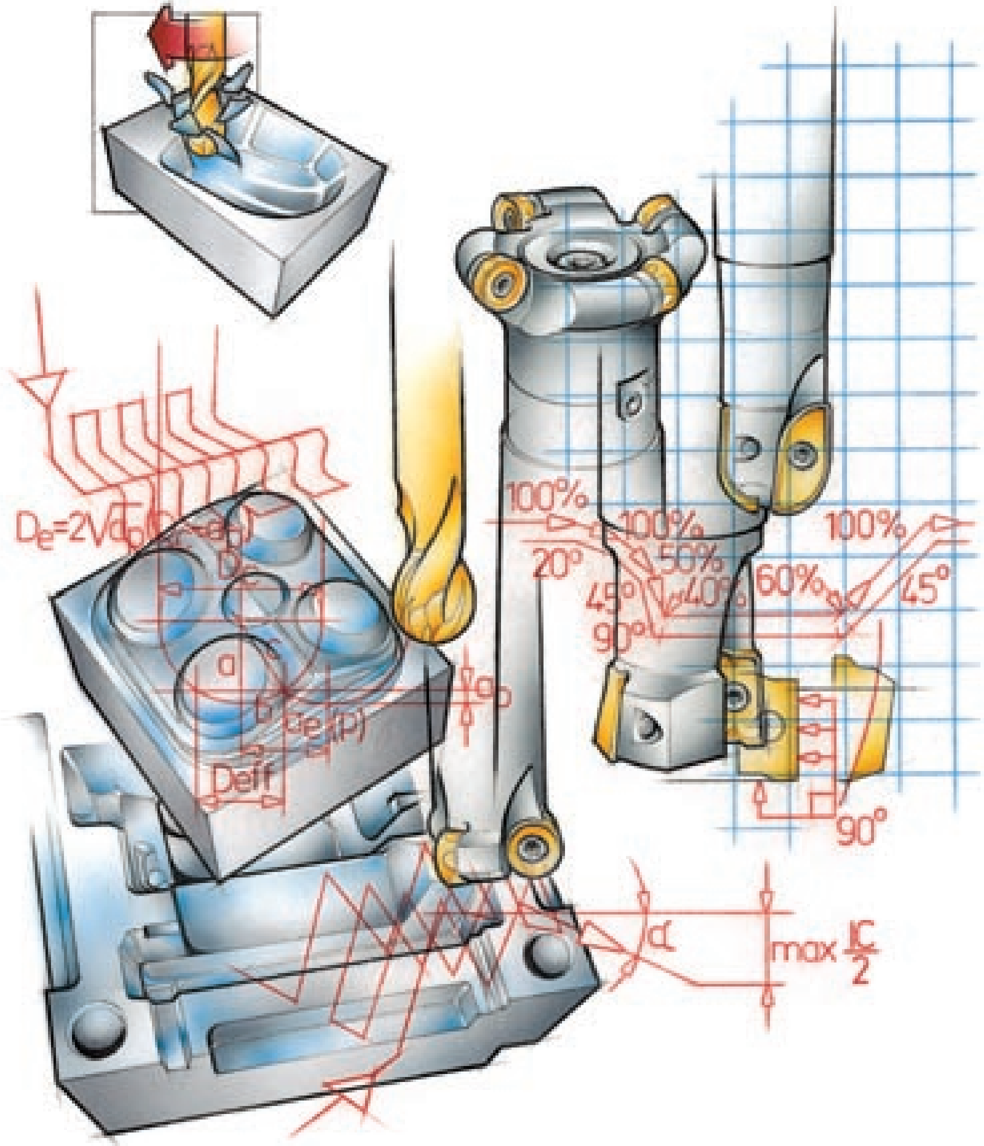
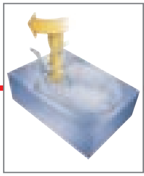
APPLICATION GUIDE

SANDVIK
Coromant

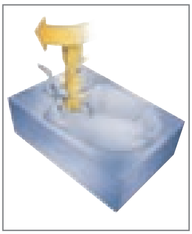
Die & Mould Making





CONTENTS

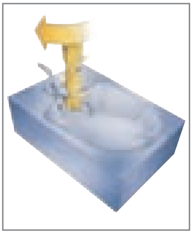
INTRODUCTION



Within the die and mould making industry the development has been strong the last years. Machine tools and cutting tools get more and more sophisticated every day and can perform applications at a speed and accuracy not even thought of ten years ago. Today CAD/CAM is very common and to machine with so called HSM (High Speed Machining) it is a necessity.

To manufacture a die or mould, many different cutting tools are involved, from deep hole drills to the smallest ball nose endmills. In this application guide the whole process of die and mould making will be explained with focus on the machining process and how to best utilise the cutting tools. However, programming of machine tools, software, workpiece materials, the function of different types of dies and mould will also be explained. First let us take a look at a simplified flowchart to see what the different stages are in the die and mould making process.

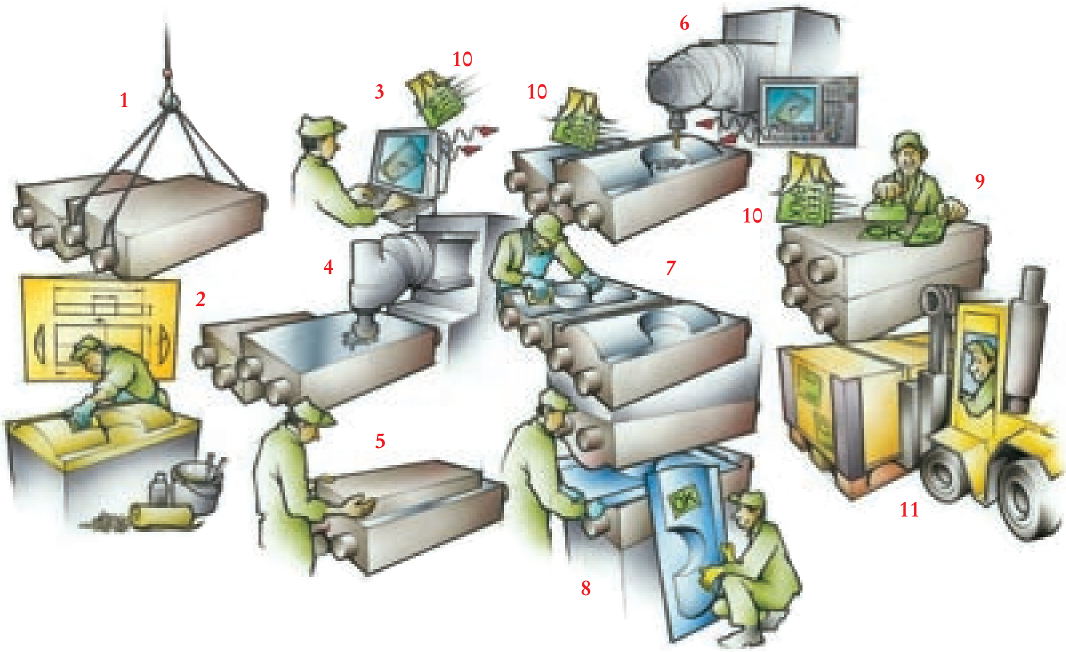
DIE CONSTRUCTION WORK FLOW



Simplified, the die construction work flow can be explained as in the illustration below.

1. Receiving - standard die parts, steel castings, planning and scheduling
2. Model shop - tooling aide and checking fixtures
3. CAM-room - schedule, exchange reports DNC/CNC programme 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



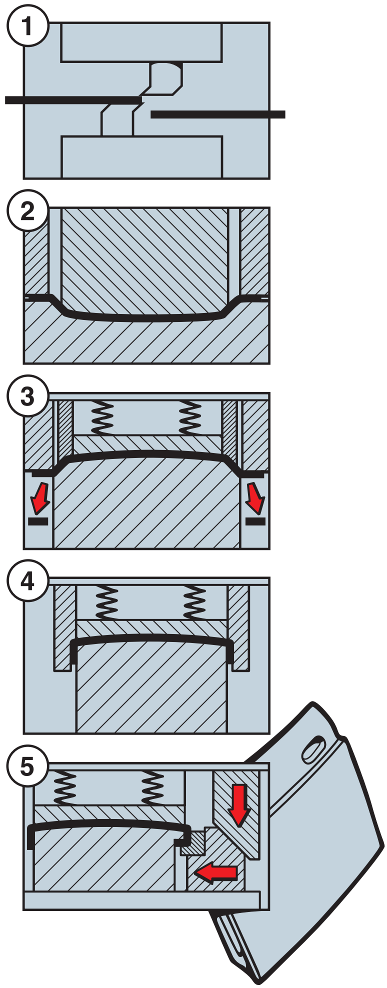
When a tool has to be made, for instance, to a hood of a car you do not make one press tool and the material for the hood goes in on one side, gets pressed and comes out finished on the other side. It is often complicated shapes, geometries, which has to be pressed, with different radii and cavities, to close tolerances. To do this the material for the hood has to be pressed in several press tools where a small change in the shape is made each time. It is not unusual that up to 10 different steps are needed to make a complete component.

There are 5 basic types of dies and moulds; pressing dies, casting dies, forging dies, injection moulds and compression moulds.

Pressing dies are for cold-forming of, for instance, automobile panels with complex shapes. When producing a bonnet for a car many pressing dies are involved performing different tasks from shaping to cutting and flanging the component. As mentioned earlier there can be over 10 different steps in completing a component. The dies are usually made of a number of components normally made of alloyed grey cast-iron. However, these materials are not suitable for trim dies with sharp cutting edges for cutting off excessive material after the component has been shaped. For this purpose an alloyed tool steel is used, often cast.

Example of a chain of process within the automotive industry

1. Blank die to cut out blanks from coiled material.
2. Draw die to shape the blank.
3. Trim die to cut off excessive material.
4. Flange die to make the initial bend for flanges.
5. Cam flange die to bend the flanges further inside.



Material properties especially influencing machinability are:

- Surface hardness - to resist abrasive and adhesive wear
- High content of carbides - to resist abrasive wear
- Toughness/ductility - to resist chipping and breakage

Dies and moulds for hot work such as die casting and forging are used for manufacturing of for instance engine blocks. These dies and moulds are exposed to a number of demanding conditions of which the following are particularly critical for the machinability of the tool material.

- Hot hardness - to resist plastic deformation and erosion
- Temperature resistance - to resist softening at high temperature
- Ductility/toughness - to resist fatigue cracking
- Hot yield strength - to resist heat checking

Moulds for plastic materials include injection, compression, blow and extrusion moulds. Factors that influnence the machinability in a plastic mould steel are:

- Hardness
- Toughness/ductility
- Homogenity of microstructure and hardness

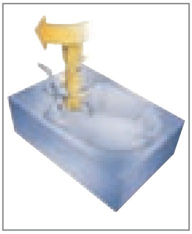
Die and mould material

The materials described and used as reference-material in this guide are mainly from the steel manufacturer Uddeholm, with a cross reference list at the end of the chapter.

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

Many different tool steels are used to produce dies and moulds. In forging and die casting the choice is generally hot-work tool steels that can withstand the relatively high working temperatures involved. Plastic moulds for thermoplastics and thermosets are sometimes made from cold-work tool steel. In addition, some stainless steels and grey cast iron are used for dies and moulds. Typical in-service hardness is in the range of 32 - 58 HRC for die and mould material.

CAST-IRON



Cast-iron is an iron-carbon alloy with a carbon content of mostly 2-4% as well as other elements like silicon, manganese, phosphorus and sulphur. Corrosion and heat resistance may be improved with additions of nickel, chromium, molybdenum and copper. Good rigidity, compressive strength and fluidity for cast iron are typical properties. Ductility and strength can be improved by various treatments, which affect the micro-structure. Cast-iron is specified, not by chemical analysis, but by the respective mechanical properties. This is partly due to that the cooling rate affects the cast-iron properties.

Carbon is presented as carbide-cementite and as free carbon-graphite. The extents of these forms depend partly on the amount of other elements in the alloy. For instance, a high-silicon cast-iron will be made up of graphite with hardly any cementite. This is the type known as grey iron. The silicone content usually varies between 1-3%. A low amount of silicone will stabilize carbides and the cast-iron will be made up dominantly of cementite with little graphite. This is a hard but weak brittle type called white iron.

In spite of the silicone content having a decisive influence on the structure, the cooling rate of cast iron in castings is also influential. Rapid cooling may not leave enough time for grey iron to form, as the silicone has not had time to decompose the cementite into the graphite. Varying sectional thicknesses in castings affect the cooling rate, affecting the state of carbon. Thick section will solidify into grey iron while thin ones will chill into white iron. Hence chilled cast-iron. Modern casting techniques control analysis, cooling rates, etc. to provide the cast-iron components with the right graphite structure. Also to provide chilled parts where needed, for instance a wear face on a component. Manganese strengthens and toughens cast-iron and is usually present in amounts of 0.5-1%.

For this reason, a thin or tapered section will tend to be more white iron because of the cooling effect in the mould. Also the surface skin of the casting is often harder, white iron while underneath is grey iron.

The basic structural consituents of the different types of cast-iron are ferritic, pearlitic or a mixture of these.

Types of cast-iron with ferritic matrix and little or no pearlite are easy to machine. They have low strength and normally a hardness of less than 150 Brinell. Because of the softness and high ductility of ferrite these types of cast-iron can be ”sticky” and result in built-up edge forming at low cutting data, but this can be avoided by increasing the cutting speed, if the operation permits.

Types of cast-iron with ferritic/pearlitic or pearlitic matrix range from about 150 HB with relatively low strength to high-strength, hard cast-irons of 280-300 HB where pearlitic matrix dominates.

Pearlite has a stronger, harder and less ductile structure than ferrite, its strength and hardness depending on whether it has rough or fine lamellar. The more fine-grained and more fine lamellar the pearlite, the higher strength and hardness. This means it has smaller carbides with less abrasive wear but is more toughness demanding due to smearing and built up edge formation.

Carbides are extremely hard constituents whether they are of pure cementite or contain alloying material. In thin plates, as in pearlite, cementite can be machined, but in larger particles which separate the constituents they drastically reduce the machinability. Carbides often occur in thin sections, projecting parts or corners of castings due to the rapid solidification, giving a finer structure, of these parts.

Hardness of cast iron-is often measured in Brinell. It is an indication of machinability, which deteriorates with increasing Brinell hardness. But the hardness value is an unreliable measurement of machinability when there are two factors that the value does not show.

In most machining operations it is the hard parts at the edges and corners of components which cause problems when machining. The Brinell test cannot be carried out on edges and corners and therefore the high hardness in these parts is not discovered before machining is undertaken.

A Brinell test says nothing about the cast-iron's abrasive hardness which is the difference between the hardness on the basic structure and the hardness of the constituent e.g. a particle of carbide.

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.

Alloy additives in cast-iron affect machinability in as much as they form or prevent the forming of carbides, affect strength and /or hardness. The structure within the cast-iron is affected by the alloying material which, depending on its individual character, can be divided into two groups.

1. Carbide forming: Chromium (Cr), cobalt (Co), manganese (Mr), vanadium (V).

2. Graphitizing elements: Silicone (Si), nickel (Ni), aluminium (Al), copper (Cu), titanium (Ti).

Grey cast-iron
There is a large range of grey cast-irons with varying tensile strengths. The silicon content/sectional area combinations form various structures of which the low-silicon, fine graphite and pearlite make the strongest and toughest material. Tensile strength varies considerably throughout the range. A coarse graphite structure means a weaker type. A typical cast-iron, where metal cutting is involved, often has a silicon content of around 2%. Common are the austenitic types.

Nodular cast-iron (SG)
The graphite is contained as round nodules. Magnesium especially is used to deposit the gobules and added to become a magnesium-nickel alloy. Tensile strength, toughness and ductility are considerably improved. Ferritic, pearlitic and martensitic types with various tensile strengths occur.

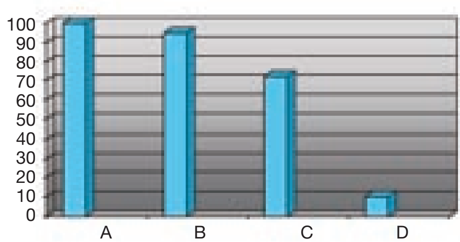
The SG cast-iron is also a graphite structure with properties in-between that of grey and nodular cast-iron. The graphite flakes are compacted into short ones with round ends through the addition of titanium and other treatment.

Malleable cast-iron
When white iron is heat treated in a particular way, ferritic, pearlitic or martensitic malleable cast-iron is formed. The heat treatments may turn the cementite into spherical carbon particles or remove the carbides. The cast-iron product is malleable, ductile and very strong. The silicon content is low. Three categories occur: ferritic, pearlitic and martensitic and they may also be categorized as Blackheart, Whiteheart and pearlitic.

Alloyed cast-iron
These are cast-irons containing larger amounts of alloying elements and, generally, these have similar effects on properties of cast-iron as they do on steel. Alloying elements are used to improve properties by affecting structures. Nickel, chromium, molybdenum, vanadium and copper are common ones. The graphite-free white cast-iron is extremely wear resistant while the graphite-containing cast-iron is also known as heat resistant ductile cast-iron. Corrosion resistance is also improved in some types. Toughness, hardness and heat resistance are typically improved.

The main difference in these types is the form in which carbon, mainly graphite occurs.

The general relative machinability of the four main kinds of cast-iron is indicated in a diagram where (A) is grey cast-iron, (B) malleable, (C) S.G. iron and (D) chilled, white cast-iron.



Relative machinability

Machinability of cast-iron

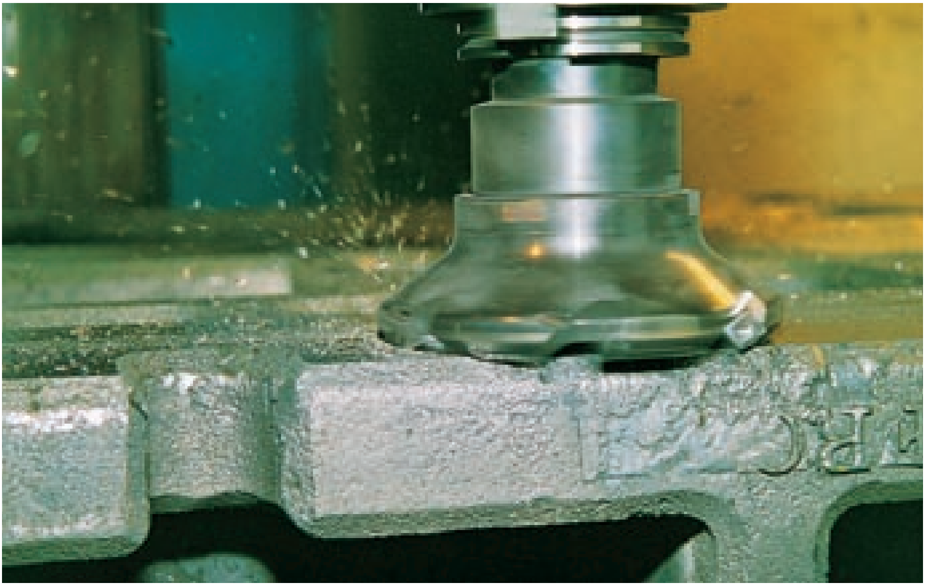
When establishing machinability characteristics of cast-iron grades, it is often useful to note the analysis and structure:

- Reduced carbon content results in lower machinability since less fracture-indicating graphite can be formed.
- Ferritic cast-iron with an increased silicon content is stronger and less ductile and tends to give less build-up edge.
- Increased pearlitic content in the matrix results in higher strength and hardness and decreased machinability.
- The more fine lamellar and fine-grained the pearlite is, the lower is the machinability.
- The presence of about 5% of free carbides in the matrix decreases machinability substantially.
- The effects of free carbides with respect to machinability is more negative in cast-iron with pearlitic matrix, because the pearlite ”anchors” the carbide particles in the matrix. This means that it is necessary for the insert edge to cut through the hardest particles instead of, as can be done with a ferritic structure, ”pulling” out or pushing into the soft ferrite.
- The top of the casting can have a somewhat lower machinability due to impurities such as slag, casting sand etc. which float up and concentrate in this surface area.

Generally it can be said that: the higher the hardness and strength that a type of cast-iron has, the lower is the machinability and the shorter the tool-life that can be expected from inserts and tools.

Machinability of most types of cast-iron involved in metal cutting production is generally good. The rating is highly related to the structure where the harder pearlitic cast-irons are somewhat more demanding to machine. Graphite flake cast-iron and malleable cast-iron have excellent machining properties while SG cast-iron is not quite as good.

The main wear types encountered when machining cast-iron are abrasion, adhesion and diffusion wear. The abrasion is produced mainly by the carbides, sand inclusions and harder chill skins. Adhesion wear with built-up edge formation takes place at lower machining temperatures and cutting speeds. This is the ferrite part of cast-iron which is most easily welded onto the insert but which can be counteracted by increasing speed and temperature. On the other hand, diffusion wear is temperature related and occurs at high cutting speeds, especially with the higher strength cast-iron grades. These grades have a greater deformation resistance, leading to higher temperature. This type of wear is related to the reaction between cast-iron and tool and has led to some cast-iron machining being carried out at high speeds with ceramic tools, achieving good surface finish.



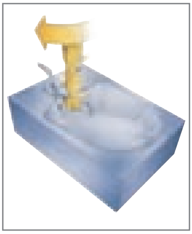
Typical tool properties needed, generally, to machine cast-iron are high hot-hardness and chemical stability but, depending upon the operation, workpieces and machining conditions, toughness, thermal shock resistance and strength are needed from the cutting edge. Ceramic grades are used to machine cast-iron along with cemented carbide.

Obtaining satisfactory results in machining cast-iron is dependent on how the cutting edge wear develops: rapid blunting will mean premature edge breakdown through thermal cracks and chipping and poor results by way of workpiece frittering, poor surface finish, excessive waviness, etc. Well developed flank wear, maintaining a balanced, sharp edge, is generally to be strived for.

Materiel type	Hardness HB	Area of use	CMC Coromant	Sweden SS	Germany DIN	USA ASTM	UK BS
Grey Cast-iron	150-200	Frames	08.1/2	0125	GG25	A48 Class 40B	BS1452 G150
Alloyed grey-iron	220-260	Dies	07.2	0852	GG26	-	G250 + Cr % Mo
Alloyed grey-iron	210-240	Dies	07.2	0852	GG26	-	BS1452 G250
Nodular cast-iron	200-260	Dies& stamps	09.1	0717-12	GGG60	-	BS2989 600/3
Nodular cast-iron	230-300	Dies& stamps	09.2	0732-3	GGG60	A536 Grade 80-55-06	BS2989 700/2

Materiel type	Hardness HB	Area of use	Japan JIS	France ANFOR	Italy UNI	Brazil
Grey Cast-iron	150-200	Frames	FC250 FC300 FC350	Ft25	G25	GG25
Alloyed grey-iron	220-260	Dies	Not available	Mn450	GNM45	GG26
Alloyed grey-iron	210-240	Dies	Not available	Mn450	GNM45	GG26
Nodular cast-iron	200-260	Dies& stamps	FCD450 FCD550	FGS400	GS370-17	GGG60
Nodular cast-iron	230-300	Dies& stamps	FCD600 FCD800	FGS600	GS600	GGG60

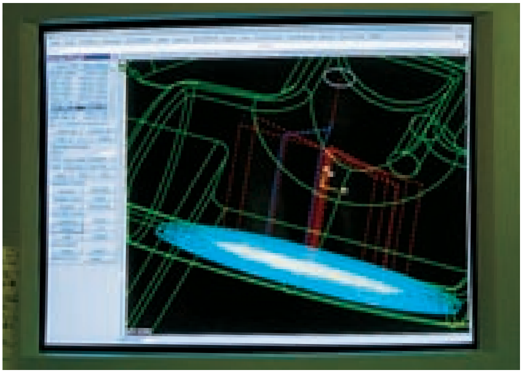
FROM QUOTATION TO A FINISHED PRESS TOOL



Finding good solutions with little material
First of all the die and mouldmaker has to do a quotation on the job, which can be hard many times since the blueprints from the customer often is pretty rough outlined due to their own ongoing development of the product.

Often the tool maker receive CAD drawings of the finished component, which looks far from the different tools that has to be manufactured to produce the component. This phenomenon has much to do with the integration of computers within the manufacturing and the companies ever shortened lead times on products.

There are often complicated shapes and geometries with deep cavities and radii, which has to be pressed to close tolerances. To be able to create these shapes several different press tools has to be manufactured. If one company can come up with a smart solution that has fewer steps in the pressing process they have a clear advantage.



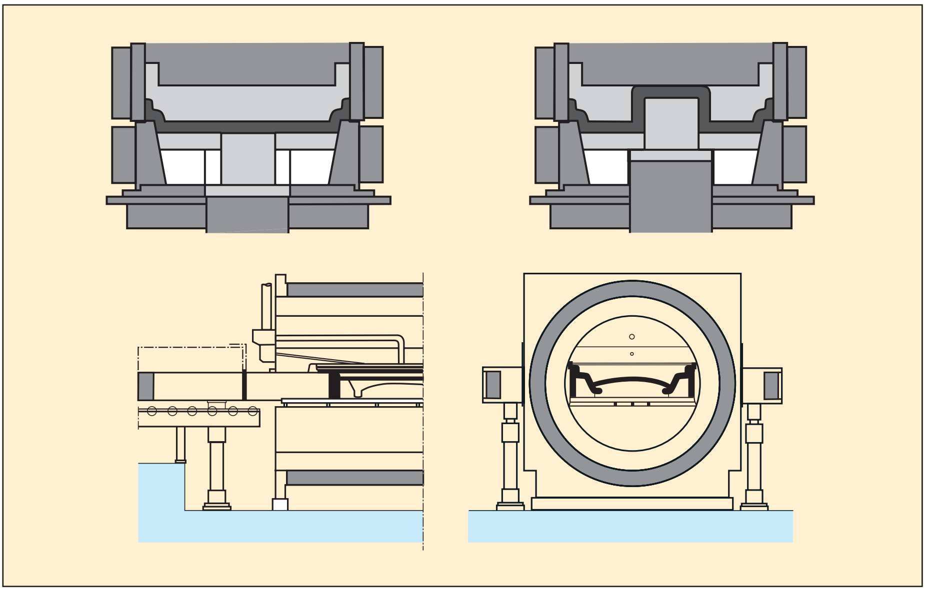
If the component to be machined is very large, a model of foamed plastic (styrofoam) is made with the shape of the component. The model of foamed plastic is then packed in sand and chill cast. When the melted metal is poured into the casting mould the foamed plastic evaporates and you will get a blank with an optimised shape to have as little material to remove as possible to the finished shape of the component. About 10 mm and sometimes even less stock is left to the final shape of the die, which saves a lot of time as much rough machining is eliminated.

