
Cold work steels							
ARNE ®	DF-2	02.1	2140	1.2510	O1	B01	SKS3
VANADIS 23	ASP-23		2725	1.3344	M3 Class 2		
VANADIS 30	ASP-30		2726	(1.3204)			
VANADIS 60	ASP-60		2727	1.3241			
CARMO/CALMAX	/635H/635			1.2358			
CHIPPER ®/VIKING	VIKING			(1.2631)			
FERMO ®	-						
RIGOR ®	XW-10	03.11	2260	1.2379	A2	BA2	SKD12
SVERKER ® 3	XW-5	03.11		1.2436	D6	B06	(SKD2)
SVERKER ® 21	XW-41	03.11	2312	1.2379	D2	BD2	SKD11
VANADIS 10	VANADIS 0		2310				
VANADIS 4	VANADIS 4						
VANADIS 6	VANADIS 6						
Hot work steel							
HOTVAR							
ALVAR ® 14				1.2714			SKD4Mod
ORVAR 2 M	8402	03.11/03.22	2242	1.2344	H13	BH13	SKD61
ORVAR ® SUPREME	8407	03.11/03.22	2242	1.2344	H13	BH13	SKD61
ORO 90 SUPREME	QRO 90						SKD7Mod
VIDAR ® SUPREME				1.2343	H11	BH11	SKD6
DIEVAR							
Mould steel							
ELMAX	ELMAX						
GRANE ®		02.1	2250	(1.2721)	(L6)		
IMPAX ® SUPREME	718 SUPREME			1.2738	(P20)		
OPTIMAX			2314	(1.2083)			SUS420
RAMAX ® S	168						
STAVAX ® ESR	STAVAX ESR s-136		2314	(1.2083)			SUS420
CORRAX							
Holder steel							
FORMAX ®		01.2	(2172)				
HOLDAX ®	HOLDAX			1.2312	4140		
UHB ® 11	760	01.2	1650/1672	1.1730	1148		

	France ANFOR	Italy UNI	Spain UNE	Brasil
	90MWCV5 Z120WDCV06-05-04-03 ASP30 ASP60 Z100CDV5 Z210CW121 Z160 (CDV12)	95MnWCr5KU HS6-5-3 X155CrMoV51KU X25CRW121KU X155CrVMo121KU	F-5220 F-5605 F-5227 F-5213 F-5219	VND VC131 VD2
	Z5NCDV7 Z40CDV5 Z40CDV5 Z38CDV5	56NiCrMo7KU X40CrMoV511KU X40CrMoV511KU	F-5307 F-5310 F-5318 F-5317	VMO VH13 VH13 VPCW
	40CMND8 Z30C17 Z40C13	X41Cr13KU	F-5305 F-5303	VCO VP20 VC150 VP420IM VC150
	40CMD8+S XC48		F-5304 F-1142	





*Sandvik Coromant Tailor Made concept
The quickest way for you to obtain a custom-built tool.*

- **Quick quotation**
 - **Easy to order**
 - **Competitive delivery**

Define the tool you require and we will tailor it for you!

Choose the tool you require and we will tailor it for you to your dimensions. The choice is almost endless with our Tailor Made Service.

Easy to order — Quick delivery

1. Define your type of operation and specify your tool dimensions, then simply forward your inquiry to us.
2. Within a few hours a quotation is dispatched to you, including delivery time, price and a drawing.
3. On receipt of your order we transmit data to the shop floor and production starts immediately.
4. You will be surprised how soon your Tailor Made tool will arrive at your work place.

And, of course, we are certified according to ISO 9001.



By controlling every step in the production chain of our drills — from carbide powder to finished product — we always guarantee consistent performance, predictable tool life and a fast delivery.

The Dream cutter
Enjoy the low noise and
high shearing action of the
long edge milling cutters
with optimized design for
superior surface finishes.



When standard and Tailor Made tooling solutions are not enough

.... Sandvik Coromant manufacture engineered solutions for improved productivity.

In die and mould making - when the cost of components is often high - an engineered solution can be very cost effective.

Sandvik Coromant has extensive experience in value engineering - effective tool solutions dedicated for extraordinary demands.

Please contact your local Sandvik Coromant sales representative for consultation.