A model of foamed plastic, which is close to the shape of the component to save time in rough machining

The next step is to start up the machining of the component. Usually this is done directly on the optimized blank of the tool. However, sometimes the customer or the tool manufacturer himself wants to make a prototype of the tool to see that everything is correct before starting to cut chips out of the optimized blank. Which is both expensive and can take a long time to produce.

The prototype is normally machined in aluminium or kirkzite. Only half the tool is machined, the lower part, and is then put up in a Quintus press. This type of press has a rubber stamp working as the upper part of the press tool, which press down on the sheet metal which forms after the prototype tool half. Instead of a rubber stamp there are also press techniques where liquid is used to press the sheet metal over the prototype tool. These procedures are often used within the automotive industry in order to produce several prototype components to crash test to see if any changes has to be made to the component. There are several advantages with this type of prototype methods when it is used in low volume part production:

- Only a single rigid tool half is required to form, trim and flange a part.
- Tool cost reductions of up to 90%.
- Reduction in project lead-time.
- Reduction in storage space for tools.
- Increased design and material possibilities.
- The single tool half can easily be modified to accommodate part design changes.
- No matching or fixing of tool-halves.
- Several different parts can be formed in one press cycle.
- Prototype tools can often be used in series production tools.

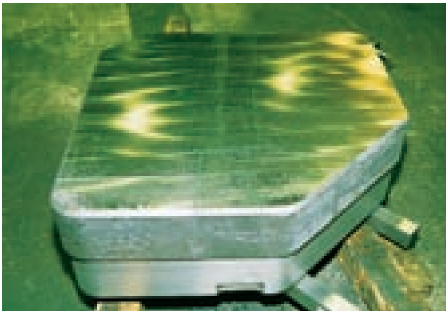
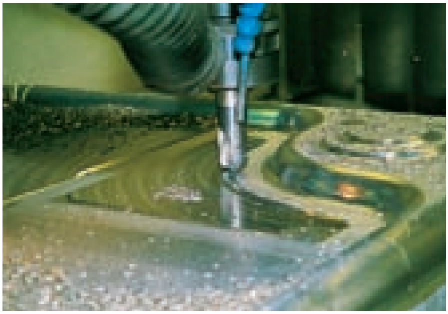


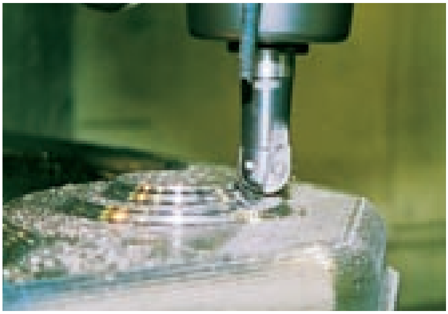
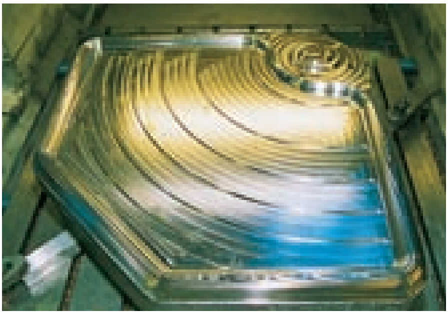
However, it is also very common that both halves of a tool is machined as the only difference is that it is made of aluminium. Which is easy to machine and cheaper than the real tool steel.

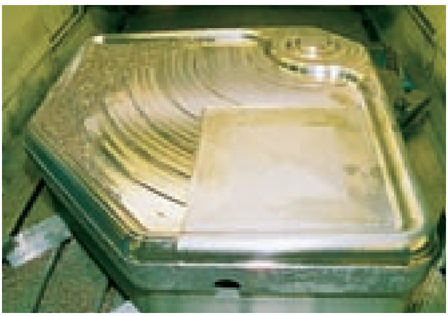
While the blank is being cast the machining strategy and the tool paths are being decided with the help of CAM equipment at the programming department. When the blank arrives the CNC-programme should be out by the machine tool in the workshop. In some work shops the machine tools are connected to a CAM work-station, which enables the machinist to make changes in the programme if he realises that there is too much material to remove in certain places or another tool might be more suitable.

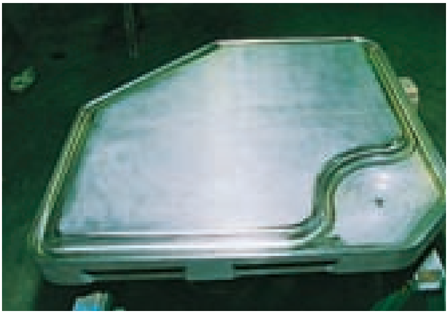
The machinist can, in fact, decide the whole milling strategy by the machine in a WOP-station (Workshop Oriented Programming).

The machining is often structured to perform the roughing and restmilling operations during the day shift, while attended by a machinist. The time consuming finishing operations are often done unmanned during the nights and week ends. When doing this it is important with a good monitoring system on the machine to prevent that the component gets damaged if a cutting tool breaks. If a good tool wear analysis has been made and the tool life has been established, automatic tool changes can be made to utilize the machine tool even further. However, this calls for very accurate tool settings especially in the Z-axis to get as small mismatch as possible.



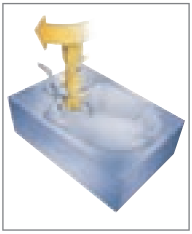
The die for a cutting tool after manual polishing

A pressed and Spotted Workpiece

When the machining is finished the die or mould has to be ground, stoned or polished manually, depending on the surface requirements. At this stage much time and money can be saved if more efforts and consideration has been put down on the previous machining operations.

When the press tool is thought to be finished the two halves must be fitted, trimmed, together. This is done by spotting, the surface of one of the halves is covered with ink, then a component is test pressed in the tool. If there is a clean spot somewhere on the sheet metal there might, for instance, be a radius which is wrong and need some additional polishing done to it. You also check that there is an even sheet metal column all over the test piece. This spotting-work is time consuming and if there is a tool, e. g. 3000 mm x 1500 mm and it shows that there is a corner 0.1 mm lower than everywhere else, the whole surface has to be ground and polished down 0.1 mm, which is a very extensive job.

PROCESS PLANNING



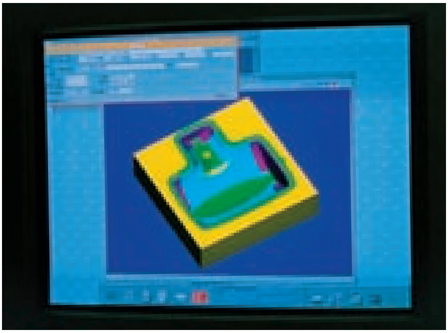
The larger the component and the more complicated the more important the process planning becomes. It is very important to have an open minded approach in terms of machining methods and cutting tools. In many cases it might be very valuable to have an external speaking partner who has experiences from many different application areas and can provide a different perspective and offer some new ideas.

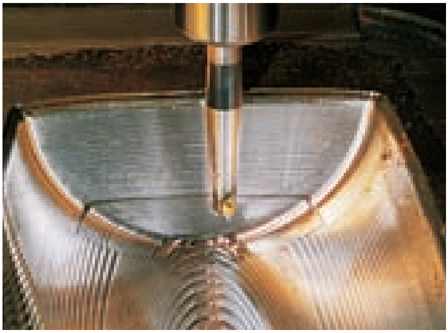
An open minded approach to the choice of methods, tool paths, milling and holding tools

In todays world it is a necessity to be competitive in order to survive. One of the main instruments or tools for this is computerised production. For the Die & Mould industry it is a question of investing in advanced production equipment and CAD/CAM systems.

But even if doing so it is of highest importance to use the CAM-softwares to their full potential.

In many cases the power of tradition in the programming work is very strong. The traditional and easiest way to program tool paths for a cavity is to use the old copy milling technique, with many entrances and exits into the material. This technique is actually linked to the old types of copy milling machines with their stylus that followed the model.





This often means that very versatile and powerful softwares, machine and cutting tools are used in a very limited way.

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

If instead using new ways of thinking and approaching an application, there will be a lot of wins and savings in the end.
If using a programming technique in which the main ingredients are to ”slice off” material with a constant Z-value, using contouring tool paths in combination with down milling the result will be:

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

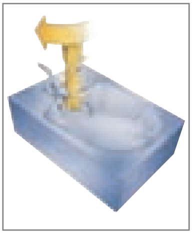
In combination with modern holding and cutting tools it has been proven many times that this concept can cut the total production cost considerably.

Initially a new and more detailed programming work is more difficult and usually takes somewhat longer time. The question that should be asked is, ”Where is the cost per hour highest? In the process planning department, at a workstation, or in the machine tool”?

The answer is quite clear as the machine cost per hour often is at least 2-3 times that of a workstation.

After getting familiar with the new way of thinking/programming the programming work will also become more of a routine and be done faster. If it still should take somewhat longer time than programming the copy milling tool paths, it will be made up, by far, in the following production. However, experience shows that in the long run, a more advanced and favourable programming of the tool paths can be done faster than with conventional programming.

THE RIGHT CHOICE OF HIGHLY PRODUCTIVE CUTTING TOOLS FOR ROUGHING TO FINISHING



First of all:

- Study the geometry of the die or mould carefully.

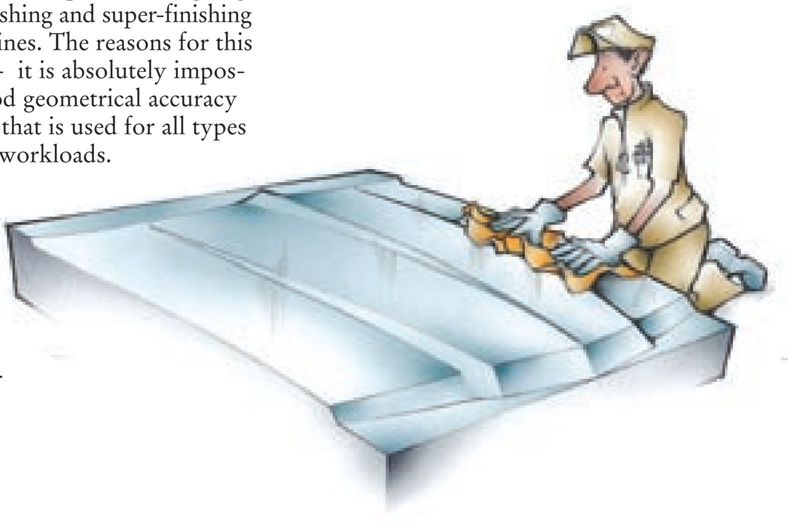
- Define minimum radii demands and maximum cavity depth.

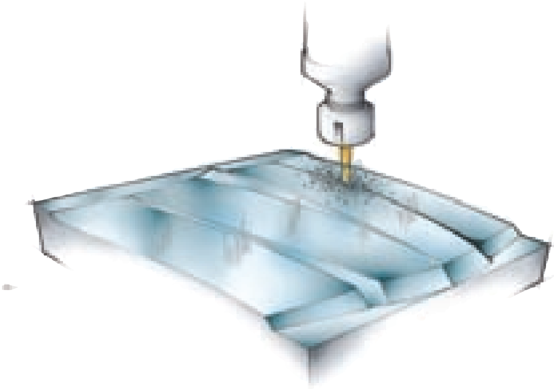
- Estimate roughly the amount of material to be removed. It is important to understand that roughing and semi-finishing of a big sized die or mould is performed far more efficiently and productively with conventional methods and tooling. Also for big sized dies and moulds. This is due to the fact that the material removal rate in HSM is much lower than in conventional machining. With exception for machining of aluminium and non-ferrous materials. However the finishing is always more productive with HSM.

- The preparation (milled and parallel surfaces) and the fixturing of the workpiece is of great importance. This is always one classic source for vibrations. If performing HSM this point is extra important. When performing HSM or also in conventional machining with high demands on geometrical accuracy of the die or mould, the strategy should always be to perform roughing, semi-finishing, finishing and super-finishing in dedicated machines. The reasons for this are quite obvious - it is absolutely impossible to keep a good geometrical accuracy on a machine tool that is used for all types of operations and workloads.

The guide ways, ball screws and spindle bearings will be exposed to bigger stresses and workloads when roughing for instance. This will of course have a big impact on the surface finish and geometrical accuracy of the dies or moulds that are being finish machined in that machine tool. It will result in a need of more manual polishing and longer try out times. And if remembering that today's target should be to reduce the manual polishing, then the strategy to use the same machine tools for roughing to finishing points in totally wrong direction. The normal time to manually polish, for instance, a tool for a large bonnet is roughly 350-400 hours.

If this time can be reduced by good machining it not only reduces the cost, but also enhance the geometrical accuracy of the tool. A machine tool machines pretty much exactly what it is programmed for and therefore the geometrical accuracy will be better the more the die or mold can be machined. However, when there is extensive manual finishing the geometrical accuracy will not be as good because of many factors such as how much pressure and the method of polishing a person uses to mention two of them.





If adding, totally, some 50 hours on advanced programming (minor part) and finishing in an accurate machine tool, the polishing can often be reduced down to 100-150 hours, or sometimes even less. There will also be other considerable benefits by machining to more accurate tolerances and surface structure/finish. One is that the improved geometrical accuracy gives less try out times. Which means shorter lead times. Another is that, for instance, a pressing tool will get a longer tool life and that the competitiveness will increase via higher component quality. Which is of highest importance in today's competition.

A human being can not compete, no matter how skilled, with a computerised tool path when it comes to precision. Different persons use different pressures when doing stoning and polishing, resulting most often in too big dimensional deviations. It is also difficult to find and recruit skilled, experienced labour in this field. If talking about HSM applications it is absolutely possible, with an advanced and adapted programming strategy, dedicated machine tools and holding and cutting tools, to eliminate manual polishing even up to 100%. If using the strategy to do roughing and finishing in separate machines it can be a good solution to use fixturing plates. The die or mould can then be located in an accurate way. If doing 5-sided machining it is often necessary to use fixturing plates with clamping from beneath. Both the plate and the blank must be located with cylindrical guide pins.

The machining process should be divided into at least three operation types; roughing, semi-finishing and finishing, some times even super-finishing (mostly HSM applications). Restmilling operations are of course included in semi-finishing and finishing operations.

Each of these operations should be performed with dedicated and optimised cutting tool types.

In conventional die & mould making it generally means:

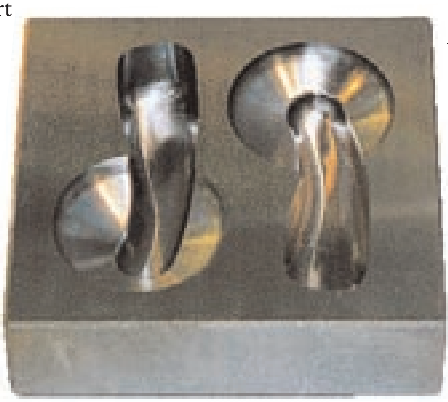
Roughing: Round insert cutters, end mills with big corner radii

Semi-finishing: Round insert cutters, toroid cutters, ball nose endmills

Finishing: Round insert cutters (where possible), toroid cutters, ball nose endmills (mainly)

Restmilling: Ball nose endmills, endmills, toroid and round insert cutters

In high speed machining applications it may look the same. Especially for bigger sized dies or moulds. In smaller sizes, max 400 X 400 X 100 (l,w,h), and in hardened tool steel, ball nose end mills (mainly solid carbide) are usually first choice for all operations. But, it is definitely possible to compete in productivity also by using inserted tools with specific properties. Such as round insert cutters, toroid cutters and ball nose end mills. Each case has to be individually analysed...



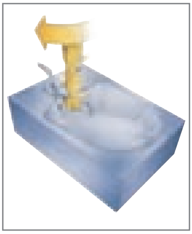
To reach maximum productivity it is also important to adapt the size of the milling cutters and the inserts to a certain die or mould and to each specific operation. The main target is to create an evenly distributed working allowance (stock) for each tool and in each operation. This means that it is most often more favourable to use different diameters on cutters, from bigger to smaller, especially in roughing and semi-finishing. Instead of using only one diameter throughout each operation. The ambition should always be to come as close as possible to the final shape of the die or mould in each operation.

An evenly distributed stock for each tool will also guarantee a constant and high productivity. The cutting speed and feed rate will be on constant high levels when the a_e/a_p is constant. There will be less mechanical variations and work load on the cutting edge. Which in turn gives less heat generation, fatigue and an improved tool life.

A constant stock also enables for higher cutting speed and feed together with a very secure cutting process. Some semi-finishing operations and practically all finishing operations can be performed unmanned or partially manned. A constant stock is of course also one of the real basic criterias for HSM.

Another positive effect of a constant stock is that the impact on the machine tool - guide ways, ball screws and spindle bearings will be less negative. It is also and always, very important to adapt the size and type of milling cutters to the size of the machine tool.

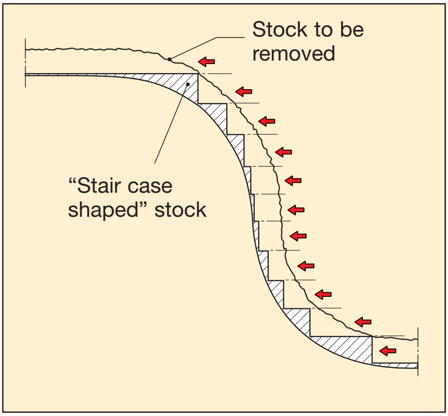
THE VERSATILITY OF ROUND INSERT CUTTERS

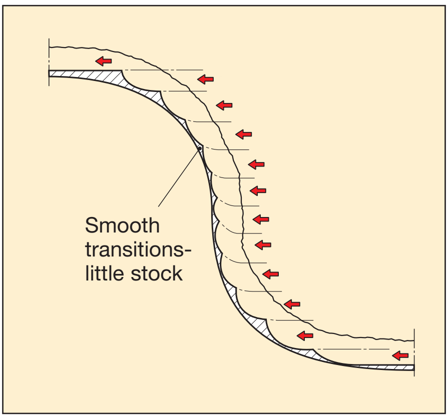


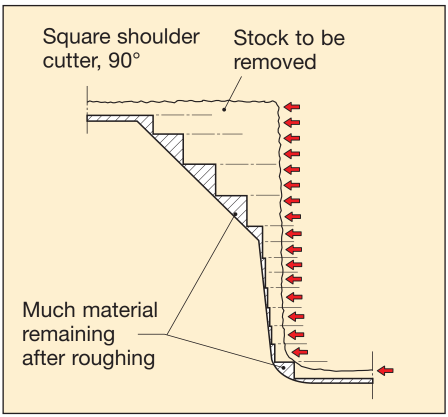
If the rough milling of a cavity is done with a square shoulder cutter much stair-case shaped stock has to be removed in semi-finishing. This of course creates varying cutting forces and tool deflection. The result is an uneven stock for finishing, which will influence the geometrical accuracy of the die or mould.

If a square shoulder cutter with triangular inserts is used it will have relatively weak corner cross sections, creating an unpredictable machining behaviour. Triangular or rhombic inserts also creates big radial cutting forces and due to the number of cutting edges they are less economical alternatives in these operations.

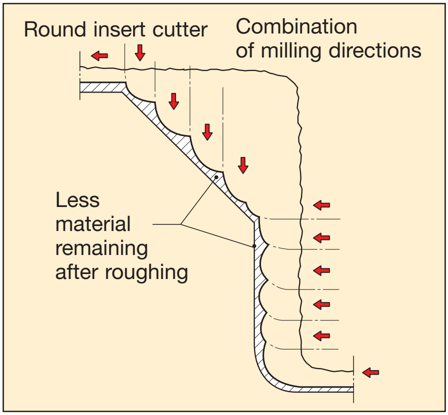
On the other hand if round inserts, which allows milling in all materials and in all directions, are used this will give smooth transitions between the passes and also leaves less and more even stock for the semi-finishing. Resulting in a better die or mould quality. Among the features of round inserts is that they create a variable chip thickness. This allows for higher feed rates compared with most other insert shapes. The cutting action of round inserts is also very smooth as the entering angle successively alters from nearly zero (very shallow cuts) to 90 degrees. At maximum depth of cut the entering angle is 45 degrees and when copying with the periphery the angle is 90 degrees. This also explains the strength of round inserts - the work-load is built up successively.

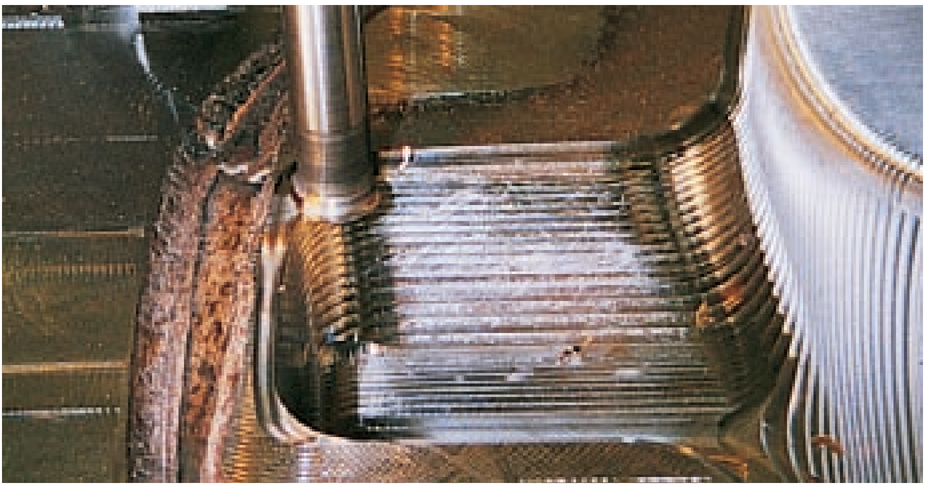






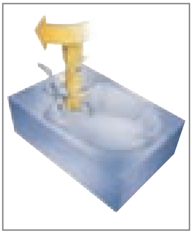
Round inserts should always be regarded as first choice for roughing and medium roughing operations. In 5-axis machining round inserts fit in very well and have practically no limitations. With good programming round insert cutters and toroid cutters can replace ball nose end mills to a very big extent. The productivity increase most often ranges between 5-10 times (compared with ball nose end mills). Round insert cutters with small run-outs can in combination with ground, positive and light cutting geometries also be used in semi-finishing and some finishing operations.





SANDVIK Coromant

APPLICATION TECHNOLOGY

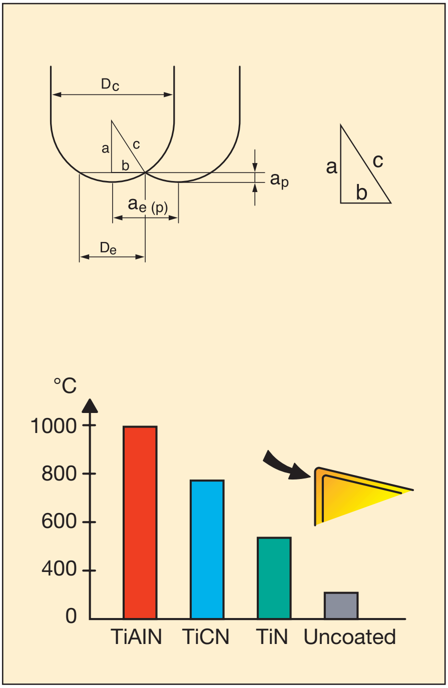


This is very much a question about optimising cutting data, grades and geometries in relation to the specific type of material, operation and productivity and security demands.

It is always important to base calculations of effective cutting speed on the true or effective diameter in cut. If not doing this, there will be severe miscalculations of the feed rate as it is dependent on the rpm for a certain cutting speed.

If using the nominal diameter value of the tool, when calculating cutting speed, the effective or true cutting speed will be much lower if the depth of cut is shallow. This is valid for tools such as, round insert cutters (especially in the small diameter range), ball nose end mills and end mills with big corner radii. The feed rate will of course also be much lower and the productivity severely hampered.

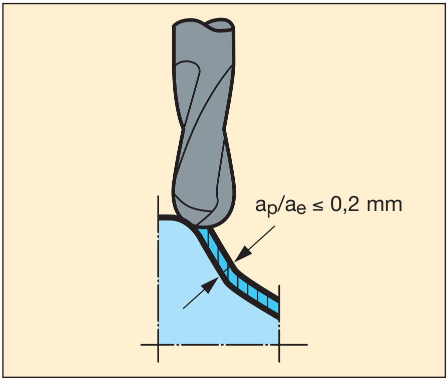
Most important is that the cutting conditions for the tool will be much under its capacity and recommended application range. Often this leads to premature frittering and chipping of the cutting edge due to too low cutting speed and heat in the cutting zone.



When doing finishing or super-finishing with high cutting speed in hardened tool steel it is important to choose tools that have a coating with high hot hardness. Such as TiAlN, for instance.

One main parameter to observe when finishing or super-finishing in hardened tool steel with 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 holding/ cutting tool and to keep a high tolerance level and geometrical accuracy on the machined die or mould.

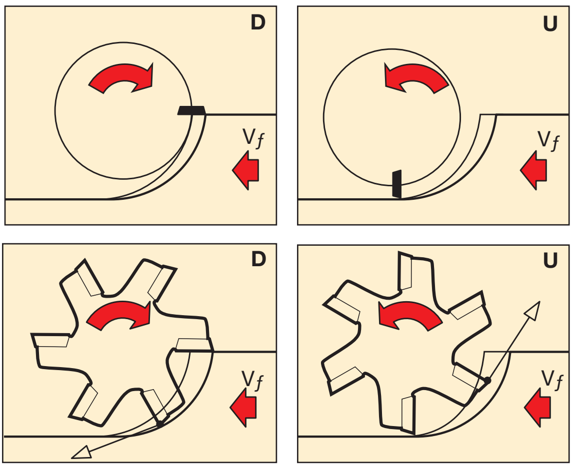
Choose very stiff holding and cutting tools. When using solid carbide it is important to use tools with a maximum core diameter (big bending stiffness).



When using inserted ball nose end mills, for instance, it is favourable to use tools with shanks made of heavy metal (big bending stiffness). Especially if the ratio overhang/diameter is large.

Another application parameter of importance is to try to use down milling tool paths as much as possible. It is, nearly always, more favourable to do down milling than up milling. When the cutting edge goes into cut in down milling the chip thickness has its maximum value. And in up milling, it has its minimum value. The tool life is generally shorter in up milling than in down-milling due to the fact that there is considerably more heat generated in up-, than in down milling. When the chip thickness in up milling increases from zero to maximum the excessive heat is generated as the cutting edge is exposed to a higher friction than in down milling. The radial forces are also considerably higher in up milling, which affects the spindle bearings negatively.

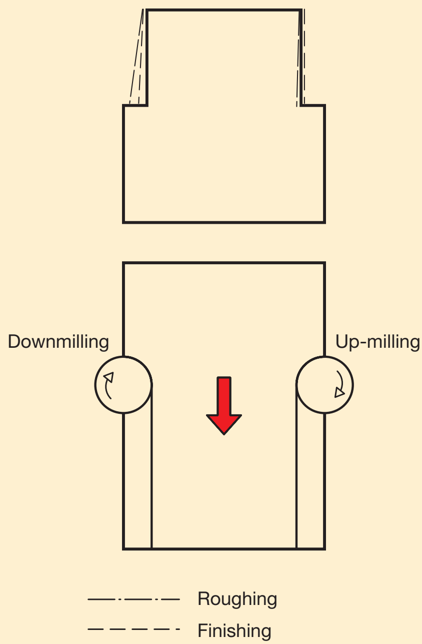
In down milling the cutting edge is mainly exposed to compressive stresses, which are much more favourable for the properties of cemented or solid carbide compared with the tensile stresses developed in up milling.



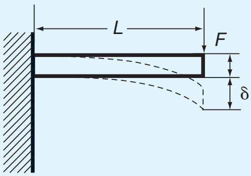
When doing side milling (finishing) with solid carbide, especially in hardened materials, up milling is first choice. It is then easier to get a better tolerance on the straightness of the wall and also a better 90 degree corner. The mismatch between different axial passes will also be less, if none. This is mainly due to the direction of the cutting forces. If having a very sharp cutting edge being in cut the cutting forces tend to ”pull” or ”suck” the cutter towards the material.

Up milling can be favourable when having old manual milling machines with large play in the lead screw, because a ”counter pressure” is created which stabilizes the machining.

The best way to ensure down milling tool paths in cavity milling is to use contouring type of tool paths. Contouring with the periphery of the milling cutter (for instance a ball nose end mill) often results in a higher productivity, due to more teeth effectively in cut on a larger tool diameter. If the spindle speed is limited in the machine, contouring will help keeping up the cutting speed. This type of tool paths also creates less quick changes in work load and direction. This is of specific importance in HSM applications and hardened materials as the cutting speed and feed are high and the cutting edge and process is more vulnerable to any changes that can create differences in deflection and create vibrations. And ultimately total tool breakdown.



Bending	Roughing	Finishing
Upmilling	- 0.02 mm	0.00 mm
Downmilling	0.06 mm	0.05 mm



L = overhang; d = diameter ; F = radial force
E, π = constants; k = all constants etc. put together

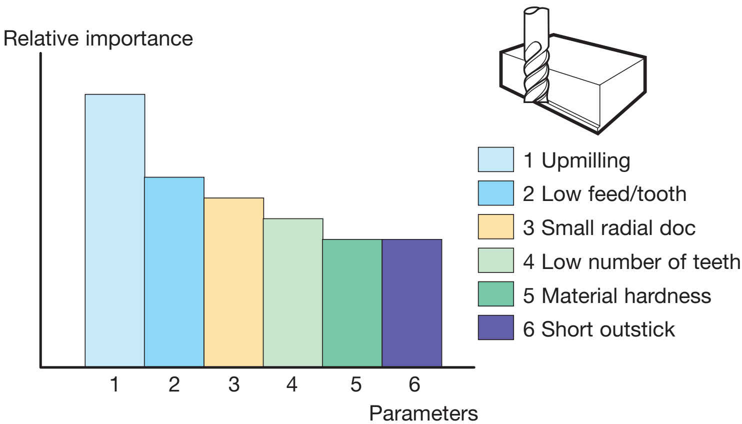
$$\delta = \frac{F \times L^3}{3 \times E \times I}$$

$$I = \frac{\pi \times d^4}{64}$$

$$\Rightarrow \delta = k \times \frac{F \times L^3}{d^4} \qquad k = \frac{64}{3 \times \pi \times E}$$

⇒ 20 % Overhang reduction reduce tool deflection by 50 %
⇒ 20 % Increased D (Ø10 ⇒ Ø12mm) Reduce Tool Deflection By 50

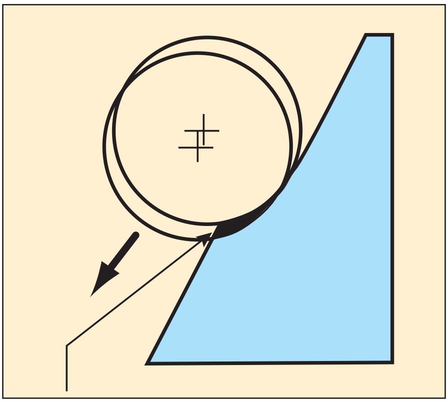
Solid Carbide Endmills - Finishing/Deflection



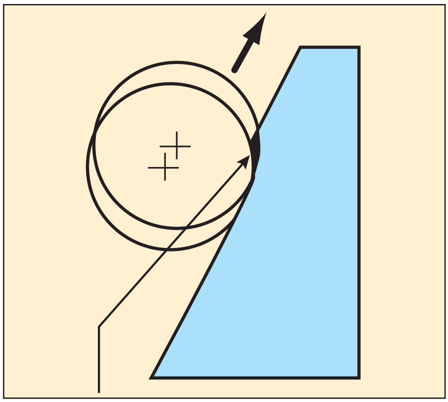
Endmills with a higher helix angle have less radial forces and usually run smoother. Endmills with a higher helix angle has more axial forces and the risk of being pulled out from the collet is greater.

Copy milling and plunging operations along steep walls should be avoided as much as possible! When plunging, the chip thickness is large at a low cutting speed. Risk for frittering of the centre. Especially when the cutter hits the bottom area. If the control has no, or a poor, look ahead function the deceleration will not be fast enough and there will most likely be damages on the centre.

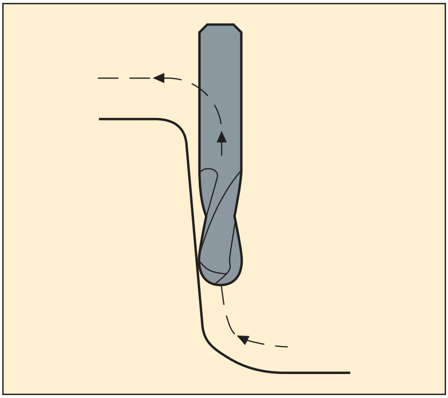
It is somewhat better for the cutting process to do up-copying along steep walls as the chip thickness has its maximum at a more favourable cutting speed.



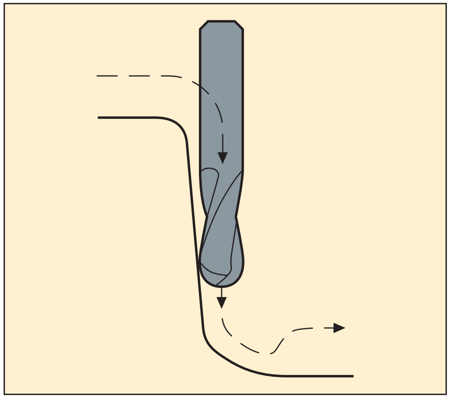
Large chip thickness at very low v_c



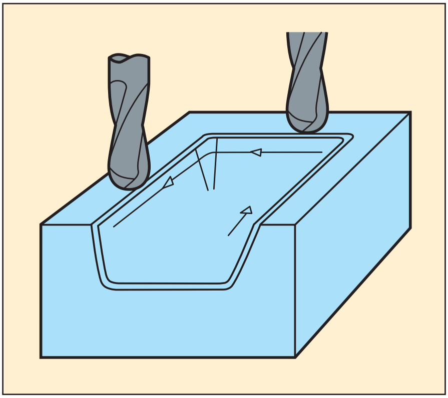
Max chip thickness att recommended v_c

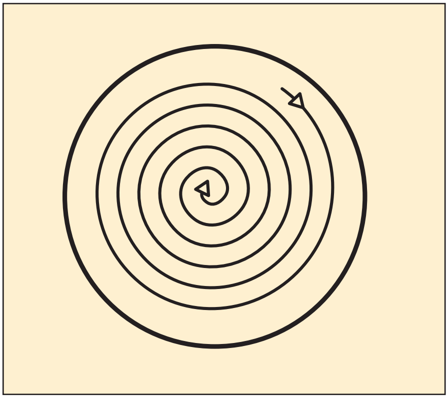


But, there will be a big contact length when the cutter hits the wall, with risk for vibration, deflection or even tool breakage if the feed speed does not decelerate fast enough. There is also a risk of pulling out the cutter from the holder due to the direction of the cutting forces.



The most critical area when using ball nose end mills is the centre portion. Here the cutting speed is zero, which is very disadvantageous for the cutting process. Chip evacuation in the centre is also more critical due to the small space at the chisel edge. Avoid using the centre portion of a ball nose end mill as much as possible. Tilt the spindle or the workpiece 10 to 15 degrees to get ideal cutting conditions. Sometimes this also gives the possibility to use shorter (and other type of) tools.





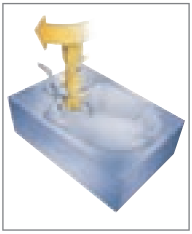
For a good tool life it is also more favourable in a milling process to stay in cut continuously and as long as possible. All milling operations have an interrupted or intermittent character due to the usage of multi-teeth tools.

The tool life will be considerably shorter if the tool has many entrances and exits in the material. Which adds the amount of thermal stresses and fatigue in the cutting edge. It is more favourable for modern cemented carbide to have an even and high temperature in the cutting zone than to have large fluctuations.

Usage of coolant also adds temperature differences and is in general harmful for milling operations. This will be treated more in detail in another chapter.

Copy milling tool paths are often a mix of up-, and down milling (zig-zag) and gives a lot of engagements and disengagements in cut. This is, as mentioned above, not favourable for any milling cutter, but also harmful for the quality of the die or mould. Each entrance means that the tool will deflect and there will be an elevated mark on the surface. This is also valid when the tool exits. Then the cutting forces and the bending of the tool will decrease and there will be a slight undercutting of material in the exit portion. These factors also speak for contouring and down milling tool paths as the preferred choice.

SCULPTURED SURFACES

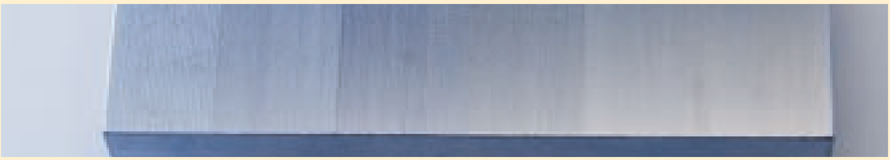


In finishing and super-finishing, especially in HSM applications, the target is to reach a good geometrical and dimensional accuracy and reduce or even eliminate all manual polishing.

In many cases it is favourable to choose the feed per tooth, fz, identical with the radial depth of cut, a_e ($f_z = a_e$).

This gives following advantages:

- very smooth surface finish in all directions
- very competitive, short machining time
- very easy to polish this symmetrical surface texture, self detecting character via peaks and valleys
- increased accuracy and bearing resistance on surface gives longer tool life on die or mould
- minimum cusp or scallop height decides values on f_z/a_e/R

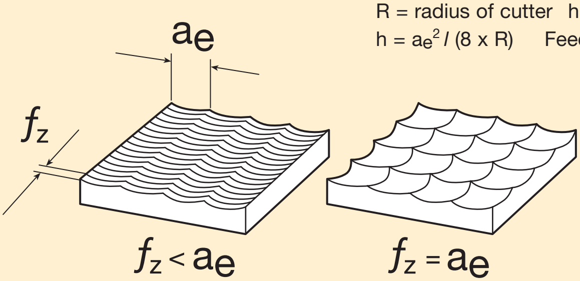


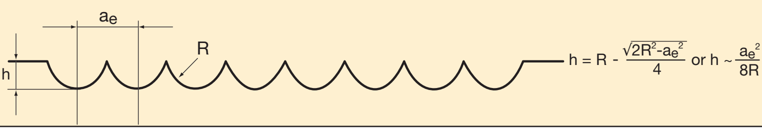
Spinle speed n = 12000 rpm, cutter diameter = 6 mm

f_z/a_e	0.05	0.075	0.1	0.15	0.2	0.25
Feed v_f	F1200	F1800	F2400	F3600	F4800	F6000
Cusp/h	h0.0001	h0.0002	h0.0004	h0.0009	h0.002	h0.003
Time (min)	10	4.44	2.50	1.11	0.62	0.40

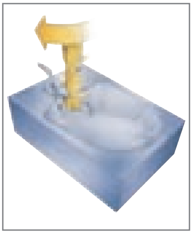
R = radius of cutter h = cusp height

$h = a_e^2 / (8 \times R)$ Feed = Rpm x f_z x z

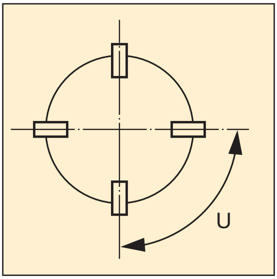




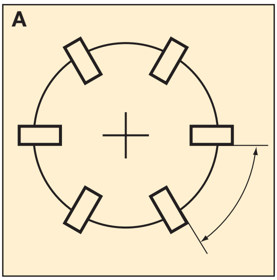
PITCH



A milling cutter, being a multi-edge tool, can have a variable number of teeth (z) and there are certain factors that help to determine the number for the type of operation. The material and size of workpiece, stability, finish and the power available are the more machine orientated factors while the tool related include sufficient feed per tooth, at least two cutting edges engaged in cut simultaneously and that the chip capacity of the tool is ample.

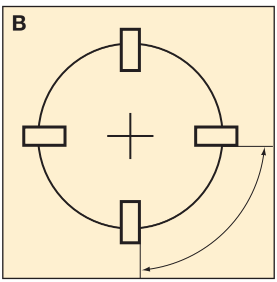


The pitch (u) of a milling cutter is the distance between a point on the edge to the same point on the next edge. Milling cutters are classified into coarse, close or extra-close pitch cutters and most cutters have these three options.



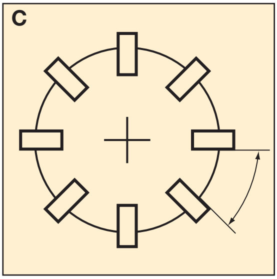
(A) Close pitch means more teeth and moderate chip pockets and permit high metal-removal rate. Normally used for cast-iron and for medium duty machining operations in steel

Close pitch is the first choice for general purpose milling and is recommended for mixed production.



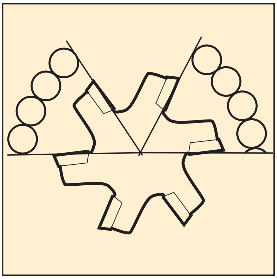
(B) Coarse pitch means fewer teeth on the cutter periphery and large chip pockets. Coarse pitch is often used for roughing to finishing of steel and where vibration tendencies are a threat to the result of the operation.

Coarse pitch is the true problem solver and is the first choice for milling with long overhang, low powered machines or other applications where cutting forces must be minimized.



(C) Extra-close pitch cutters have small chip pockets and permits very high table feeds. These cutters are suitable for machining interrupted cast-iron surfaces, roughing cast-iron and small depth of cut in steel. Also in materials where the cutting speed has to be kept low, for instance in titanium.

Extra close pitch is the first choice for cast iron.



The milling cutters can have either even or differential pitch. The latter means unequal spacing of teeth round the cutter and is a very effective means of coming to terms with problems of vibrations.

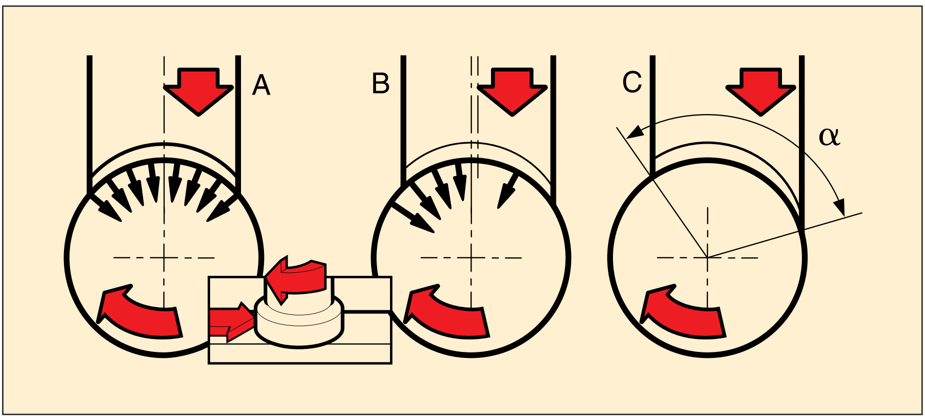
When there is a problem with vibration it is recommended that a milling cutter with as coarse pitch as possible is used, so that fewer inserts give less opportunities for vibration to arise. You can also remove every second insert in the milling cutter so that there are fewer inserts in cut. In full slot milling you can take out so many of the inserts that only two remain. However, this means that the cutter being used must have an even number of teeth, 4, 6, 8, 10 etc. With only two inserts in the milling cutter, the feed can be increased and the depth of cut can usually be increased several times. The surface finish will also be very good. A surface finish of Ra 0.24 in hardened steel with a hardness of 300 HB has been measured after machining with a milling cutter with an overhang of 500 mm. In order to protect the insert seats, the inserts sitting in the seats which are not being in cut can be ground down and allowed to remain in the cutter as dummy inserts.

Positioning and length of cut

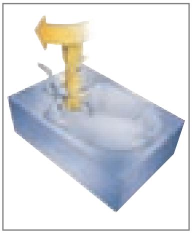
The length of cut is affected by the positioning of the milling cutter. Tool-life is often related to the length of cut which the cutting edge must undergo. A milling cutter which is positioned in the centre of the workpiece gives a shorter length of cut, while the arc which is in cut will be longer if the cutter is moved away from the centre line (B) in either direction.

Bearing in mind how the cutting forces act, a compromise must be reached. The direction of the radial cutting forces (A) will vary when the insert edges go into and out of cut and play in the machine spindle can give rise to vibration and lead to insert breakage.

By moving the milling cutter off the centre, B and C, a more constant and favourable direction of the cutting forces will be obtained. With the cutter positioned close to the centre line the largest average chip thickness is obtained. With a large facemill it can be advantageous to move it more off centre. In general, when facemilling, the cutter diameter should be 20-25% larger than the cutting width.



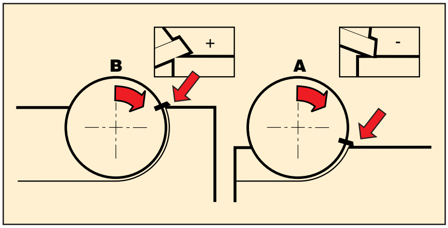
ENTRANCE AND EXIT OF CUT



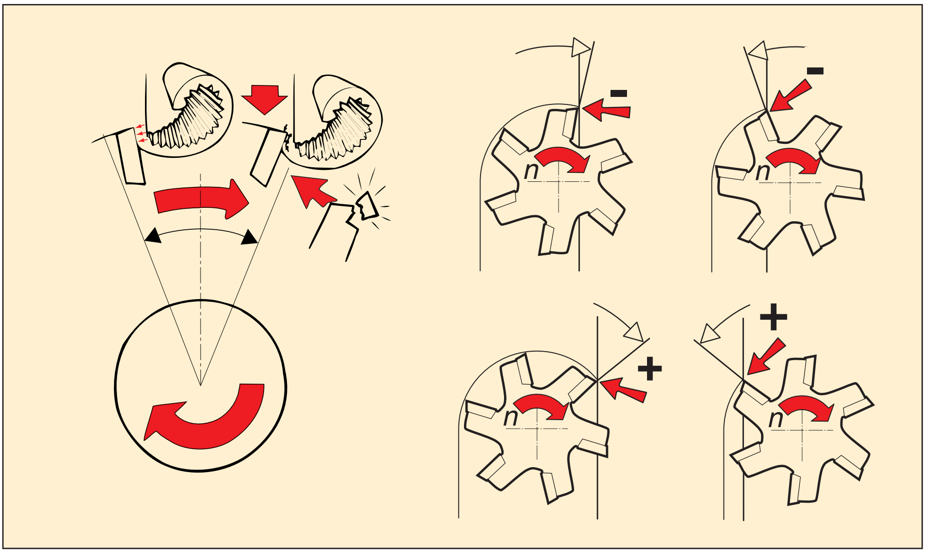
Every time a cutter goes into cut, the inserts are subjected to a large or small shock load depending on material, chip cross section and the type of cut. The initial contact between the cutting edge and workpiece may be very unfavourable depending on where the edge of the insert has to take the first shock. Because of the wide variety of possible types of cut, only the effects of the cutter position on the cut will be considered here.

Where the centre of the cutter is positioned outside the workpiece (A) an unfavourable contact between the edge of the insert and the workpiece results.

Where the centre of the cutter is positioned inside the workpiece (B) the most favourable type of cut results.



The most dangerous situation however, is when the insert goes out of cut leaving the contact with the workpiece. The cemented carbide inserts are made to withstand compressive stresses which occur every time an insert goes into cut (down milling). On the other hand, when an insert leaves the workpiece when hard in cut (up milling) it will be affected by tensile stresses which are destructive for the insert which has low strength against this type of stress. The result will often end in rapid insert failure.



The basic action to take when there is a problem with vibration is to reduce the cutting forces. This can be done by using the correct tools and cutting data.

Choose milling cutters with a coarse and differential pitch.

Choose positive insert geometries.

Use as small milling cutter as possible. This is particularly important when milling with tuned adaptors.

Small edge rounding (ER). Go from a thick coating to a thin one, if necessary use uncoated inserts.

Use a large feed per tooth, reduce the rotational speed and maintain the table feed (= larger feed/tooth). Or maintain the rotational speed and increase the table feed (and feed/tooth). Do not reduce the feed per tooth.

Reduce the radial and axial cutting depths.

Choose a stable tool holder such as Coromant Capto. Use the largest possible adapter size to achieve the best stability. Use tapered extensions for the best rigidity.

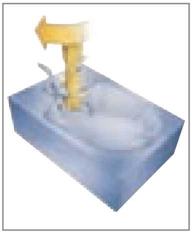
With long overhangs, use tuned adaptors in combination with coarse and differential pitch milling cutters. Position the milling cutter as close to the tuned adapter as possible.

Position the milling cutter off centre of the workpiece, which leads to a more favourable direction of the cutting forces.

Start with normal feed and cutting speeds. If vibration arises try introducing these measures gradually, as previously described:

a) increase the feed and keep the same rpm
b) decrease the rpm and keep the same feed
c) reduce the axial or/and radial depth of cut
d) try to reposition the cutter

RAMPING AND CIRCULAR INTERPOLATION



Axial feed capability is an advantage in many operations. Holes, cavities as well as contours can be efficiently machined. Facemilling cutters with round inserts are strong and have big clearance to the cutter body.

Those lend themselves to drill/mill operations of various kinds. Ramping at high feed rates and the ability to reach far into workpieces make round insert cutters a good tool for complicated forms. For instance, profile milling in five-axis machines and roughing in three-axis machines.

Ramping is an efficient way to approach the workpiece when machining pockets and for larger holes circular interpolation is much more power efficient and flexible than using a large boring tool. Problems with chip control is often eliminated as well. When ramping, the operation should be started around the centre, machining outwards in the cavity to facilitate chip evacuation and clearance. As milling cutters has limitations in the axial depth of cut and varies depending on the diamater, the ramping angle for different sizes of cutters should be checked.

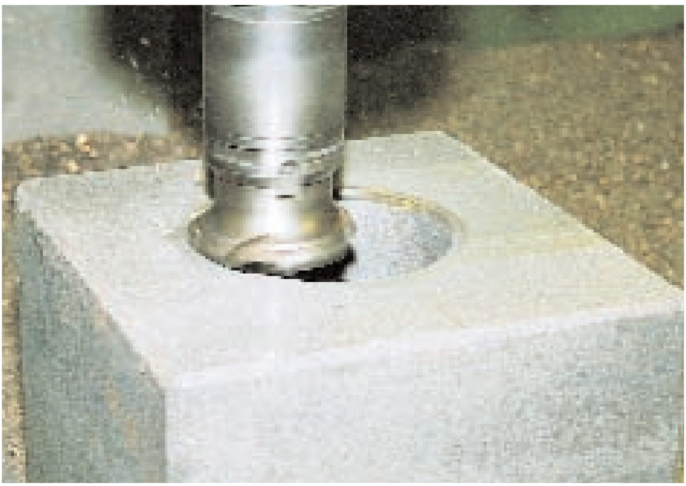
The ramping angle is dependent upon the diameter of the cutters used, clearance to the cutter body, insert size and depth of

cut. A 32 mm CoroMill 200 cutter with 12 mm inserts and a cutting depth of 6 mm can ramp at an angle of 13 degrees. Whilst an 80 mm cutter manages 3.5 degrees. The amount of clearance also depends upon the diameter of the cutter.

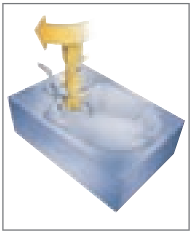
Often used within die & mould making is when the tool is fed in a spiral shaped path in the axial direction of the spindle, while the workpiece is fixed. This is most common when boring and have several advantages when machining holes with large diameters. First of all the large diameter can be machined with one and the same tool, secondly chip breaking and evacuation is usually not a problem when machining this way, much because of the smaller diameter of the tool compared to the diameter of the hole to be machined and third, the risk of vibration is small.

It is recommended that the diameter of the hole to be machined is twice the diameter of the cutter. Remember to check maximum ramping angle for the cutter when using circular interpolation as well.

These methods are favourable for weak machine spindles and when using long overhangs, since the cutting forces are mainly in the axial direction.



CHOICE OF HOLDING TOOLS



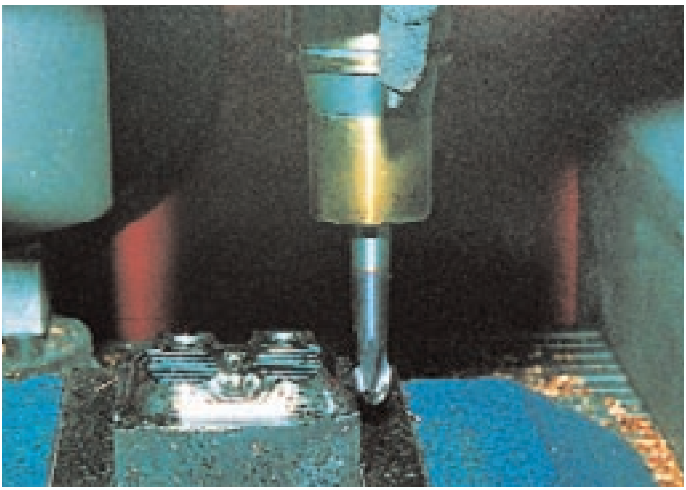
One of the main criteria when choosing both holding and cutting tools is to have as small run-out as possible. The smaller the run-out is, the more even the workload will be on each insert in a milling cutter. (Zero run-out would of course theoretically give the best tool life and the best surface texture and finish).

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!

Balancing adds some steps to the process and typically involves:

- Measuring the unbalance of a tool/ toolholder assembly.
- Reducing the unbalance by altering the tool, machining it to remove mass or by moving counterweights on toolholders.
- Often the procedure has to be repeated, involving checking the tool again, refining the previous adjustments until the balance target is achieved.

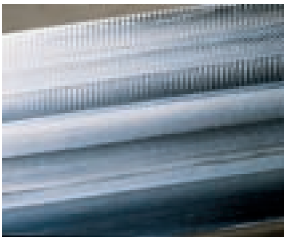
Tool balancing leaves several sources of process instability untouched. One of these is error in the fit between toolholder and spindle interface. The reason is that there is often a measurable play in this clamp, and there may also be a chip or dirt inside the taper. The taper will not likely line up the same way every time. The presence of any such contamination would create unbalance even if the tool, toolholder, and spindle were perfect in every other way. To balance tools is an additional cost to the machining process and it should be analysed in each case if cost reduction gained by balancing is viable. Sometimes, however, there is no alternative to get the required quality.



The aluminium workpiece in the picture illustrates tool balancing affecting surface finish. The balanceable toolholder used to machine both halves of the surface was set to two unbalance values, 100 gmm and 1.4 gmm. The more balanced tool produced the smoother surface. Conditions of the two cuts were otherwise identical: 12000 rpm, 5486 mm/min feed rate, 3.5 mm depth- and 19 mm width of cut, using a toolholder with a combined mass of 1.49 kg.

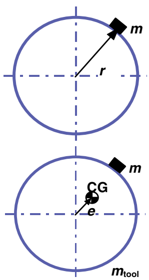
Balancing tools to G-class targets, as defined by ISO 1940-1, may demand holding the force from unbalance far less than the cutting force the machine will see anyway. In reality, an endmill run at 20000 rpm may not need to be balanced to any better than 20 gmm, and 5 gmm is generally appropriate for much higher speeds. The diagram refers to unbalance force relating to tool and adaptor weight of 1 kg. Field A shows the approximate cutting force on a 10 mm diameter solid endmill.

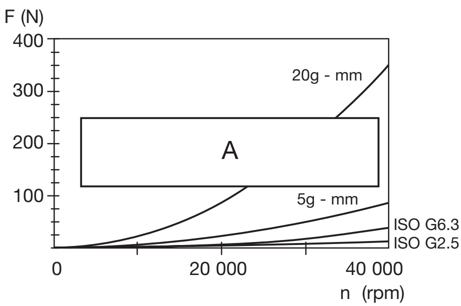
The balance equations contain:
F: force from unbalance (Newton)
G: G-class value, which has units of mm/sec
m: tool mass in kg
n: spindle speed in rpm
u: unbalance in gmm



$$u = m \times \frac{9549 \times G}{n} \ \text{(gmm)}$$

$$F = u \left(\frac{n}{9549}\right)^2 \ \text{(N)}$$





However, much can be done by just aiming for good balance through proper tool selection, and here are some points to think of when selecting tools:

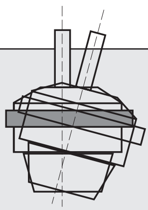
- Buy quality tools and toolholders. Look for toolholders that have been premachined to remove unbalance.
- Favour tools that are short and as lightweight as possible.
- Regularly inspect tools and toolholders for fatigue cracks and signs of distortion.

The tool unbalance that the process can accept is determined by aspects of the process itself. These include the cutting forces in the cut, the balance condition of the machine, and the extent to which these two affect each other. Trial and error is the best way to find the unbalance target. Run the intended operation several times to a variety of different values, for instance from 20 gmm and below. After each run, upgrade to a more balanced tool and repeat. The optimal balance is the point beyond which further improvements in tool balance fail to improve the accuracy or surface finish of the workpiece, or the point in which the process can easily hold the specific workpiece tolerances.

The key is to stay focused on the process and not aim for a G value or other arbitrary balance target. The aim should be to achieve the most effective process as possible. This involves weighing the costs of the tool balancing and the benefit it can deliver, and strike the right balance between them.

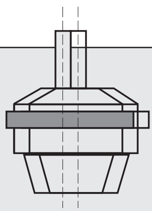
At high speed, the centrifugal force might be strong enough to make the spindle bore grow slightly. This has a negative effect on some V-flange tools which contact the spindle bore only in the radial plane. Spindle growth can cause the tool to be

Angular error



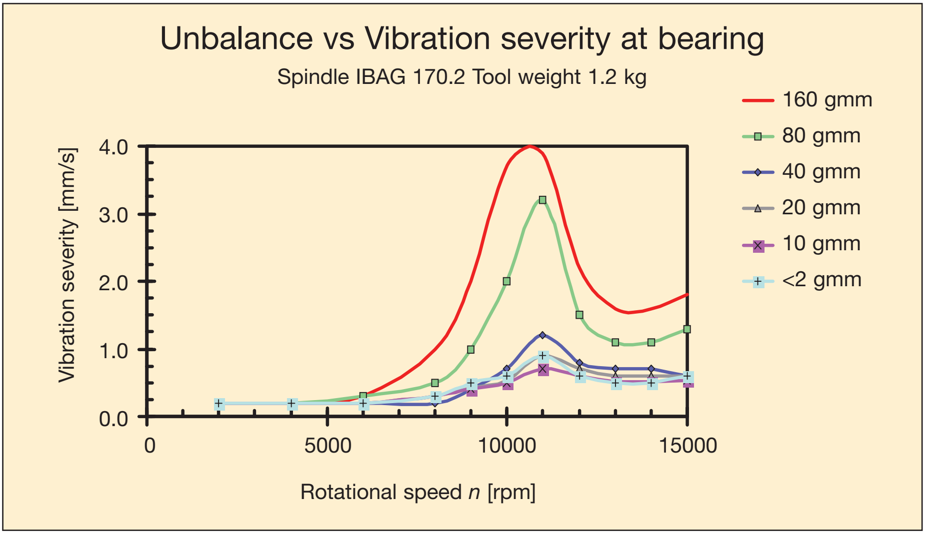
	Coromant Capto C5	HSK 50 A
Unbalance	Up to 0.9 gmm	up to 3.3 ggm
Balance class	up to G1.5	up to G5.6
TIR	up to 3.5 µm	up to 13.4 µm

Parallel error



	Coromant Capto C5	HSK 50 from A
Unbalance	Up to 2.6 gmm	up to 9.6 ggm
Balance class	up to G4.4	up to G516.8
TIR	up to 4.2 µm	up to 16 µm

***n* = 20 000 rpm, weight of adapter and toll *m* = 1.2 kg**

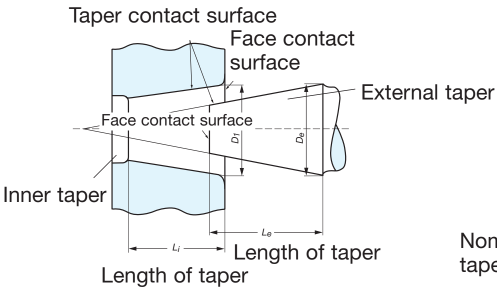


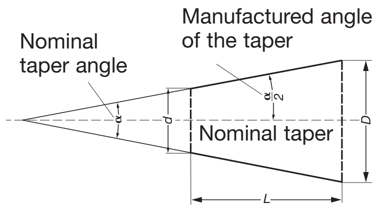
drawn up into the spindle by the constant pull of the drawbar. This can lead to a stuck tool or dimensional inaccuracy in the Z-axis.

Tools with contact both in the spindle bore and face, radial and axial contact, simultaneous fit tools are more suited for machining at high speeds. When the spindle begins to grow, the face contact prevents the tool from moving up the bore. Tools with hollow shank design are also susceptible to centrifugal force but they are designed to grow with the spindle bore at high speeds. The tool/spindle contact in both radial and axial direction also gives a rigid tool clamping enabling aggressive machining. The Coromant Capto coupling, due to its polygon design, is superior when it comes to high torque and productive machining.

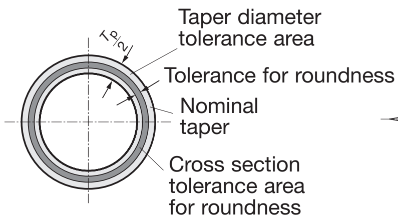
Surface contact of spindle interface at high spindle speed

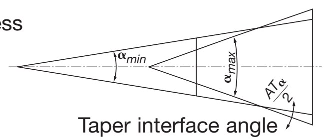
Spindle speed	ISO40	HSK 50A	Coromant Capto C5
0	100%	100%	100%
20000	100%	95%	100%
25000	37%	91%	99%
30000	31%	83%	95%
35000	26%	72%	91%
40000	26%	67%	84%





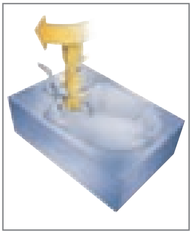
The roundness and concentricity are the most crucial factors for toolholders and not the tolerance class (AT).





When planning for HSM one should strive to build tools using a holder cutter combination that is symmetrical. There are some different tool systems which can be used. However, a shrink fit system where the toolholder is heated up and the bore expands and then clamps the tool when cooling down is considered to be one of the best and most reliable for HSM. First because it provides very low run-out, secondly the coupling can transmit a high torque, thirdly it is easy to build customized tools and tool assemblies and finally it gives high total stiffness in the assembly.

EXTENDED TOOLS IN ROUGHING OF A CAVITY



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 the productivity will be very low.

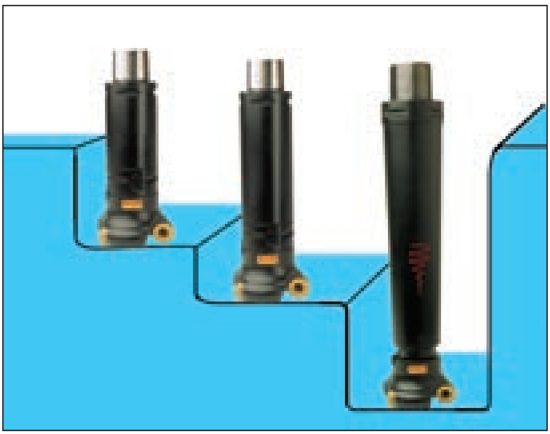
It is recommended to change to extended tools at pre-determined positions in the program. The geometry of the die or mould decides where to change.

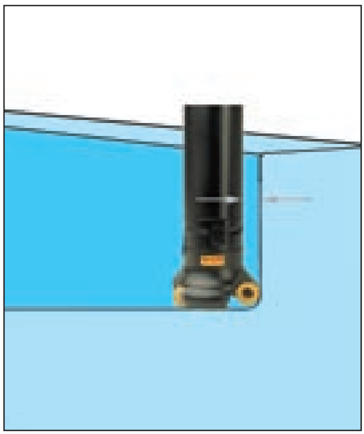
Cutting data should also be adapted to each tool length to keep up maximum productivity.

When the total tool length, from the gauge line to the lowest point on the cutting edge, exceeds 4-5 times the diameter at the gauge line, tuned, tapered bars should be used. Or, if the bending stiffness must be radically increased, extensions made of heavy metal should be used.

When using extended tools it is important to choose biggest possible diameter on the extensions and adaptors relatively to the cutter diameter. Every millimeter is important for maximum rigidity, stiffness and productivity. It is not necessary to have more than 1 mm radially in difference between holding and cutting tool. The easiest way to achieve this is to use oversized cutters.

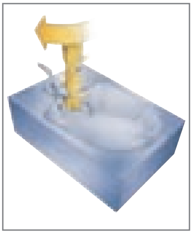
Modular tools increases the flexibility and the number of tool combination possibilities.





SANDVIK Coromant

MACHINING IN SEGMENTS



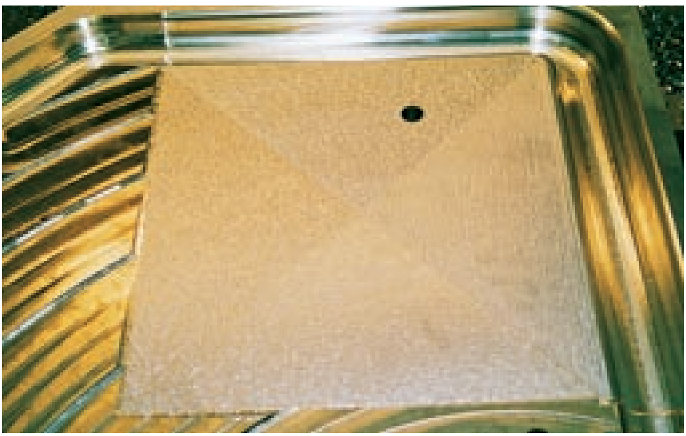
When machining huge press dies it is often necessary to index the inserts several times. Instead of doing this manually and interrupting the cutting process, this can be done in an organised way if precautions are taken in the process planning and programming.

Based on experience, or other information, the amount of material, or the surface to machine, can be split up in portions or segments. The segments can be chosen according to natural boundaries or be based on certain radii sizes in the die or mould. What is important is that each segment can be machined with one set of insert edges or solid carbide edges, plus a safety margin, before being changed to next tool in that specific family of replacement tools.

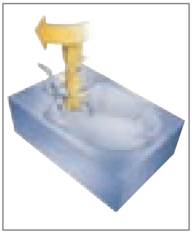
This technique enables full usage of the ATC (Automatic Tool Changer) and replacement tools (sister tools).

The technique can be used for roughing to finishing. Today's touch probes or laser measuring equipment gives very precise measuring of tool diameter and length and a matching (of surfaces) lower than 10 microns. It also gives several benefits such as:

- **Better machine tool utilisation-** less interruptions, less manual tool changing
- **Higher productivity-**easier to optimise cutting data
- **Better cost efficiency**-optimisation vs real machine tool cost per hour
- **Higher die or mould geometrical accuracy-**the finishing tools can be changed before getting excessive wear



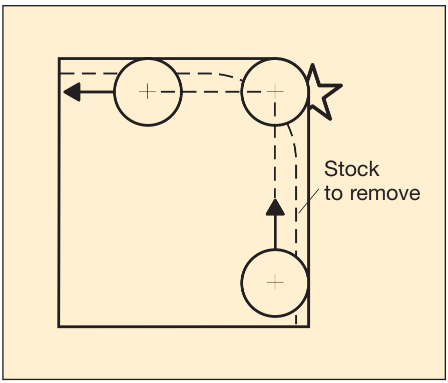
METHODS FOR MACHINING OF CORNERS

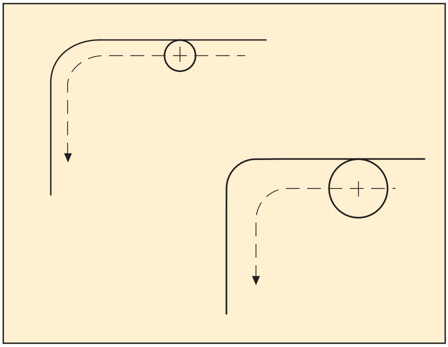


The traditional way of machining a corner is to use linear movements (G1) with non-continuous transitions in the corner. Which means that when the cutter comes to the corner it has to be slowed down because of dynamic limitations of the linear axes. And there will even be a very short stop before the motors can change the feed direction. As the spindle speed is the same, the situation creates a lot of excessive friction and heat. If for instance aluminium or other light alloys are machined they can get burning marks or even start to burn due to this heat. The surface finish will deteriorate optically and in some materials even structurally, even beyond the tolerance demands.

In traditional machining of corners the tool radius is identical with the corner radius. Which gives maximum contact length and deflection (often one quadrant).

The most typical result is vibrations, the bigger the longer the tool, or total tool overhang is. The wobbling cutting forces often also creates undercutting of the corner.

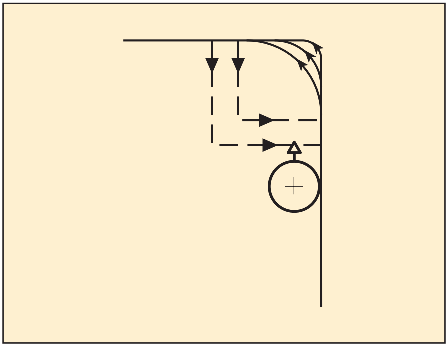


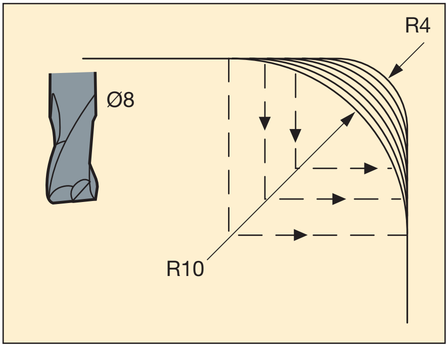


There is of course also a risk for frittering of edges or total tool break down.
Some solutions on this problem are:

- Use a cutter with a smaller radius to produce the desired corner radius on the die or mould. Use circular interpolation (G2, G3) to produce the corner. This movement type does not create any definite stop at block borders. Which means that the movement gives smooth continuous transitions and there is only a small chance that a vibration should start.

- Another solution is to produce a bigger corner radius, via circular interpolation, than stated in the drawing. This can be favourable sometimes as it allowes to use a bigger cutter diameter in roughing to keep up maximum productivity.

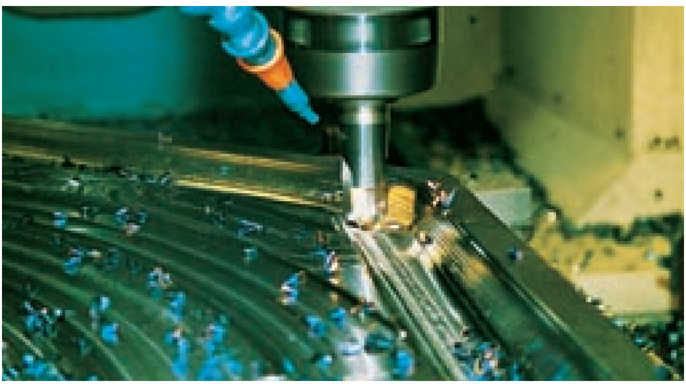




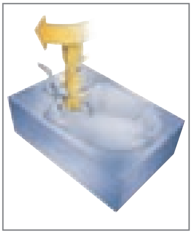
• The remaining stock in the corner can then be machined via restmilling (rest = remaining stock) with a smaller cutter radius and circular interpolation. The rest-milling of corners can also be performed by axial milling. It is important to use a good programming technique with a smooth approach and exit. It is very important to perform the restmilling of corners before or as a semi-finishing operation - gives even stock and high productivity in finishing. If the cavity is deep (long overhang) the ap/ae should be kept low to avoid deflection and vibration (ap/ae appr. 0,1-0,2 mm in HSM applications in hardened tool steel).

If consequently using a programming technique based on circular interpolation (or NURBS-interpolation), which gives both continuos tool paths and commands of feed and speed rates, it is possible to drive the mechanic functions of a machine tool to much higher speeds, accelerations and decelerations.

This can result in productivity gains ranging between 20-50%!



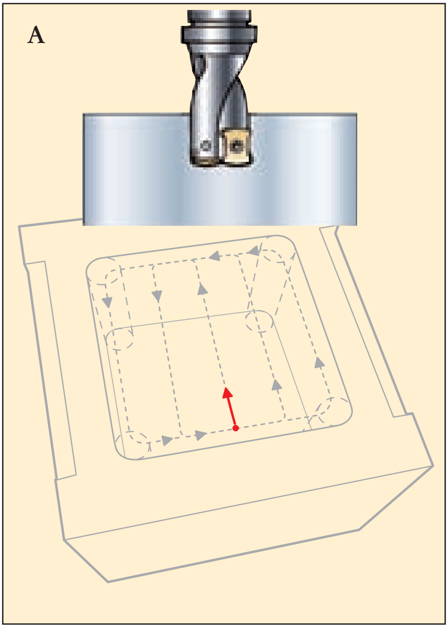
METHODS FOR MACHINING OF A CAVITY

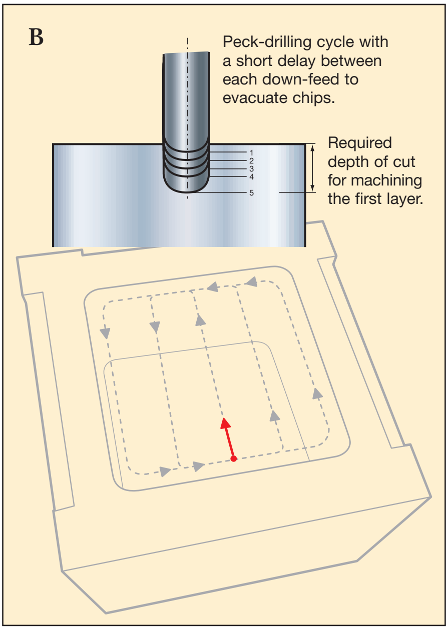


A. Pre-drilling of a starting hole. Corners can be pre-drilled as well. Not recommendable method as one extra tool is needed. Which also adds more unproductive positioning and tool changing time. The extra tool also blocks one position in the tool magazine. From a cutting point of view the variations in cutting forces and temperature when the cutter breaks through the pre-drilled holes in the corners is negative. The re-cutting of chips also increases when using pre-drilled holes.

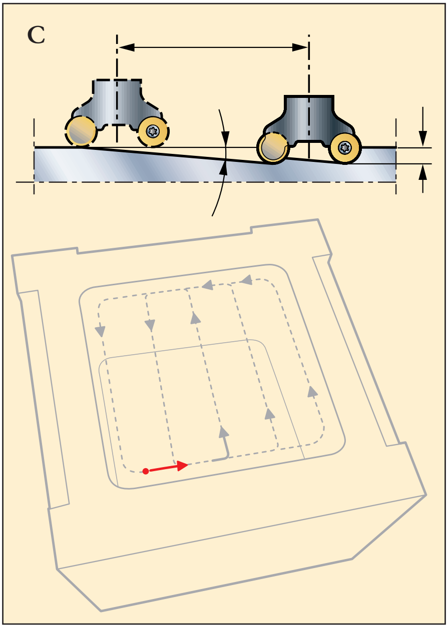
B. If using a ball nose end mill, inserted or solid carbide, it is common to use a peck-drilling cycle to reach a full axial depth of cut and then mill the first layer of the cavity. This is then repeated until the cavity is finished. The drawback with this start is chip evacuation problems in the centre of the end mill. Better than using a peck-drilling cycle is to reach the full axial depth of cut via circular interpolation in helix. Important also then to help evacuate the chips.

C. One of the best methods is to do linear ramping in X/Y and Z to reach a full axial depth of cut. Note that if choosing the right starting point, there will be no need of milling away stock from the ramping part. The ramping can start from in to out or from out to in depending on the geometry of the die or mould. The main criteria is how to get rid of the chips in the best way. Down milling should be practised in

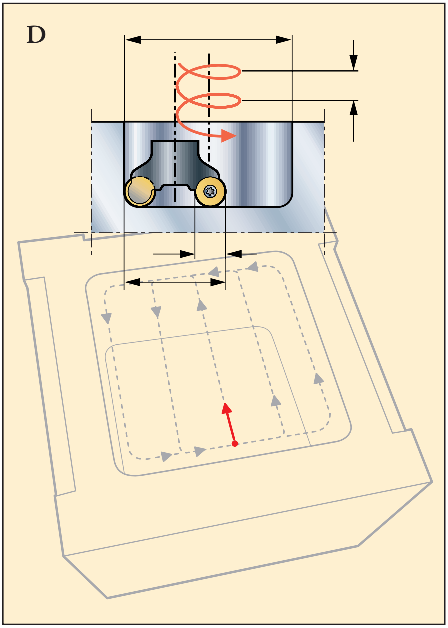




SANDVIK Coromant

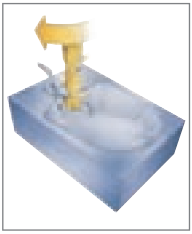


a continuous cutting. When taking a new radial depth of cut it is important to approach with a ramping movement or, better, with a smooth circular interpolation. In HSM applications this is crucial.



D. If using round insert cutters or end mills with a ramping capacity the most favourable method is to take the first axial depth of cut via circular interpolation in helix and follow the advice given in the previous point.

HSM - HIGH SPEED MACHINING



There are a lot of questions about HSM today and many different, more or less complicated, definitions can be seen frequently. Here the matter will be discussed in an easy fashion and from a practical point of view.

Historical background

The term High Speed Machining (HSM) commonly refers to end milling at high rotational speeds and high surface feeds. For instance, the routing of pockets in aluminum airframe sections with a very high material removal rate. Over the past 60 years, HSM has been applied to a wide range of metallic and non-metallic workpiece materials, including the production of components with specific surface topography requirements and machining of materials with a hardness of 50 HRC and above. With most steel components hardened to approximately 32-42 HRC, machining options currently include:

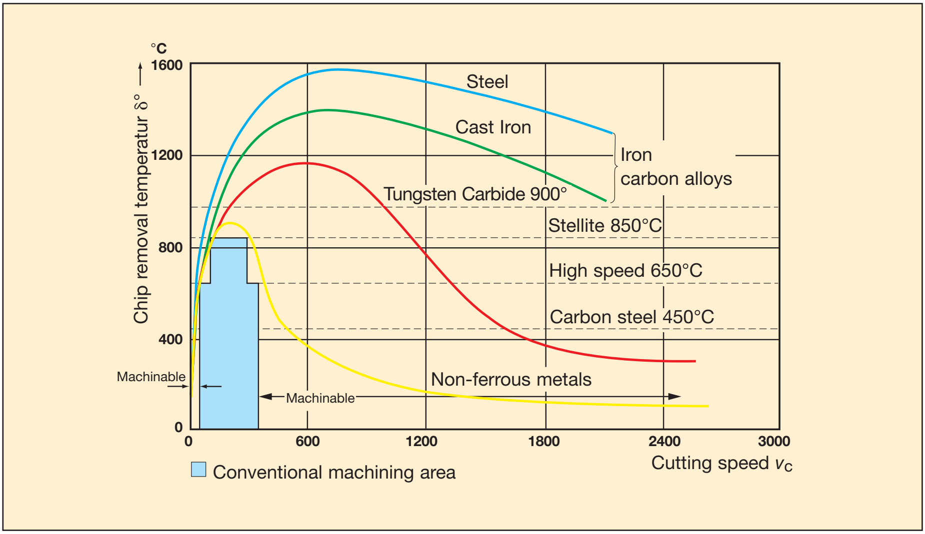
- rough machining and semi-finishing of the material in its soft (annealed) condition
- heat treatment to achieve the final required hardness (</= 63 HRC)
- machining of electrodes and Electrical Discharge Machining (EDM) of specific parts of the dies or moulds (specifically small radii and deep cavities with limited accessibility for metal cutting tools)
- finishing and super-finishing of cylindrical/flat/cavity surfaces with appropriate cemented carbide, cermet, solid carbide, mixed ceramic or polycrystalline cubic boron nitride (PCBN)

With many components, the production process involves a combination of these options and in the case of dies and moulds it also includes time consuming hand finishing. Consequently, production costs can be high and lead times excessive.

Typical for the die and mould industry is to produce one or a few tools of the same design. The process includes constant changes of the design. And because of the need of design changes there is also a corresponding need of measuring and reverse engineering.

The main criteria is the quality of the die or mould regarding dimensional, geometrical and surface accuracy. If the quality level after machining is poor and if it can not meet the requirements there will be a varying need of manual finishing work. This work gives a satisfying surface accuracy, but it always has a negative impact on the dimensional and geometrical accuracy.

One of the main targets for the die and mould industry has been, and is, to reduce or eliminate the need of manual polishing and thus improve the quality, shorten the production costs and lead times.



Chip removal temperature as a result of the cutting speed

Main economical and technical factors for the development of HSM

Survival - the ever increasing competition on the marketplace is setting new standards all the time. The demands on time and cost efficiency is getting higher and higher. This has forced the development of new processes and production techniques to take place. HSM provides hope and solutions...

Materials - the development of new, more difficult to machine materials has underlined the necessity to find new machining solutions. The aerospace industry has its heat resistant and stainless steel alloys. The automotive industry has different bimetal compositions, Compact Graphite Iron and an ever increasing volume of aluminum. The die and mold industry mainly has to face the problem to machine high hardened tool steels. From roughing to finishing.

Quality - the demand on higher component or product quality is a result of the hard competition. HSM offers, if applied correctly, solutions in this area. Substitution of manual finishing is one example. Especially important on dies or moulds or components with a complex 3D geometry.

Processes - the demands on shorter through - put times via fewer set-ups and simplified flows (logistics) can be solved to a big extent via HSM. A typical target within the die and mould industry is to make a complete machining of fully hardened small sized tools in one set-up. Costly and time consuming EDM-processes can also be reduced or eliminated via HSM.

Design & development - one of the main tools in today's competition is to sell products on the value of novelty. The average product life cycle on cars is today 4 years, computers and accessories 1,5 years, hand phones 3 months... One of the prerequisites of this development of fast design changes and rapid product development time is the HSM technique.

Complex products - there is an increase of multifunctional surfaces on components. Such as new design of turbine blades giving new and optimised functions and features. Earlier design allowed polishing by hand or with robots (manipulators). The turbine blades with the new, more sophisticated design has to be finished via machining and preferably by HSM. There are also more and more examples of thin walled workpieces that has to be machined (medical equipment, electronics, defence products, computer parts).

Production equipment - the strong development of cutting materials, holding tools, machine tools, controls and especially CAD/CAM features and equipment has opened possibilities that must be met with new production methods and techniques.

The original definition of HSM

Salomons theory, "Machining with high cutting speeds..." on which he got a German patent 1931, assumes that "at a certain cutting speed (5-10 times higher than in conventional machining), the chip removal temperature at the cutting edge will start to decrease...".

Giving the conclusion: ..."seem to give a chance to improve productivity in machining with conventional tools at high cutting speeds..."

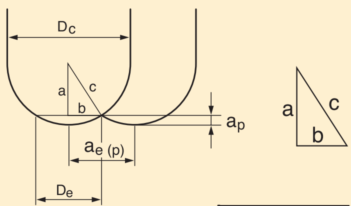
Modern research has unfortunately not been able to verify this theory to its full extent. There is a relative decrease of the temperature at the cutting edge that starts at certain cutting speeds for different materials. The decrease is small for steel and cast iron. And bigger for aluminum and other non-ferrous metals. The definition of HSM must be based on other factors.

What is today's definition of HSM?

The discussion about high speed machining is to some extent characterised by confusion. There are many opinions, many myths and many different ways to define HSM. Looking upon a few of these definitions HSM is said to be:

On following pages the parameters that influence the machining process and having connections with HSM will be discussed. It is important to describe HSM from a practical point of view and we also want to give as many practical guidelines for the application of HSM as possible.

True cutting speed
As cutting speed is dependent on both spindle speed and the diameter of the tool, HSM should be defined as “true cutting speed“ above a certain level. The linear dependence between cutting speed and



$$D_e = 2\sqrt{a_p\,(D_c - a_p)}$$

Effective cutting speed (v_e)

$$v_e = \frac{\pi \times n \times D_e}{1000}\ m/min$$

feed rate results in “high feeds with high speeds“. The feed will become even higher if a smaller cutter diameter is chosen, provided that the feed per tooth and the number of teeth is unchanged. To compensate for a smaller diameter the rpm must be increased to keep the same cutting speed...and the increased rpm gives a higher v_f.

$$V_f = f_z \times n \times z_n\ [mm/min]$$

Formula for feed speed.

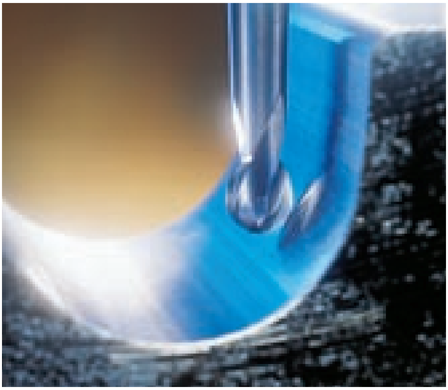
Shallow cuts
Very typical and necessary for HSM applications is that the depths of cut, ae and ap and the average chip thickness, hm, are much lower compared with conventional machining. The material removal rate, Q, is consequently and considerably smaller than in conventional machining. With the exception when machining in aluminium, other non- ferrous materials, in finishing and superfinishing operations in all types of materials.

$$Q = \frac{a_p \times a_e \times v_f}{1000}\ [cm^3/min]$$

Formula for material removal rate.

Characteristics of today’s HSM in hardened tool steel
Within the die & mould area the maximum economical workpiece size for roughing to finishing with HSM is approximately 400 X 400 X 150 (l, w, h). The maximum size is related to the relatively low material removal rate in HSM. And of course also to the dynamics and size of the machine tool.

Most dies or moulds have a considerably smaller size, than mentioned above, in complete machining (single set-up). Typical operations performed are, roughing, semi-finishing, finishing and in many cases super-finishing. Restmilling of corners and

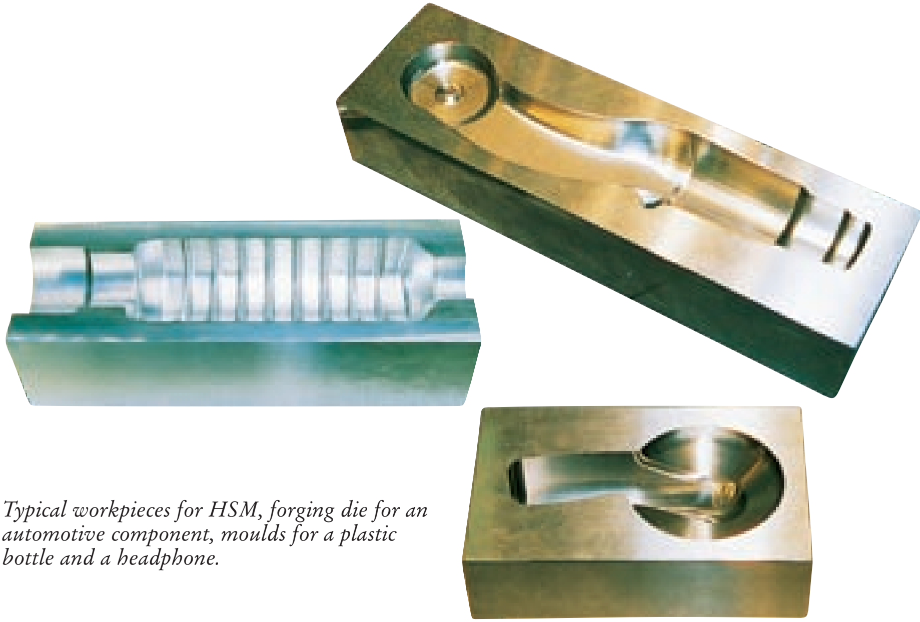


radii should always be done to create constant stock for the following operation and tool. In many cases 3-4 tool types are used.

The common diameter range is from 1 - 20 mm. The cutting material is in 80 to 90% of the cases solid carbide end mills or ball nose end mills. End mills with big corner radii are often used. The solid carbide tools have reinforced cutting edges and neutral or negative rakes (mainly for materials above 54 HRC). One typical and important design feature is a thick core for maximum bending stiffness.

It is also favourable to use ball nose end mills 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 a small clearance. It is also possible to use smaller sized cutting tools with indexable inserts. Especially for roughing and semi-finishing. These should have maximum shank stability and bending stiffness. A tapered shank improves the rigidity. And so does also shanks made of heavy metal.

SANDVIK Coromant



Typical workpieces for HSM, forging die for an automotive component, moulds for a plastic bottle and a headphone.

The geometry of the die or mould could preferably be shallow and not too complex. Some geometrical shapes are also more suited for high productive HSM. The more possibilities there are to adapt contouring tool paths in combination with downmilling, the better the result will be.

One main parameter to observe when finishing or super-finishing in hardened tool steel with HSM is to take shallow cuts. The depth of cut should not exceed 0,2/0,2 mm (ae/ap). This is to avoid excessive deflection of the holding/cutting tool and to keep a high tolerance level and geometrical accuracy on the machined die or mould. An evenly distributed stock for each tool will also guarantee a constant and high productivity. The cutting speed and feed rate will be on constant high levels when the ae/ap is constant. There will be less mechanical variations and work load on the cutting edge plus an improved tool life.

Cutting data

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

Roughing

True v_c: 100 m/min, ap: 6-8% of the cutter diameter, a_e: 35-40% of the cutter diameter, f_z: 0,05-0,1 mm/z

Semi-finishing

True v_c: 150-200 m/min, ap: 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

True vc: 200-250 m/min, ap: 0,1-0,2 mm,

ae: 0,1-0,2 mm, fz: 0,02-0,2 mm/z
The values are of course dependent of outstick, overhang, stability in the application, cutter diameters, material hardness etc. They should be looked upon as typical and realistic values. In the discussion about HSM applications one can sometimes see that extremely high and unrealistic values for cutting speed is referred to. In these cases vc has probably been calculated on the nominal diameter of the cutter. Not the effective diameter in cut.

One example:

- End mill with 90 degree corner, diameter 6 mm. Spindle speed at true cutting speed of 250 m/min = 13 262 rpm

- Ball nose end mill, nominal diameter 6 mm. a_p 0,2 mm gives effective diameter in cut of 2,15 mm. Spindle speed at true cutting speed of 250 m/min = 36 942 rpm

HSM Cutting Data by Experience

Material	Hardness	Conv. v_c	HSM v_e, R	HSM v_e, F
Steel 01.2	150 HB	<300	>400	<900
Steel 02.1/2	330 HB	<200	>250	<600
Steel 03.11	300 HB	<100	>200	<400
Steel 03.11	39 -48 HRc	<80	>150	<350
Steel 04	48-58 HRc	<40	>100	<250
GCI 08.1	180 HB	<300	>500	<3000
Al/Kirksite	60-75 HB	<1000	>2000	<5000
Non-ferr	100 HB	<300	>1000	<2000

Dry milling with compressed air or oil mist under high pressure is recommended.

Practical definition of HSM - conclusion

- HSM is not simply high cutting speed. It should be regarded as a process where the operations are performed with very specific methods and production equipment.

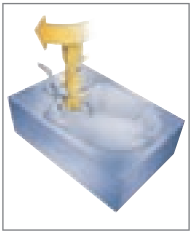
- HSM is not necessarily high spindle speed machining. Many HSM applications are performed with moderate spindle speeds and large sized cutters.

- HSM is performed in finishing in hardened steel with high speeds and feeds, often with 4-6 times conventional cutting data.
- HSM is High Productive Machining in small-sized components in roughing to finishing and in finishing and super-finishing in components of all sizes.

- HSM will grow in importance the more net shape the components get.

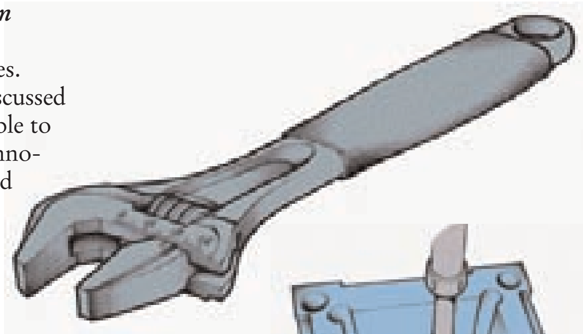
- HSM is today mainly performed in taper 40 machines.

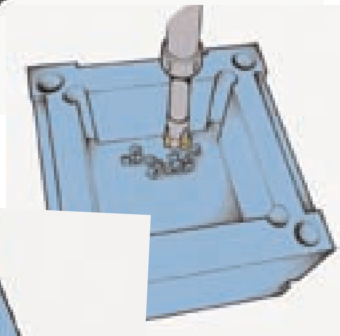
THE APPLICATION OF HIGH SPEED MACHINING



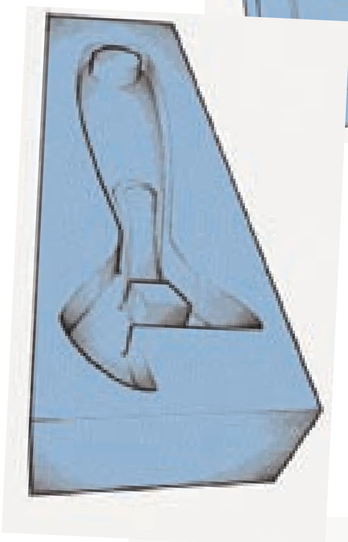
Main application areas for HSM

Milling of cavities.
As have been discussed earlier, it is possible to apply HSM-technology (High Speed Machining) in qualified, high-alloy tool steels up to 60-63 HRc.





When milling cavities in such hard materials, it is crucial to select adequate cutting and holding tools for each specific operation; roughing, semi-finishing and finishing. To have success, it is also very important to use optimised tool paths, cutting data and cutting strategies. These things will be discussed in detail in coming chapters.

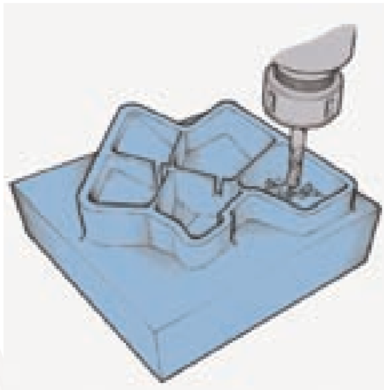


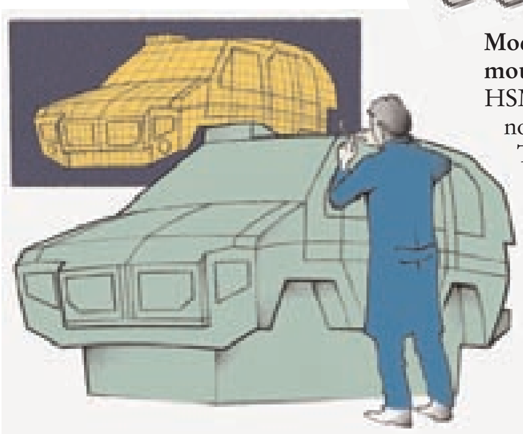
Die casting dies. This is an area where HSM can be utilised in a productive way as most die casting dies are made of demanding tool steels and have a moderate or small size.

Forging dies. Most forging dies are suitable for HSM due to the shallow geometry that many of them have. Short tools always results in higher productivity due to less bending (better stability). Maintenance of forging dies (sinking of the geometry) is a very demanding operation as the surface is very hard and often also has cracks.

Injection moulds and blow moulds are also suitable for HSM, especially because of their (most often) small size. Which makes it economical to perform all operations (from roughing to finishing) in one set up. Many of these moulds have relatively deep cavities. Which calls for a very good planning of approach, retract and overall tool paths. Often long and slender shanks/extensions in combination with light cutting tools are used.

Milling of electrodes in graphite and copper. An excellent area for HSM. Graphite can be machined in a productive way with TiCN-, or diamond coated solid carbide endmills. The trend is that the manufacturing of electrodes and employment of EDM is steadily decreasing while material removal with HSM is increasing.





Modelling and prototyping of dies and moulds. One of the earliest areas for HSM. Easy to machine materials, such as non-ferrous, aluminium, kirkzite et cetera. The cutting speeds are often as high as 1500-5000 m/min and the feeds are consequently also very high.

HSM is also very often used in direct production of -

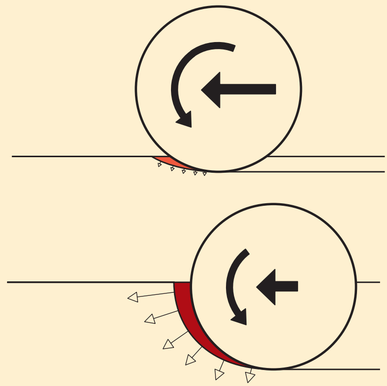
- Small batch components
- Prototypes and pre-series in Al, Ti, Cu for the Aerospace industry, Electric/Electronic industry, Medical industry Defence industry
- Aircraft components, especially frame sections but also engine parts
- Automotive components, GCI and Al
- Cutting and holding tools (through hardened cutter bodies)

Targets for HSM of dies and moulds
One of the main targets with HSM is to cut production costs via higher productivity. Mainly in finishing operations and often in hardened tool steel.

Another target is to increase the overall competitiveness through shorter lead and delivery times. The main factors, which enables this are:

- production of dies or moulds in (a few or) a single set-up
- improvement of the geometrical accuracy of the die or mould via machining, which in turn will decrease the manual labour and try-out time
- increase of the machine tool and workshop utilisation via process planning with the help of a CAM-system and workshop oriented programming

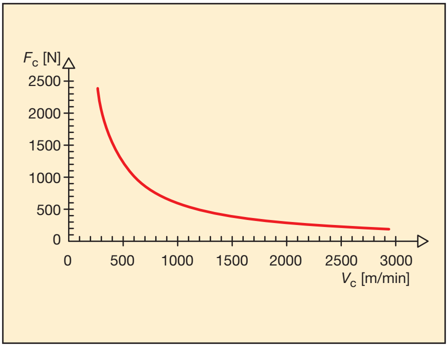
Advantages with HSM
Cutting tool and workpiece temperature are kept low. Which gives a prolonged tool life in many cases. In HSM applications, on the other hand, the cuts are shallow and the engagement time for the cutting edge is extremely short. It can be said that the feed is faster than the time for heat propagation.



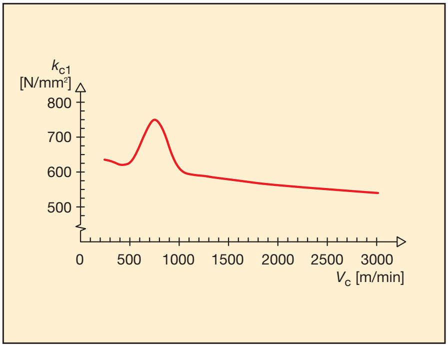
Top picture HSM, feed faster than heat propagation. Lower picture, conventional milling, time for heat propagation.

Low cutting force gives a small and consistent tool deflection. This, in combination with a constant stock for each operation and tool, is one of the prerequisites for a highly productive and safe process.

As the depths of cut are typically shallow in HSM, the radial forces on the tool and spindle are low. This saves spindle bearings, guide-ways & ball screws. HSM and axial milling is also a good combination as the



Cutting force (F_c) vs cutting speed (v_c) for a constant cutting power of 10 kW



Cutting speed vs specific cutting force in aluminium 7050

impact on the spindle bearings is small and the method also allows longer tools with less risk for vibrations.

Productive cutting process in small sized components. Roughing, semi-finishing and finishing is economical to perform when the total material removal is relatively low.

Productivity in general finishing and possibility to achieve extremely good surface finish. Often as low as Ra ~ 0,2 microns.

Machining of very thin walls is possible. As an example the wall thickness can be 0,2 mm and have a height of 20 mm if utilising the method shown in the figure. Downmilling tool paths to be used. The contact time, between edge and work piece, must be extremely short to avoid vibrations and deflection of the wall. The microgeometry of the cutter must be very positive and the edges very sharp.

The impulse law

$F_c \times t_{engage} = m_{wall} \times \Delta v_{wall}$

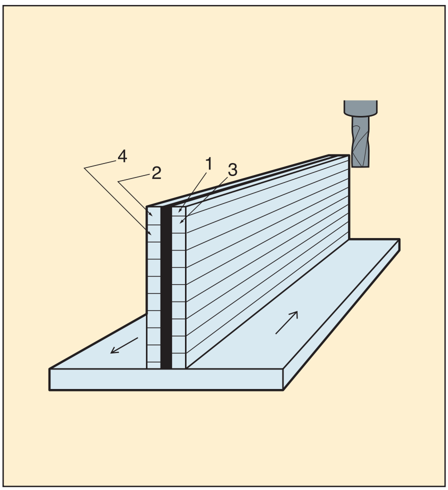
Time of engagement is reduced with HSM

▼

Impulse is reduced

▼

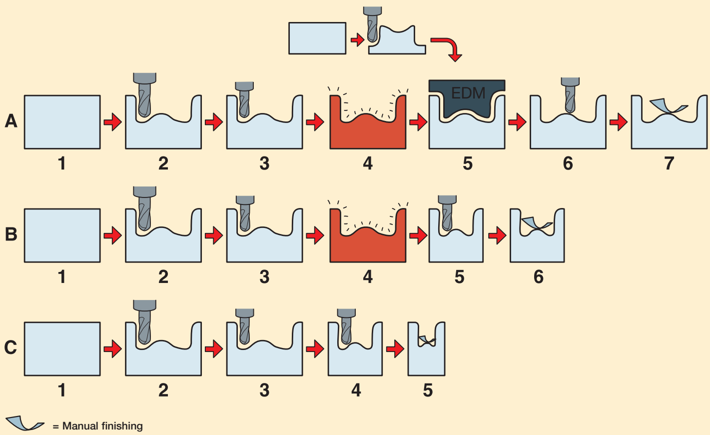
Deflection is reduced



Geometrical accuracy of dies and moulds gives easier and quicker assembly. No human being, no matter how skilled, can compete with a CAM/CNC-produced surface texture and geometry. If some more hours are spent on machining, the time consuming manual polishing work can be cut down dramatically. Often with as much as 60-100%!

Reduction of production processes as hardening, electrode milling and EDM can be minimised. Which gives lower investment costs and simplifies the logistics. Less floor space is also needed with fewer EDM-equipment. HSM can give a dimensional tolerance of 0,02 mm, while the tolerance with EDM is 0,1-0,2 mm. The durability, tool life, of the hardened die or mould can sometimes be increased when EDM is replaced with machining. EDM can, if incorrectly performed, generate a thin, re-hardened layer directly under the melted top layer. The re-hardened layer can be up to ~20 microns thick and have a hardness of up to 1000 Hv. As this layer is considerably harder than the matrix it must be removed. This is often a time consuming and difficult polishing work. EDM can also induce vertical fatigue cracks in the melted and resolidified top layer. These cracks can, during unfavourable conditions, even lead to a total breakage of a tool section.

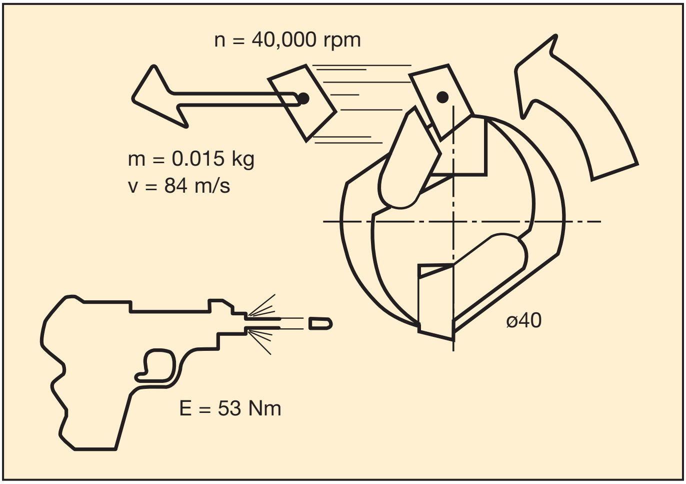
Design changes can be made very fast via CAD/CAM. Especially in cases where there is no need of producing new electrodes.



A) *Traditional process. Non-hardened (soft) blank (1), roughing (2) and semi-finishing (3). Hardening to the final service condition (4). EDM process - machining of electrodes and EDM of small radii and corners at big depths (5). Finishing of parts of the cavity with good accessability (6). Manual finishing (7).*

B) *Same process as (A) where the EDM-process has been replaced by finish machining of the entire cavity with HSM (5). Reduction of one process step.*

C) *The blank is hardened to the final service condition (1), roughing (2), semi-finishing (3) and finishing (4). HSM most often applied in all operations (especially in small sized tools). Reduction of two process steps. Normal time reduction compared with process (A) by approximately 30-50%.*



An example of the consequences of breakage at high speed machining is that of an insert breaking loose from a 40 mm diameter end mill at a spindle speed of 40.000 rpm. The ejected insert, with a mass of 0.015 kg, will fly off at a speed of 84 m/s, which is an energy level of 53 N_m - equivalent to the bullet from a pistol and requiring armour plated glass.

Some disadvantages with HSM

- The higher acceleration and deceleration rates, spindle start and stop give a relatively faster wear of guide ways, ball screws and spindle bearings. Which often leads to higher maintenance costs…
- Specific process knowledge, programming equipment and interface for fast data transfer needed.
- It can be difficult to find and recruit advanced staff.
- Considerable length of "trial and error" period.
- Emergency stop is practically unnecessary! Human mistakes, hard-, or software errors give big consequences!
- Good work and process planning necessary - "feed the hungry machine..."
- Safety precautions are necessary: **Use machines with safety enclosing - bullet proof covers! Avoid long overhangs on tools. Do not use "heavy" tools and adaptors. Check tools, adapters and screws regularly for fatigue cracks. Use only tools with posted maximum spindle speed. Do not use solid tools of HSS!**

Some typical demands on the machine tool and the data transfer in HSM (ISO/BT40 or comparable size, 3-axis)

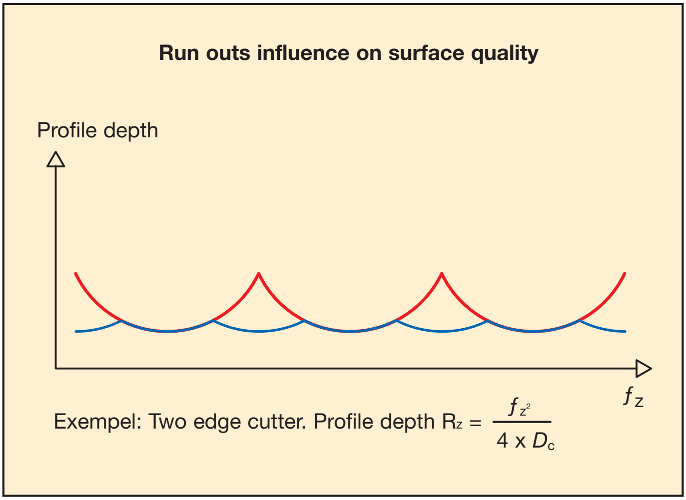
- Spindle speed range < = 40 000 rpm
- Spindle power > 22 kW
- Programmable feed rate 40-60 m/min
- Rapid traverse < 90 m/min
- Axis dec./acceleration > 1 g (faster w. linear motors)
- Block processing speed 1-20 ms
- Data flow via Ethernet 250 Kbit/s (1 ms)
- Increments (linear) 5-20 microns
- Or circular interpolation via NURBS (no linear increments)
- High thermal stability and rigidity in spindle - higher pretension and cooling of spindle bearings
- Air blast/coolant through spindle
- Rigid machine frame with high vibration absorbing capacity
- Different error compensations - temperature, quadrant, ball screw are the most important
- Advanced look ahead function in the CNC

Some specific demands on cutting tools made of solid carbide

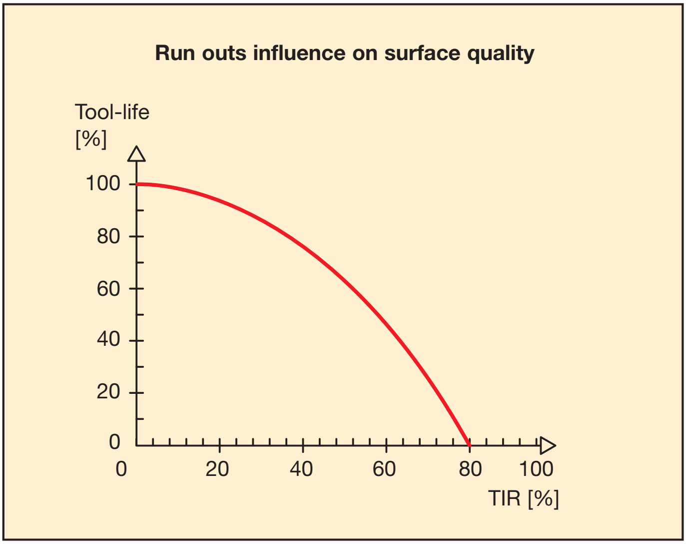
- High precision grinding giving run-out lower than 3 microns
- Shortest possible protrusion and overhang, maximum stiffness, and thick core for lowest possible deflection
- Short edge and contact length for lowest possible vibration risk, low cutting forces and deflection
- Oversized and tapered shanks, especially important on small diameters
- Micro grain substrate, TiAlN-coating for higher wear resistance/hot hardness
- Holes for air blast or coolant
- Adapted, strong micro geometry for HSM of hardened steel
- Symmetrical tools, preferably balanced by design

Specific demands on cutters with indexable inserts

- Balanced by design
- High precision regarding run-out, both on tip seats and on inserts, maximum 10 microns totally...
- Adapted grades and geometries for HSM in hardened steel
- Good clearance on cutter bodies to avoid rubbing when tool deflection (cutting forces) disappears
- Holes for air blast or coolant
- Marking of maximum allowed rpm directly on cutter bodies.



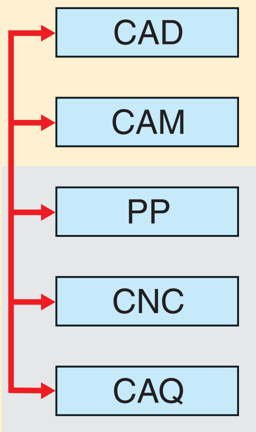
Surface with (red line) and without (blue line) run-out.



Tool life as a function of TIR % of chip thickness

CAD/CAM and CNC structures
HSM processes has underlined the necessity to develop both the CAM-, and CNC-technology radically. HSM is not simply a question of controlling and driving the axes and turning the spindles faster. HSM applications creates a need of much faster data communication between different units in the process chain. There are also specific conditions for the cutting process in HSM applications that conventional CNCs can not handle.

The typical structure for generating data and perform the cutting and measuring process may look like this:



Workpiece geometry. CAD - creation/design of a geometrical 3D model based on advanced mathematical calculations (Bezier curves or NURBS)

Generic tool path. Creation of CAM - files representing tool paths, methods of approach, tool and cutting data et cetera

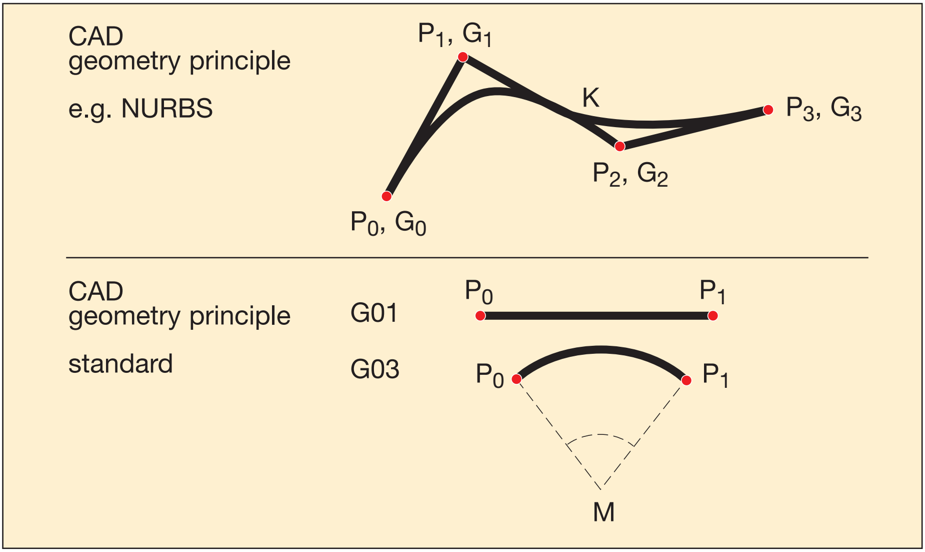
NC program. Generation of a part program (NC-program) via post-processing of CAM - files to a specific type of control

Workpiece. Machining of the component, die or mould etc. via commands from a CNC

Measuring data. Registration and feedback of measuring data, CAQ, Computer Aided Quality assurance

Re-design, re-engineering or direct tool compensations due to tool wear or change of tools etc.. New loop begins.

This type of process structure is characterised by specific configuration of data for each computer. The communication of data between each computer in this chain has to be adapted and translated. And the communication is always of one way-type. There are often several types of interfaces without a common standard.



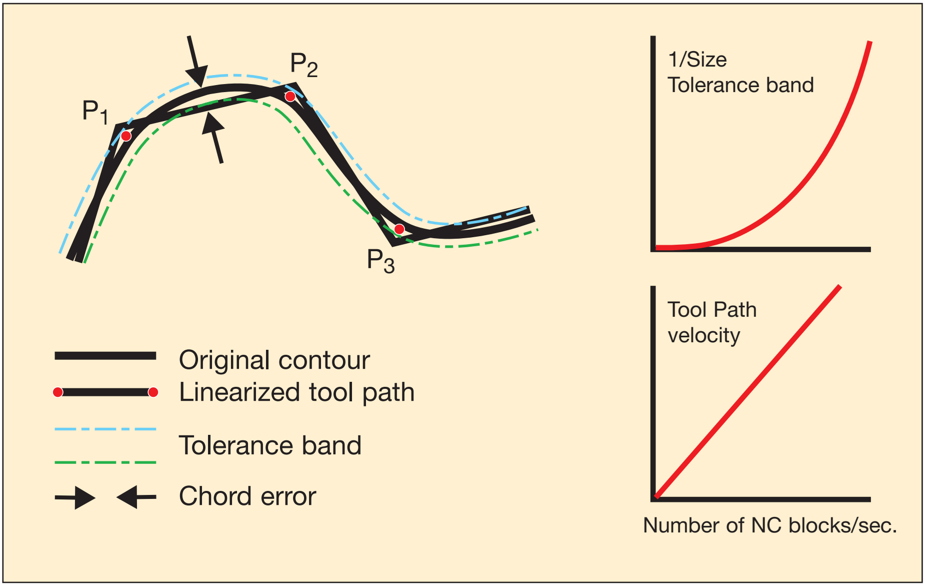
Problem areas

The main problem is that a conventional control (CNC) does not understand the advanced geometrical information from the CAD/CAM systems without a translation and simplification of the geometry data.

This simplification means that the higher level geometry (complex curves) from the CAD/CAM is transformed to tool paths via primitive approximations of the tool paths, based on straight lines between points within a certain tolerance band. Instead of a smooth curve line geometry there will be a linearised tool path. In order to avoid visible facets, vibration marks and to keep the surface finish on a high level on the component the resolution has to be very high. The smaller the tolerance band is (typical values for the distance between two points range from 2 to 20 microns), the bigger the number of NC-blocks will be. This is also true for the speeds - the higher cutting and surface speed the bigger the number of NC-blocks.

This has today resulted in limitations of some HSM applications as the block cycle times have reached levels close to 1 msec. Such short block cycle times requires a very huge data transfer capacity. Which will create bottle necks for the entire process by overloading factory networks and also demand large CNC-memory and high computing power.

If one NC-block typically consists of 250 bits and if the block cycle time ranges between 1 - 5 msec the CNC has to handle between 250 000 to 50 000 bits/sec!



New NURBS-based technology

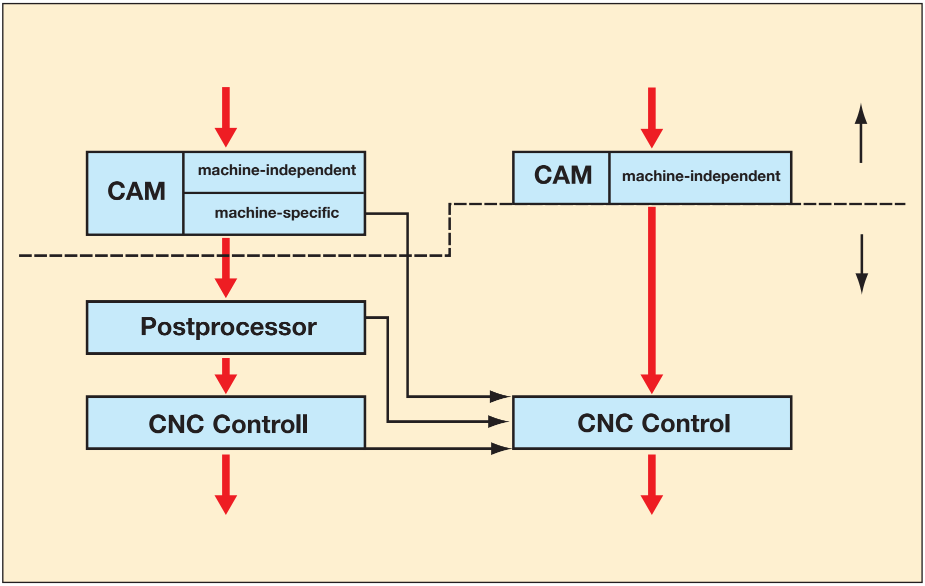
The recently developed solution on the above problems is based on what could be called ”machine independent NC-programming”.

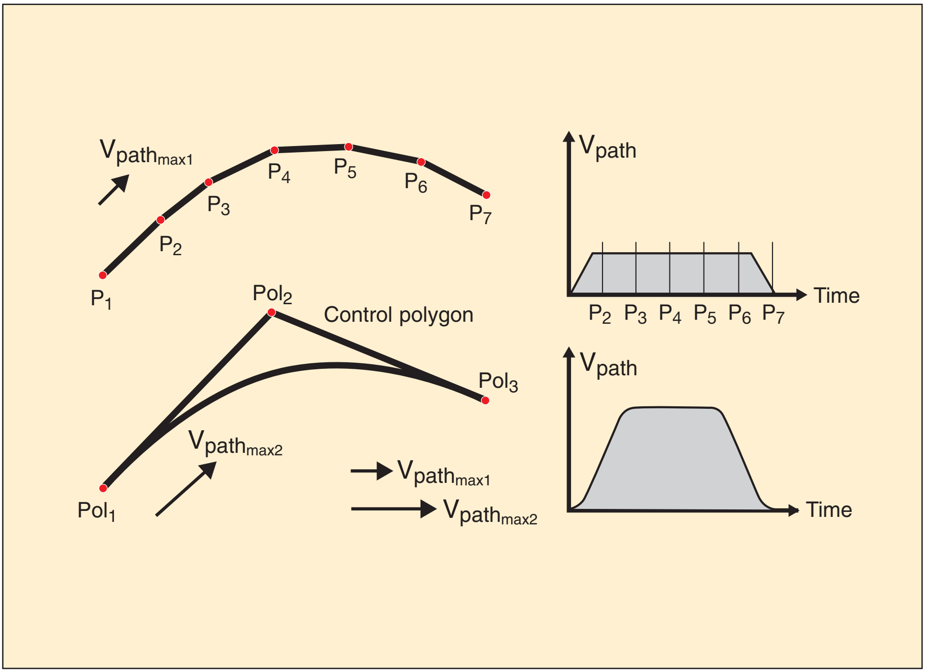
This integration of CAD ⇒ CAM ⇒ CNC implys that the programming of the CNC considers a generic machine tool that understands all geometrical commands coming from the NC-programming. The technique is based on the CNC automatically adapting the specific axle and cutter configuration for each specific machine tool and set-up.

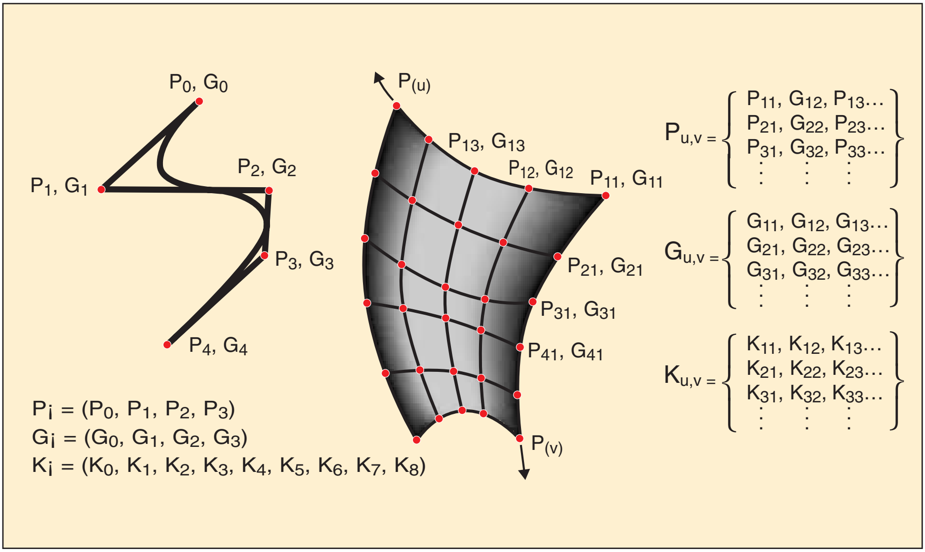
This includes, for instance, corrections of displacements of workpieces (on the machine table) without any changes in the NC-program. This is possible as the NC-program is relative to different deviations from the real situation.

Tool paths based on straight lines have non continuous transitions. For the CNC this means very big jumps in velocity between different directions of the machine axes. The only way the CNC can handle this is by slowing down the speed of the axes in the ”change of direction situation”, for instance in a corner. This means a severe loss of productivity.

A NURBS is built up by three parameters. These are poles, weights and knots. As NURBS are based on non-linear movements the tool paths will have continuous transitions and it is possible to keep much higher acceleration, deceleration and interpolation speeds. The productivity increase can be as much as 20-50%. The smoother movement of the mechanics also results in better surface finish, dimensional and geometrical accuracy.


CAM
machine-independent
machine-specific
Postprocessor
CNC Controll
CAM
machine-independent
CNC Control


Vpathmax1
P1
P2
P3
P4
P5
P6
P7
Pol2
Control polygon
Pol3
Vpathmax2
Pol1
Vpathmax1
Vpathmax2
Vpath
Time
P2 P3 P4 P5 P6 P7
Vpath
Time



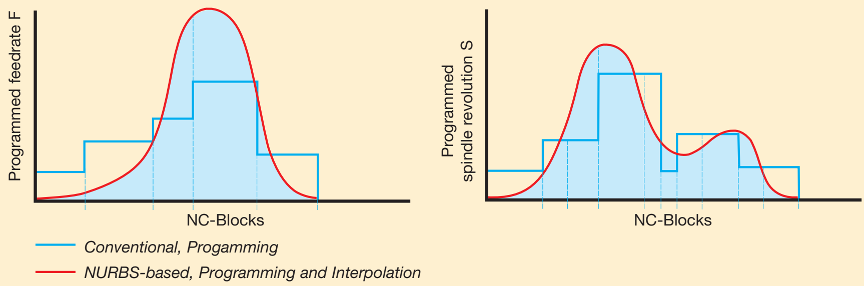
Conventional CNC-technology does not know anything about cutting conditions. CNCs strictly care only about geometry. Today's NC-programs contains constant values for surface and spindle speed.

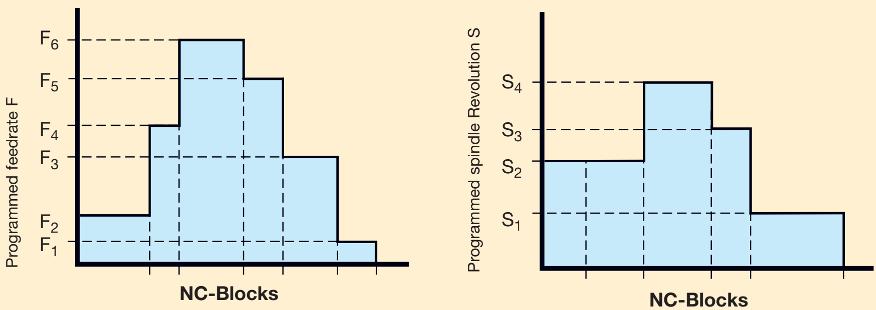
Within one NC-block the CNC can only interpolate one constant value. This gives a "step-function" for the changes of feed rate and spindle speed.

These quick and big alterations are also creating fluctuating cutting forces and bending of the cutting tool, which gives a big negative impact on the cutting conditions and the quality of the workpiece. These problems can however be solved if NURBS-interpolation is applied also for technological commands. Surface and spindle speed can be programmed with the help of NURBS, which give a very smooth and favourable change of cutting conditions.

Constant cutting conditions mean successively changing loads on the cutting tools and are as important as constant amount of stock to remove for each tool in HSM applications.

NURBS-technology represents a high density of NC-data compared to linear programming. One NURBS-block represents, at a given tolerance, a big number of conventional NC-blocks. This means that the problems with the high data communication capacity and the necessity of short block cycle time are solved to a big extent.





- Dramatic changes of cutting conditions
- Waste of machine productivity
- High tool wear
- Limited part quality

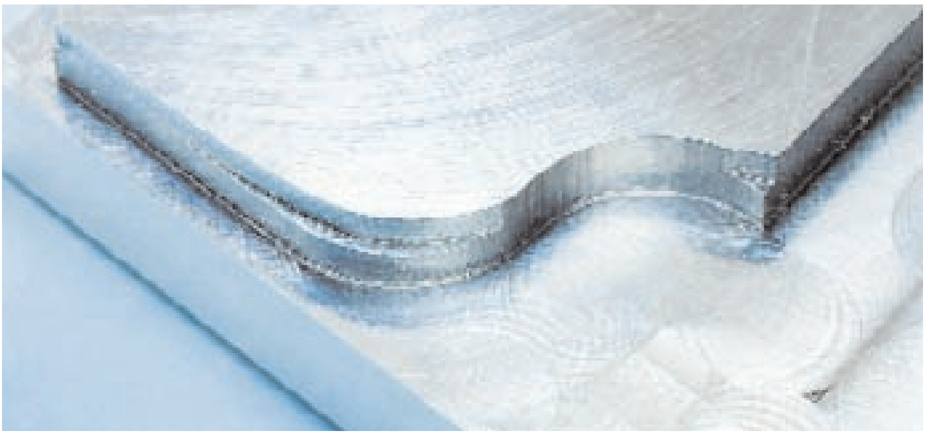
Look Ahead Function

In HSM applications the execution time of a NC-block can sometimes be as low as 1 ms. This is a much shorter time than the reaction time of the different machine tool functions - mechanical, hydraulic and electronic.

In HSM it is absolutely essential to have a look ahead function with much built in geometrical intelligence. If there is only a conventional look ahead, that can read a few blocks in advance, the CNC has to slow down and drive the axes at such low surface speed so that all changes in the feed rate can be controlled. This makes of course no HSM applications possible.

An advanced look ahead function must read and check hundreds of blocks in advance in real time and identify/define those cases where the surface speed has to be changed or where other actions must be taken.

An advanced look ahead analyses the geometry during operation and optimises the surface speed according to changes in the curvature. It also controls that the tool path is within the allowed tolerance band. A look ahead function is a basic software function in all controls used for HSM. The design, the usefulness and versatility can differ much depending on concept.



The upper pass on the component is machined with a machine control without sufficient look ahead function and it clearly shows that the corners have been cut, compared with the lower pass machined with sufficient look ahead function

Cutting fluid in milling

Modern cemented carbides, especially coated carbides, do not normally require cutting fluid during machining. GC grades perform better as regards to tool life and reliability when used in a dry milling environment.

This is even more valid for cermets, ceramics, cubic boron nitride and diamond.

Today's high cutting speeds results in a very hot cutting zone. The cutting action takes place with the formation of a flow zone, between the tool and the workpiece, with temperatures of around 1000 degrees C or more.

Any cutting fluid that comes in the vicinity of the engaged cutting edges will instantaneously be converted to steam and have virtually no cooling effect at all.

The effect of cutting fluid in milling is only emphasising the temperature variations that take place with the inserts going in and out of cut. In dry machining variations do take place but within the scope of what the grade has been developed for (maximum utilisation). 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 ending of the tool life. The hotter the machining zone is, the more unsuitable it is to use cutting fluid. Modern carbide grades, cermets, ceramics and CBN are designed to withstand constant, high cutting speeds and temperatures.

When using coated milling grades the thickness of the coating layer plays an important role. A comparison can be made to the difference in pouring boiling water simultaneously into a thick-wall and a thin-wall glass to see which cracks, and that of inserts with thin and thick coatings, with the application of cutting fluid in milling.

A thin wall or a thin coating lead to less thermal tensions and stress. Therefore, the glass with thick walls will crack due to the large temperature variations between the hot inside and the cold outside. The same theory goes for an insert with a thick coating. Tool life differences of up to 40%, and in some specific cases even more, are not unusual, to the advantage of dry milling.

If machining in sticky materials, such as low carbon steel and stainless steel, has to take place at speeds where built-up edges are formed, certain precautions need to be taken. The temperature in the cutting zone should be either above or below the unsuitable area where built-up edge appears.

Achieving the flow-zone at higher temperatures eliminates the problem. No, or very small built-up edge is formed. In the low cutting speed area where the temperature in the cutting zone is lower, cutting fluid may be applied with less harmful results for the tool life.

There are a few exceptions when the use of cutting fluid could be "defended" to certain extents:

- Machining of heat resistant alloys is generally done with low cutting speeds. In some operations it is of importance to use coolant for lubrication and to cool down the component. Specifically in deep slotting operations.
- Finishing of stainless steel and aluminium to prevent smearing of small particles into the surface texture. In this case the coolant has a lubricating effect and to some extent it also helps evacuating the tiny particles.
- Machining of thin walled components to prevent geometrical distortion.
- When machining in cast iron and nodular cast iron the coolant collects the material dust. (The dust can also be collected with equipment for vacuum cleaning).
- Flush pallets, components and machine parts free from swarf. (Can also be done with traditional methods or be eliminated via design changes).
- Prevent components and vital machine parts from corrosion.

If milling has to be performed wet, coolant should be applied copiously and a cemented carbide grade should be used which is recommended for use in wet as well as in dry conditions. It can either be a modern grade with a tough substrate having multi-layer coatings. Or a somewhat harder, micro-grain carbide with a thin PVD coated TiN layer.

Essential savings can be done via dry machining:

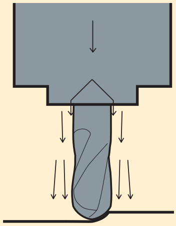
- Increases in productivity as per above.
- Production costs lowered. The cost of coolant and the disposal of it represent 15-20% of the total production costs! This could be compared to that of cutting tools, amounting to 4-6% of the production costs.
- Environmental and health aspects. A cleaner and healthier workshop with bacteria formation and bad smells eliminated.
- No need of maintenance of the coolant tanks and system. It is usually necessary to make regular stoppages to clean out machines and coolant equipment.
- Normally a better chip forming takes place in dry machining.

Cutting fluid in HSM

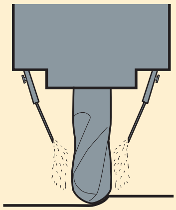
In conventional machining, when there is much time for heat propagation, it can sometimes be necessary to use coolant to prevent excessive heat from being conducted into; the workpiece, cutting and holding tool and eventually into the machine spindle. The effects on the application may be that the tool and the workpiece will extend somewhat and tolerances can be in danger.

This problem can be solved in different ways. As have been discussed earlier, it is much more favourable for the die or mould accuracy to split roughing and finishing into separate machine tools. The heat conducted into the workpiece or the spindle in finishing can be neglected. Another solution is to use a cutting material that does not conduct heat, such as cermet. In this case the main portion of the heat goes out with the chips, even in conventional machining.

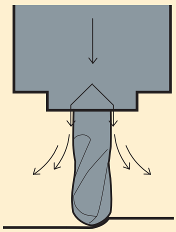
It may sound trivial, but one of the main factors for success in HSM applications is the total evacuation of chips from the cutting zone. Avoiding recutting of chips when working in hardened steel is absolutely essential for a predictable tool life of the cutting edges and for a good process security.



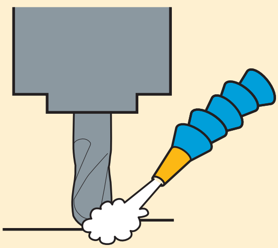
The best way to ensure a perfect chip evacuation is to use compressed air. It should be well directed to the cutting zone. Absolutely best is if the machine tool has an option for air through the spindle.



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



Third comes coolant with high pressure (approximately 70 bar or more) and good flow. Preferably also through the spindle.

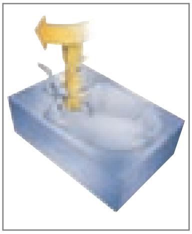


The worst case is ordinary, external coolant supply, with low pressure and flow.

If using cemented carbide or solid carbide the difference in tool life between the first and the last alternative may be as much as 50% .

If using cermet, ceramic or cubic boron nitride coolant should not be an option at all.

THE INSERT AND ITS PARAMETERS

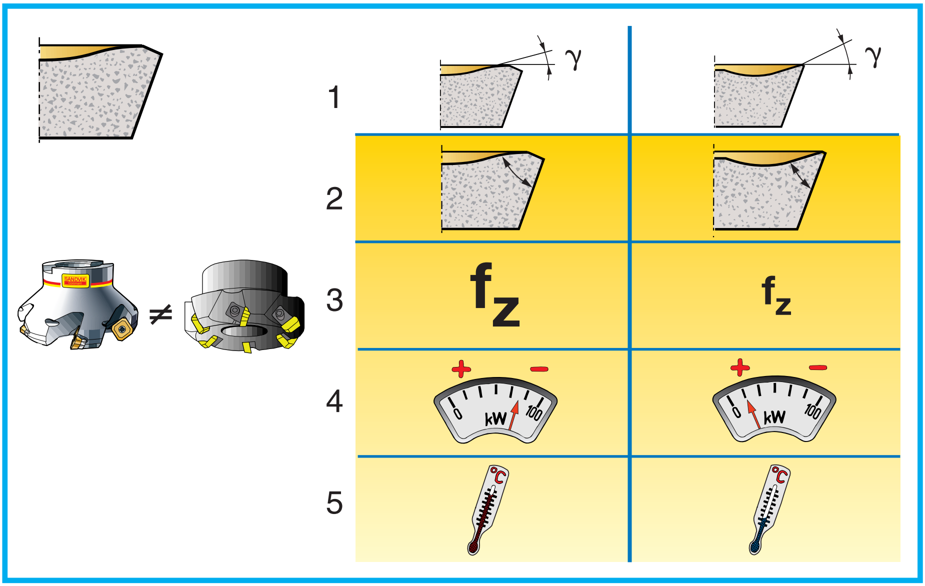


There are a wide range of different parameters that make up an insert for different conditions and workipece materials, different substrates with different grain sizes, different coatings, different angles and shapes, to mention a few. Let us take a short look at the most important ones.

Macro-geometry
The macro-geometry decides the type of rake-angle (γ), which determines if it is a geometry for roughing or if it is a geometry for light cutting. How this effects some of the cutting parameters is shown in the table.

If there is a light cutting geometry there is a large rake angle resulting in a sharper cutting edge (1). This cutting edge is not as strong as an insert with a smaller rake angle, such as a geometry for roughing (2). Due to this the feed per tooth has to be kept low in comparison (3). However, the sharp cutting edge is favourable in terms of cutting forces which will be low and machines with limited effect can be used (4). The temperature (5) will also be low due to the light cutting action which often is an advantage.

Micro-geometry
The edge rounding (ER), size and angle of the negative chamfer is decided by the micro-geometry. With larger (ER) or negative chamfer the stronger the cutting edge becomes and resulting in a possibility to use high feed rates. Large ER and negative chamfer will also produce higher cutting forces and higher temperature, which may cause plastic deformation of the cutting edge. If a component with thin walls or



light cutting action is needed small ER and negative chamfer should be used.
Some examples of ER and negative chamfer for different materials:

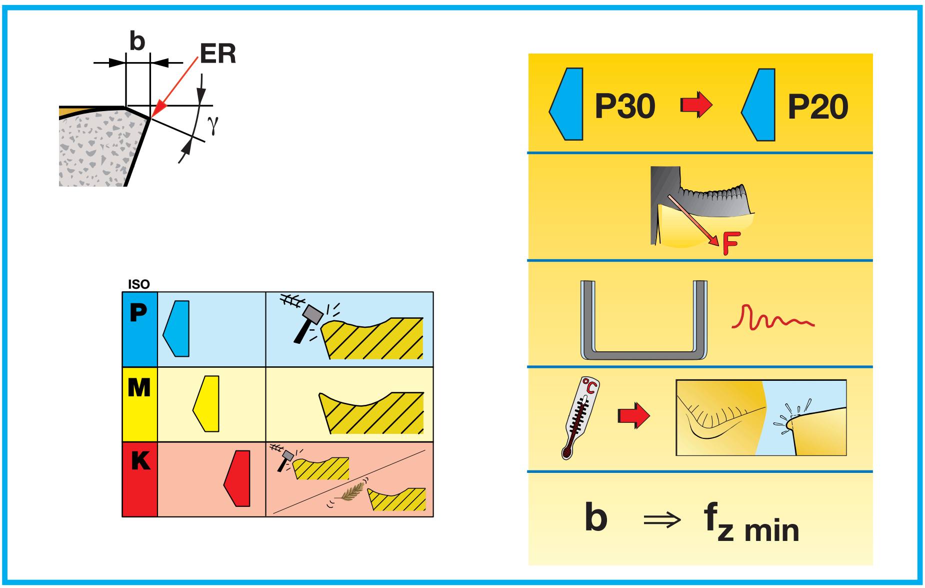
Steel geometries usually have large ER and negative chamfers to produce a strong cutting edge capable of withstanding any shock impacts when machining steel.

Stainless steel is a work-hardening material and if an insert with large ER and negative chamfer, such as one for steel milling which produces high tempratures, is used the workpiece material usually work-hardens and becomes more difficult to machine when it is time for the second pass. Sharp geometries with small ER and negative chamfer are therefore used to get a light cutting action.

When rough machining cast-iron, strong cutting edges are needed to cope with large depths of cut and sand inclusions. This calls for large ER and negative chamfer. On the other hand when finishing in cast-iron, e.g. an engine block, the planeness and surface finish is vital. Cast-iron also has a tendency of frittering and therefore a sharp cutting edge with a small ER and negative chamfer is needed.

The chart (page 98) shows how different parameters affects the inserts wear resistance to different types of wear mechanisms, in terms of substrate content and micro-geometry.

A wolfram-carbide based insert, usually a P-grade is favourable in terms of withstanding abrasion-wear, although, it has little diffusion- and oxidation-wear resistance.



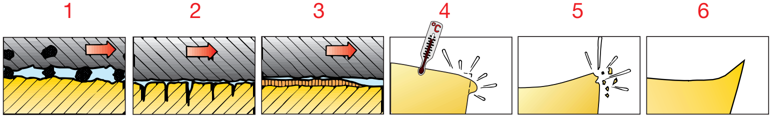
Furthermore it is good in terms of fatigue-wear, but the high content of WC is negative when it comes to adhesion-wear. To prevent plastic-deformation, a high content of WC is favourable but not in terms of chipping.

High content of cobalt, which is a binding material in carbide inserts, have some negative effects on the wear resistance of inserts. Abrasion-, diffusion- and oxidation-wear mechanisms are negatively affected by a high content of Co. Fatigue- and adhesion-wear plus chipping tendencies are enhanced by cobalt, while plastic-deformation tends to occur more easily with cobalt as binding material.

The so called g-phase TaC + TiC + NbC also affects the wear resistance of the insert in certain ways. Modern insert grades have little or no g-phase at all. However, g-phase is positive in terms of preventing abrasion-, diffusion-, oxidation-, adhesion-wear and plastic-deformation. While it is negative for prevention of fatigue and chipping.

A high proportion of hard particles makes the cemented carbide more wear resistant in that the hardness, HV, and compressive strength is greater. A high content of binding material Co, makes the insert more tough. High hardness is positive for most wear mechanisms discussed earlier except for fatigue wear and chipping. The hardness has little effect either way in terms of oxidation wear.

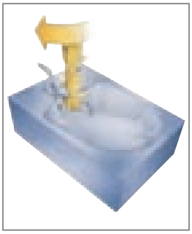
1. Abrasion wear
2. Fatique wear (static or dynamic)
3. Adhesion wear
4. Plastic deformation
5. Chipping/fracture
6. Possibility to make a scharp edge



	1	2	3	4	5	6
WC	+	o	-	+	-	o
Co	-	-/+	-/+	-	+	o
γ	+	-	+	+	-	o
HV3	+	-	-	+	-	o
Size WC	-	-/+	-	-	+	-

The last parameter to be discussed is the size of the WC grains in the insert substrate. By adjusting the size of the grains in the substrate, many different features can be accomplished even though within the same insert grade. This affects the hardness and wear resistance as well as strength and toughness. Generally, small grains means higher hardness, coarse grains more toughness. This means that cemented carbide is not brittle, but has a broad range of possibilities within machining. The insert in the wear mechanism example is an insert with coarse grains. The positive aspects with coarse grains is, as mentioned earlier, toughness and therefore favourable to prevent fatigue- and chipping. The coarse grains has negative effect on the rest of the wear mechanisms featured except for oxidation wear where it makes no difference either way.

COATING METHODS



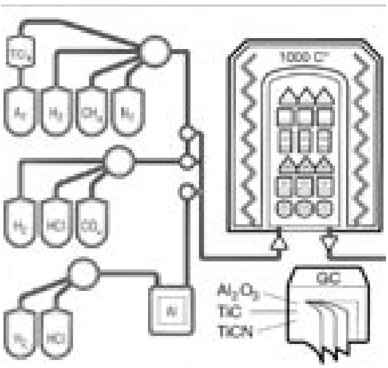
PVD

PVD (Physical Vapour Deposition) is a well established coating technique used mainly for coating of high speed steel, but is also good for coating of cemented carbide, where a sharp cutting edge is needed.

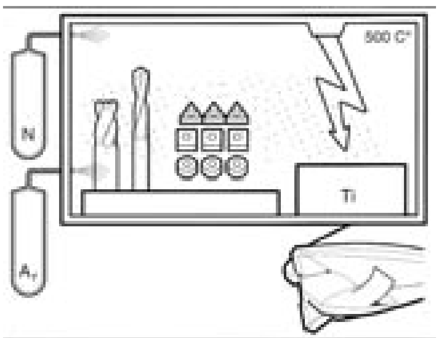
The process is based on that the coating material is moved from a material source to the substrate through either vapourization or sputtering. There are several variants of these processes as they are used widely by high speed steel suppliers. The PVD process takes place in temperatures around 500 degrees C. For instance, titanium is ionized with a focused electric beam as energy source, to form a plasma stream. Along with nitrogen this is coated on the insert. Normally, a PVD coating is thinner than a comparable CVD coating. With the CVD process a thicker coating means improved wear resistance, especially with aluminium oxide, up to a thickness of 12 micrones.

CVD

CVD (Chemical Vapour Deposition) is a more modern type of coating technique enabling multi layer coating and different layer thickness from 2-12 microns.



CVD

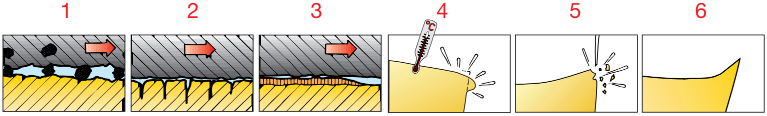


PVD

SANDVIK Coromant

Basically, CVD coating is done through chemical reactions of different gases. In the case of coating with titanium carbide; hydrogen, titanium chloride and methane. Inserts are heated to about 1000 degrees C. Like sintering this is a carefully controlled process where the carbon content, either free or as h-phase, has to be monitored through an extra carburization stage, before coating. Aluminium oxide coating carried out in a similar way as is titanium nitride coating using other gases, aluminium-chloride or nitrogen gas, respectively. The CVD process is well adapted to apply multi layered coatings as the process is relatively easy to regulate as regards to various gases. Different types of coating can be performed in the same equipment. In the table you can see how different types of coating methods are affecting the insert positive (+) or negative (-) in terms of withstanding different types of wear. Also shown is how the various types of coating can cope with the wear-mechanisms; TiAlN (Titian Aluminium Nitride), TiCN) (Titanium Carbo Nitride), Al2O3 (Aluminium Oxide), TiN (Titanium Nitride).

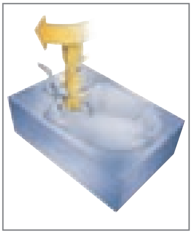
1. Abrasion wear
2. Fatique wear (static or dynamic)
3. Adhesion wear
4. Plastic deformation
5. Chipping/fracture
6. Possibility to make a scharp edge



	1	2	3	4	5	6
PVD	-	+	+	-	+	+
CVD	+	o	o	+	o	-
MT-CVD	+	o	o	+	o	-
TiAlN	+	o	+	+	-	o
Ti(C,N)	+	o	o	o	o	o
Al_2O_3	++	-	+*	+	-	-
TiN	o	o	o	o	o	o

* Polished

Choose the right grade for milling



Since steel is the most common material to be machined within the die and mould area, the Coromant grades optimized for steel milling will be explained a little bit more in detail.

For the milling operation that need a very strong cutting edge, where toughness demands are highest, the GC4040 grade (P30-P50) provides a new combination of toughness and wear resistance. A product of modern coating technology, the grade copes with the most demanding machining conditions. The objective has been to achieve high edge security primarily at low to moderate cutting speeds - yet some 25% higher than its predecessor.

GC4040 is an insert grade that will not need for operators to stand next to the stop button or continually stop the cutter worrying about the cutting edges for chipping tendencies. If applied correctly, it copes with strength-demanding steel milling, unstable conditions, vibration tendencies, varying workpiece materials and when the insert is subjected to high cutting forces.

The 90 degree entering angle in milling leaves insert corners very susceptible to high machining stresses. Not only radial forces are high, the entry and exit shock-forces are often severe. To improve the edge security, ensuring a long predictable tool-life, GC4040 is ideal. For the CoroMill 290 square shoulder facemill the grade offers improvement in performance and reliability. Also for very demanding operations, such as die and mould making, the grade means very high edge strength for round insert cutters, such as CoroMill 200. GC4040 can be applied to perform sharp light cuts as well as high feed roughing operations and is suitable for use with all three insert geometries L, M and H. With or without coolant operations (see cutting fluid in milling page 95). Also suitable for castings and forgings, the grade offers new possibilities for small batch production to be performed in a productive and very reliable way.

Allround performance

GC 4030 is more of a general purpose grade, complementary to 4040. In fact, where 4040 optimizes an area characterized by the highest strength demands, 4030 optimises milling of harder steels at higher cutting speeds and when low feed rates are involved. There is also a large steel milling application area, where toughness to a varying extent is needed, and where both insert grades can be selected.

GC4030 is capable of higher cutting speeds, offering a higher degree of wear resistance for operations that are not extreme toughness demanding.

This grade is also a product of advanced coating technology, having a harder insert substrate than 4040 but a similar aluminium oxide coating. GC4030 is especially suitable as a general, basic grade for use with 45 degree entering angle, such as in CoroMill 245, but also for round insert milling with CoroMill 200. Dry and wet milling is possible.

GC4030 is also capable of milling harder steel types as it has good resistance to thermal cracking. These inserts are excellent for milling large workpieces, requiring long cutting edge engagement times. Maintaining high cutting edge security, it provides long, predictable tool-life and generates good surface finish.

GC4020

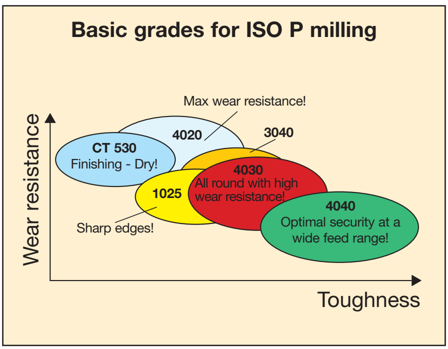
GC4020 is the High Speed Machining grade for good conditions and maximum productivity. This grade can be applied in hard and more difficult to machine materials that are becoming more and more common in die and mould making applications, especially when using HSM technology. This means that the inserts in grade GC4020 will add the possibility to machine at very high temperatures that occurs when milling hardened steel, up to HRC60, or when very high cutting speeds in general steels are used for max productivity.

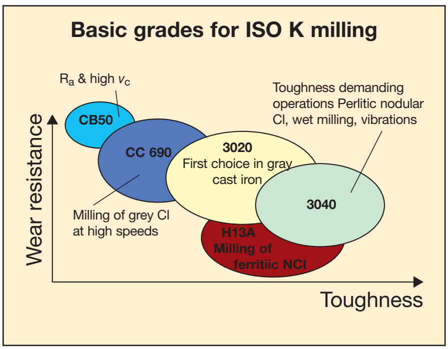
The application area for this grade is roughing to semifinishing milling in steels at fairly good conditions when highest possible chip removal rate is desired. Inserts in grade GC4020 need to be run at high temperature e g at elevated speeds and/or in hardened condition up to HRC60, as mentioned previously.

If possible, try to avoid using coolant when milling. Usage of coolant will shorten the tool-life and increase the tendency for chipping or breakage of the inserts compared to dry milling. Usage of coolant gives increased fluctuation of temperatures, which in turn leads to a larger number of thermal cracks. For a grade like GC 4020 it's important to avoid coolant in order to get the right cutting temperature and to limit the number of thermal cracks.

High wear resistance

GC3040 is an odd insert grade in this context as it was primarily developed for nodular cast iron and high tensile iron milling. However, the grade has proven to be such a good performer in steel through its high wear resistance, high temperature capability and chemical inertness, that it has been included in the range of inserts that are dedicated to optimizing steel milling.





GC3040 has specially adapted aluminium oxide coating to mill at high cutting speeds and stand up to the high temperatures of milling harder steels.

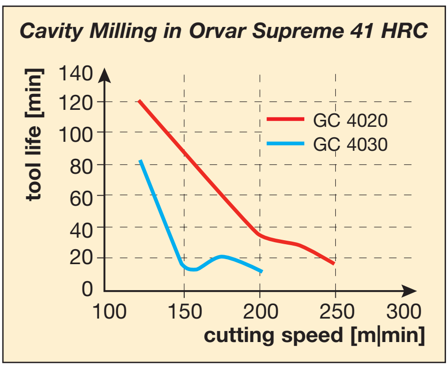
Sharp cutting edge
GC1025 is a specialist insert grade for light milling in steels such as SS1672, SS2041 and UHB steel grades Orvar and Impax high hard to mention a few. This is mainly a finishing grade which is combined with geometries to give a sharp cutting edge. Based on fine grain cemented carbide substrate, it is PVD coated and can be used in periphery ground indexable inserts.

The sharp cutting edge means that it makes satisfactory insert engagement into material also at lower feed rates and minimizes vibration tendencies in weak workpieces. It is also good at milling sticky, low carbon steels as well as super alloy materials.

High surface finish
CT530 is a cermet grade capable of generating mirror finishes (P05-P30). This is a light milling finishing insert, for small cutting depths but with a wide cutting speed range. It provides the advantageous long tool-life with consistent cutting edge performance of titanium-based cemented carbide. It has very high resistance to plastic deformation and built-up edge formation and has an added suitable application area in the form of wiper insert. Most suitable for dry conditions, this is an insert for high surface finish and accuracy demands. If round inserts in grade CT530 is combined with a strong geometry it is also good for large a_p/a_e and intermittent machinging.

The application decides the insert
When choosing grade the application must be looked upon as a whole and not just the workpiece material. In the examples it is shown what parameters should be considered.

The workpiece (mould) and workpiece material is the same for all examples. But to get the best result for each machining operation, different grades and geometries has to be used.



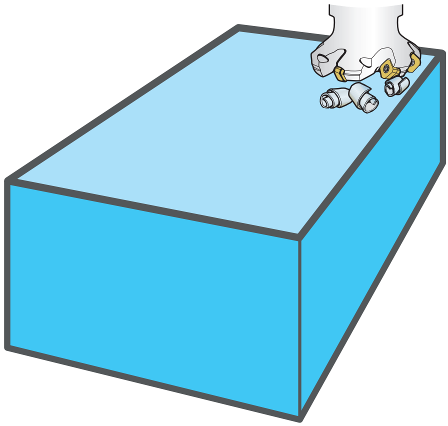
Start values for cutting speed (m/min) at different max chip thickness h_{ex} (mm) for ISO P steel milling. (a_e= 100 mm with cutter diameter 125 mm)

Material	Specific cutting force	Hardness	CT530			GC4020		
CMC	k_c 1	HB	0.05	0.15	0.25	0.1	0.2	0.3
01.1	1500	125	463	380	312	481	395	324
01.2	1600	150	427	351	288	442	363	298
01.3	1700	170	399	328	269	414	340	279
01.4	1800	210	349	287	235	362	298	244
01.5	2000	300	258	212	174	268	220	181
02.1	1700	175	334	274	225	346	284	233
02.2	2000	330	198	163	134	206	169	139
03.11	1950	200	253	208	171	300	246	202
03.13	2150	200	209	172	141	218	179	147
03.21	2900	300	181	149	122	187	154	126
03.22	3100	380	115	95	78	118	97	80
06.1	1400	150	341	280	230	353	290	238
06.2	1600	200	271	223	183	280	230	189
06.3	1950	200	198	163	134	206	169	139

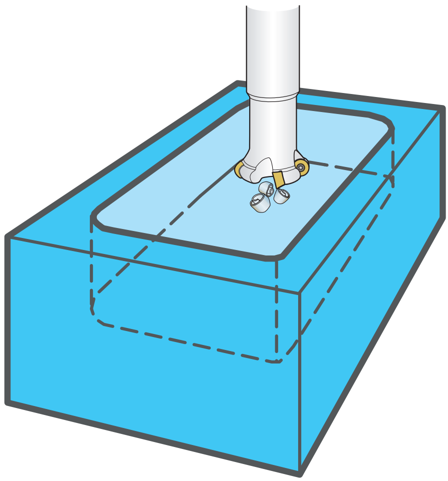
Material	Specific cutting force	Hardness	GC4030			GC4040			GC1025		
CMC	k_c 1	HB	0.1	0.2	0.3	0.1	0.2	0.4	0.05	0.1	0.2
01.1	1500	125	356	292	240	302	248	170	334	302	248
01.2	1600	150	328	269	221	278	229	154	307	278	229
01.3	1700	170	306	252	207	260	214	144	287	260	214
01.4	1800	210	268	220	181	228	188	127	252	228	188
01.5	2000	300	199	163	134	169	139	94	187	169	139
02.1	1700	175	256	210	173	218	179	121	240	218	179
02.2	2000	330	153	125	103	130	107	72	143	130	107
03.11	1950	200	194	159	131	165	136	91	182	165	134
03.13	2150	200	161	132	109	136	112	76	150	136	112
03.21	2900	300	139	114	94	118	97	66	131	118	97
03.22	3100	380	88	72	59	75	61	41	82	75	61
06.1	1400	150	262	215	177	223	183	123	246	223	183
06.2	1600	200	208	171	140	177	145	98	195	177	145
06.3	1950	200	153	125	103	130	107	72	143	130	107

Dark grey area = h_{ex} value

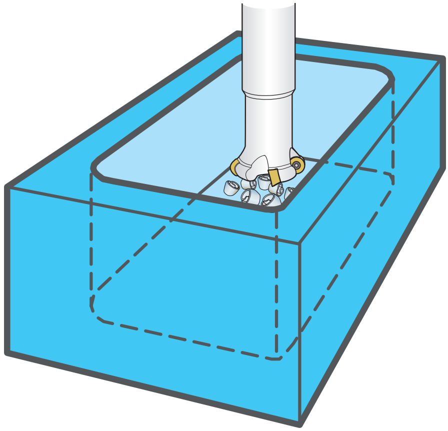
For operation one which is face-milling of the component a CoroMill 200 round insert cutter or a CoroMill 245 facemilling cutter can be used. Here the machining condition is good, no adaptors and long tools are needed. No consideration of recutting of chips has to be taken and high cutting speed can be used. Taking into account the stable and good condition of the operation an insert grade with a sharp geometry and a wear resistant grade should be used. The choice would be a grade for the P20 area.



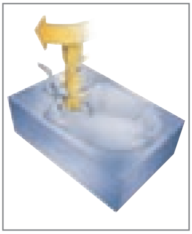
In the second operation the conditions are getting a little bit more unfavourable with a longer tool, which might mean a tendency for vibrations and recutting of chips. However, the stability and recutting of chips is not considered to have a great effect on the machining in this case and therefore medium cutting speed can be used. Here an allround geometry could be used with a slightly tougher grade than the first operation, preferably within the P30 area.



The third operation is machining deep into the cavity before semi-finishing with a ball nose endmill. The tool overhang is considerable, with a big risk of vibration occuring. Recutting of chips and sudden changes in feed direction, leading to a high variation in cutting forces. Low cutting speed has to be used to counteract the vibration and keep deflection of the tool as low as possible. An insert with strong geometry is called for in this case and a grade suitable for the P40 area. The cutting edge can at the same time not be too dull, a geometry for medium machining would be preferred instead of a geometry for roughing.



For these three operations the workpiece material is the same and the same cutter type, e.g. a CoroMill 200 cutter can be used. But to get the best machining result three different grades and insert geometries are used. The conclusion is that the operation as a whole must be examined and not just the workpiece material.

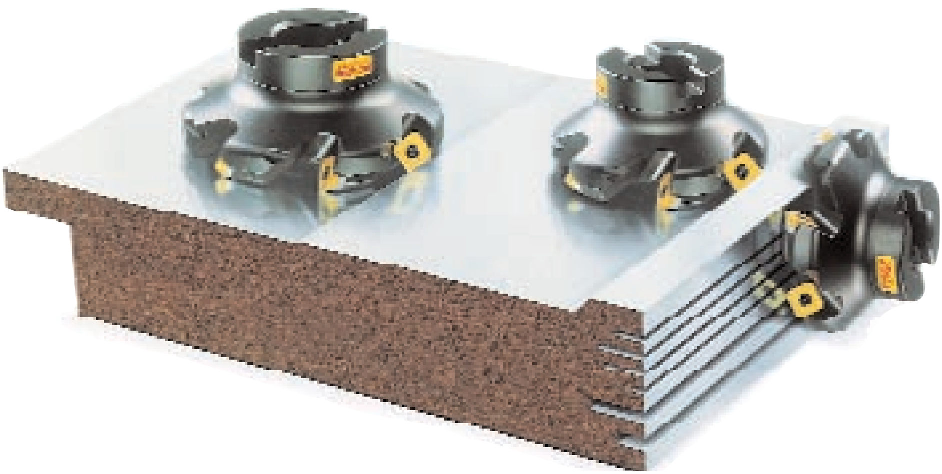


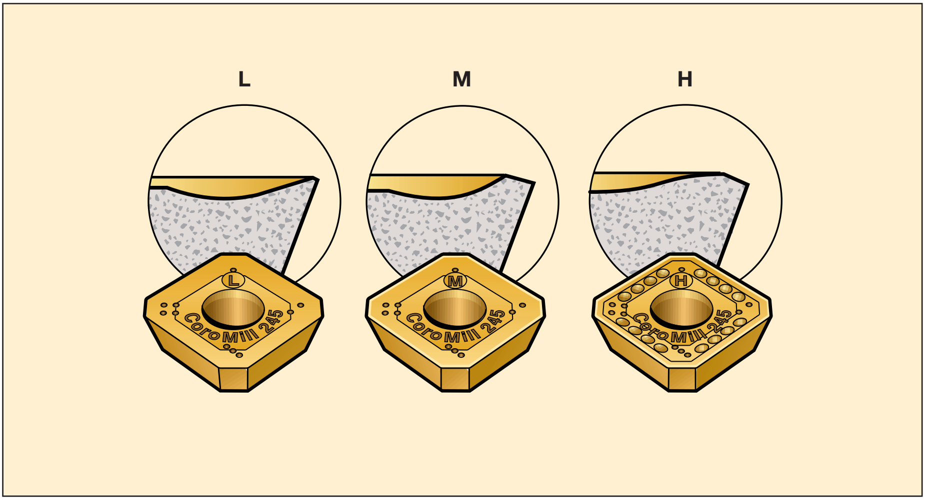
CoroMill 245

Facemilling is one of the most common milling applications and can be performed with a wide range of different tools. Facemilling cutters with 45° or 75° entering angle, square shoulder milling cutters with 90° entering angle, round insert milling cutters or even side and face milling cutters.

The most optimized of these cutters in terms of ranging from roughing to finishing with the best result and highest productivity is the 45° degree milling cutter. The CoroMill 245 cutter with 45° entering angle performs many facemilling operations better than any other tool. The cutter body is produced from hardened tool steel with a hardness of 45 HRc and all machining is done after hardening. This gives the cutter high precision because of no increase in run-out by local hardening. Furthermore it gives the cutter an even hardness with no soft zones from local hardening.

The 245 cutter programme for facemilling using a series of 12 mm square inserts covering a wide application area, from light to tough conditions. The inserts, supported by a rigid shim protected cutter body, can take depth of cut up to 6 mm. The cutter is capable of producing an excellent surface quality during a long tool life, also when used at high removal rates. Many operations can be fully finished in one pass. The main advantage of 45° entering angle is the reduced chip thickness, which allows the feed per tooth (f_z) to be substantially increased. The 45° entering angle also gives the most favourable direction of cutting forces when machining with extended tools. Furthermore, it gives less risk of edge frittering of the workpiece.



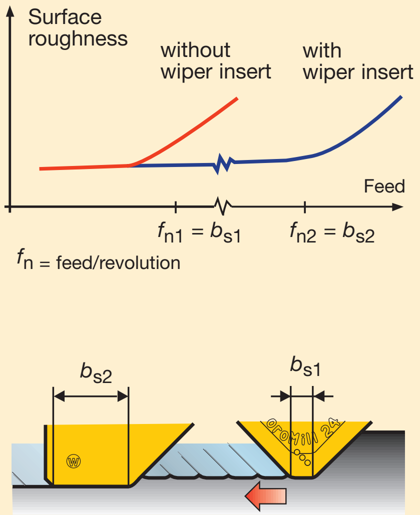


A 3 mm thick carbide shim supports the insert and protects the insert seat in case of overloading the insert. The screw mounted square inserts are self locating when positioned into the insert seat. The clear-shear geometry of the insert seats and the combination of highly positive in the axial direction and negative in the radial is of distinct advantage particularly with large depths of cut. High removal rates are easily achieved with this geometry and unfavourable positioning of the cutter becomes less critical.

The series of extra positive insert geometries cover all appropriate feed ranges required for extreme light operations as well as for the heavy ones. Wide parallel lands generate good surface finish even at high feed rates.

To further increase the surface finish or increase the feed rate with maintained surface finish the solution is to use wiper inserts. These provide a much larger flat length. The flat is located just below the ordinary parallel land of the other inserts and wipes the surface smooth, it is a finishing cutting edge that has been added to the cutter. The flat is dimensioned to cover the feed per revolution. The wiper flat insert produces good surface textures even under unfavourable conditions and are often used in short chipping materials such as grey cast-iron. Wiper inserts are also often used in aluminium alloys, in which the demands on surface finish often are high.

CoroMill® 245 wiper insert
- principal function



The CoroMilll 245 wiper insert has an 8.2 mm land which allows the feed per revolution to be increased by up to four times the normal feed, with good surface quality maintained. Especially for large diameter cutters which can fully utilize the combination of wiper- and milling inserts. Normally it is sufficient to use only one wiper insert in a cutter.

The CoroMill 245 is optimized for facemilling applications and to machine any material, type of workpiece in any machine tool.

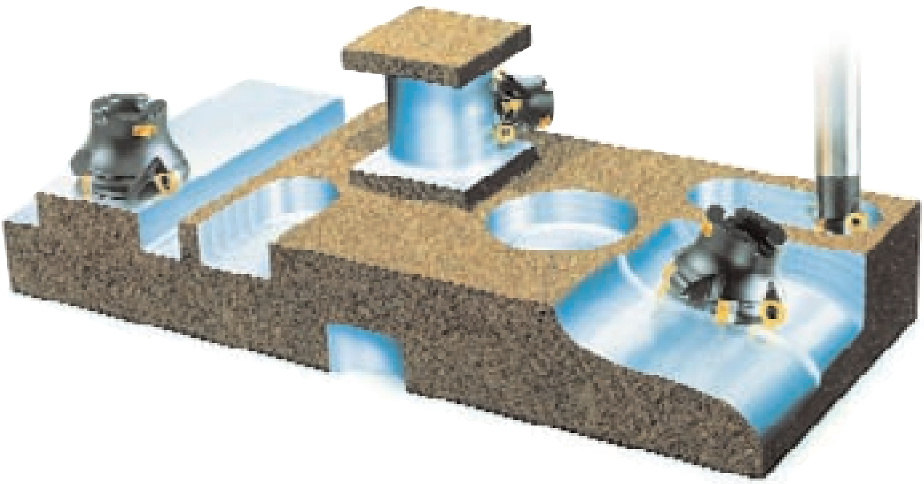
Maximum rpm and cutting speed V_c

Diameter mm	32	40	50	63	80	100	125
Max *n* rpm	29000	18250	16250	14400	12700	11300	10100
Max V_c m/min	2915	2890	2985	3365	3520	3770	4320

CoroMill 200

Round insert cutters can perform a wide variety of milling operations. The reason for round inserts being the strongest type of insert is that the entering and exit angles varies, as well as the chip cross section varies depending on the depth of cut. This type of cutter has the strongest inserts and is therefore suited for medium and rough machining. This means that it is easy to achieve high productivity with this type of cutter, and inspite of the high metal removal rates round insert cutters often have the best tool-life.

The CoroMill 200 facemilling cutter is a very versatile tool. It is a good choice for general milling as well as for die- and mouldmaking and should be first choice for roughing and medium operations. The application areas are many for the CoroMill 200; facemilling, endmilling, pocketing, contouring and multi-axis milling are applications that the 200 cutter is well suited for. Lets consider the benefits with a round inset cutter for each of these application areas.

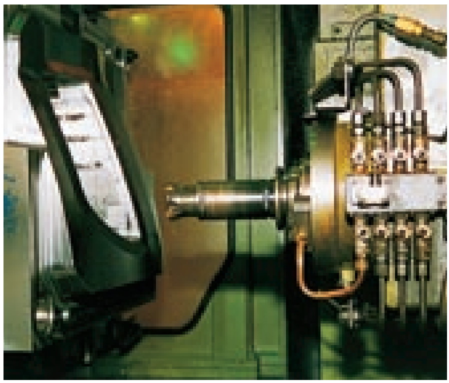


SANDVIK Coromant

Facing
In normal facemilling applications round inserts for the most part provide more effective cutting edges per insert than other insert shapes. Round inserts provide a chip thinning effect which allows higher feed rates compared to 90 or even 45 degree cutters. It should also be the first choice for hardened and heat resistant materials. The combination of curved cutting edge, negative rake and thickness of the insert makes it well suited for milling in such materials.

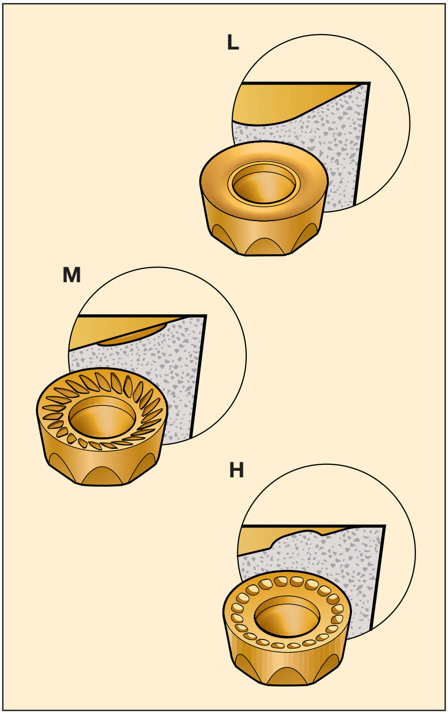
Contouring, pocketing
The CoroMill 200 insert geometries are well suited for the typical stamping die set-up where there are extended overhangs, chatter and recutting of chips. The large cutter body clearance allows ramping and plunging.

Multi-axis machining
The new generation of CNC-machines are often supplied with multi-axis capability, both in programming and set-up. Therefore the need for tools with multi-axis capability. The 200 cutter has more versatility and a wider range of possible movements than any other milling cutter. This makes the cutter ideal for curved surface machining, both concave and convex. Plus the capability to perform circular interpolation in 2- or 3 axes.



Cutter body
The CoroMill 200 has been robustly designed to cope with high metal removal rates and difficult materials in a safe way. The large and open chip pockets manages large amounts of chipflow. Even though the capability of very heavy machining, the safety has not been compromised with. The shim protected insert seat efficiently protects the cutter body in case of a tool overload. The cutters are available in diameters from Ø 25 mm up to Ø 160 mm with coarse, close or extra close differential pitch with insert sizes 10, 12, 16 or 20 mm inserts as standard.

Inserts
Round inserts means strength and ability to machine pockets with high productivity.



These milling cutters have often required powerful machines and provided insufficient accuracy. The development towards positive insert geometries has vastly broadened the application area for round inserts. The extra positive geometry has an advantageous effect on heat development, vibration tendencies and power requirements. These inserts have a sharp cutting edge and positive top rake angle which enables good chip forming ability and a smooth cutting action.

The different geometries provide extra safety for the respective applications.

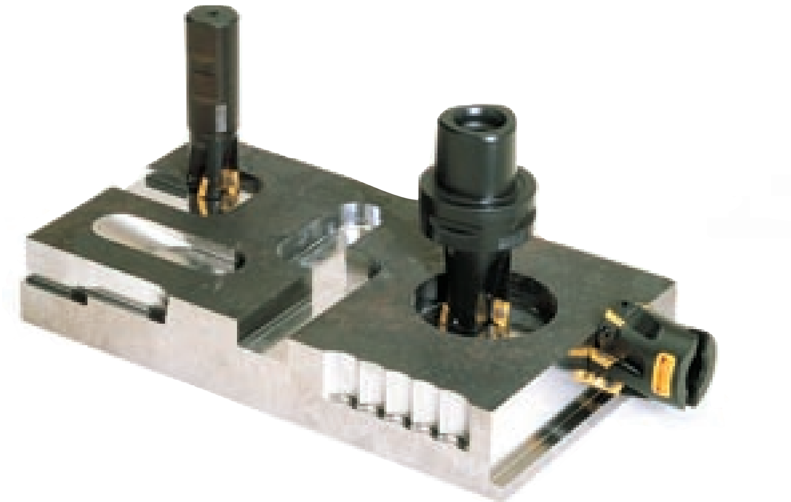
Geometry L for light operations is an extra positive geometry for machining in all materials. The L geometry also creates low cutting forces and can be used for weak workpieces and machine tools, when vibrations occur. Ground RCHT insert for CoroMill 200 cutters are also suitable for finishing operations.

Geometry M for medium operations is the positive geometry for all-round use and basic insert for most materials. Suitable for small to medium size machines and optimized performance in soft and stainless steel. It can also be used in hardened steel with lower feed rate.

Geometry H for heavy operations with reinforced cutting edge for safe production. This insert is reliable in operations when recutting of chips occur and offers highest possible feed rate. Power consumption of the H geometry is slightly higher, approximately, 10% higher compared with geometry M.

The facets which all CoroMill 200 inserts have provide distinctive indexing up to eight times which makes it easy to get a controlled indexing. The centre screw mounting of the inserts facilitates trouble free chip flow and easy mounting.

Fräser-Ø	iC 09*	iC 10	iC 12	iC 13*	iC 16	iC 19*	iC 20
25/25,4* mm	37500	37500	32100	32300	-	-	-
32/31,75* mm	31600	30900	25900	26400	26200	-	-
40/38,1* mm	27900	26500	21900	22800	21400	24600	18400
50/50,8* mm	23200	22900	18800	18600	18000	19100	15000
63/63,5* mm	20300	19900	16200	16100	15300	16100	12500
80/76,2* mm	18200	17300	14000	14400	13100	14200	10600
100/101,6* mm	15500	15300	12300	12200	11400	11800	9200
125/127* mm	13700	13500	10900	10700	10000	10300	8000
160/152,4* mm	12500	11800	9500	9700	8700	9300	6900
200/203,2* mm	10700	10500	8400	8300	7700	7900	6100
250/254* mm	9500	9300	7500	7400	6800	7000	5400



CoroMill 390

CoroMill 390 is a high positive endmilling, square shoulder and facemilling concept that produces, among other things, true 90 degree component shoulders. The concept has been developed to meet customer demands on all endmilling operations within both the roughing and finishing areas. The main focus being on precision, component quality, productivity, versatility and easy handling.

The CoroMill 390 cutter bodies are produced from hardened steel with all machining carried out after the hardening process. This method provides the following advantages;

- no increase in insert run-out from local hardening
- increased fatigue strength

This in its turn lead to

- Highest precision; longer tool life and better surface quality and dimensionel tolerances
- Longer life of the cutter body

One of the CoroMill 390 features is related to the inserts main cutting edge which is not straight. The main cutting edges has a helical shape in the axial direction and a convex configuration in the radial. Each cutter diameter has optimised axial and radial rake angles to ensure 90 degree shoulder is produced. From this design the concept produces a shoulder in the workpiece with minimum mismatch even after several axial passes.

The cutter comes with two insert sizes 11 mm and 17 mm. The 11 mm insert cutter are available in diameters ranging from 16 mm to 63 mm and the cutters with 17 mm inserts have a diameter range of 25 to 80 mm. Four insert geometries are available for the 390 cutter: E-L is a precision ground geometry for light operations and the L-geometry, also for light operations. The M-geometry is for general purpose operations and the H-geometry for demanding applications.

The maximum recommended rpm and cutting speed vc should be taken in consideration when using the cutter, as shown in the table.

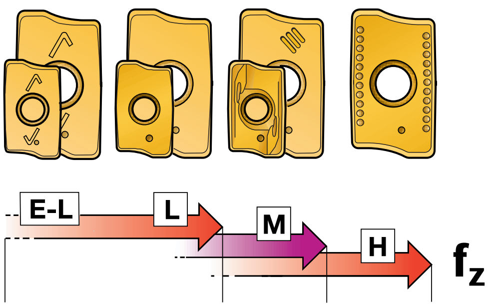
SANDVIK Coromant

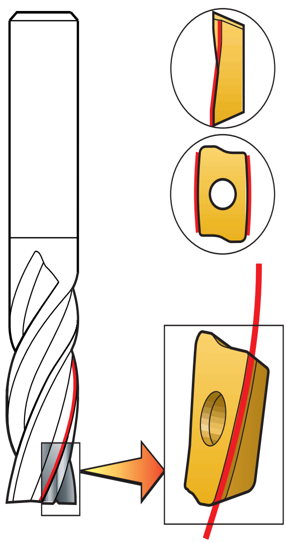
Maximum RPM Coromant Capto

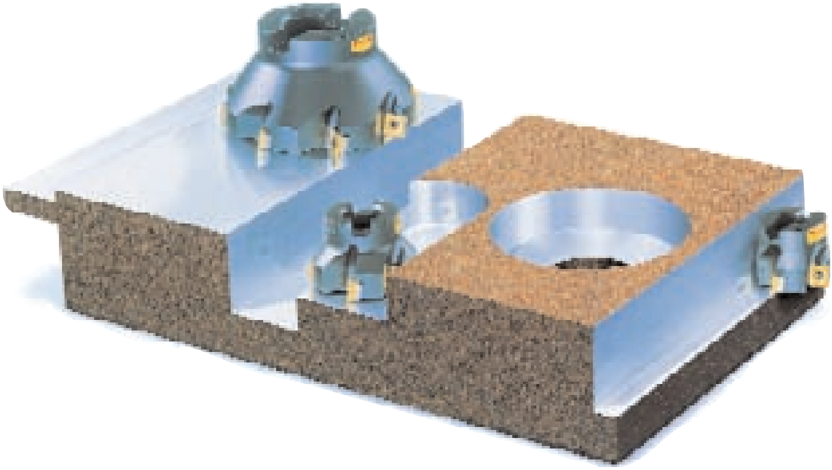
Diameter/Coromant Capto size						
	16	20	25	32	32	40
	C4	C4	C4	C4	C5	C4/C5
Max RPM	39 000	34600	36500	31000	28000	27000
Max V_c m/min	1959	2173	2865	3115	2813	3391

Square shoulder facemills

	Size - 11 insert				Size - 17 insert			
	40	50	63	80	40	50	63	80
Max RPM	27000	23700	20700	18200	21900	19000	16500	14400
Max V_c m/min	3391	3721	4095	4572	2751	2983	3264	3617







CoroMill 290

Traditionally square shoulder milling has been performed by endmills or dedicated square shoulder cutters. The reasons for that has varied. One of them has been that facemilling cutters were not able to machine 90° corners, which is often the case when square shoulder milling.

The complete CoroMill 290 programme covers the majority of square shoulder applications in many modern machining centres. The cutter is available in a wide programme, comprising coarse and close as well as extra-close pitched cutters in insert size, 12 mm.

The main characteristic of Coromill 290 is the advanced insert geometry and inserts with four cutting edges mounted in shim protected pockets offering highest possible security, combined with low cutting forces.

Inserts with advanced chip forming geometries have the ideal balance between low cutting forces and high edge security.

- M is the positive all-round geometry, covering the majority of square shoulder applications under varying conditions.

- L is the light cutting geometry, designed for machining with low powered machines, fragile workpieces, unstable clamping and for materials requiring a sharper cutting edge.

- H is the tough geometry for highest possible edge security. Inserts with large parallel land perform exellent surface finish even at high feed rates. Inserts with 2.0 mm corner radius give the highest edge security in roughing or medium operations, or when machining under unstable conditions.

These inserts not only provide a sharper, positive cutting edge, they also have chip-forming ability and have carefully designed transitions between various parts of the geometry, adding micro strength to the cutting edge. The design of the chipformer reduces contact between chip and edge and curls the chip, making it more manageable and carrying away more heat from the cutting area.

All the inserts have a 1.6 or 2.1 mm parallel land which increases the capability of producing better surface finish compared to an insert with only corner radius. To obtain the best surface finish the cutting data should be selected, whenever possible, to give a feed per revolution (f_n) less than the length of the parallel land.
A rule of thumb is to use 70% of the parallel land in feed per revolution.

Maximum recommended rpm

Diameter mm	290-12
40	21600
50	18400
63	15900
80	13700
100	12000
125	10600
160	9250
200	8200
250	7300

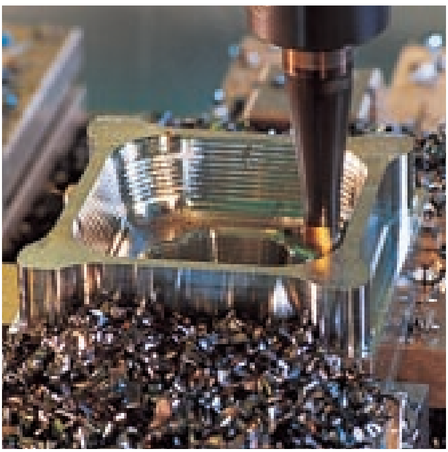
BALL NOSE ENDMILL R216

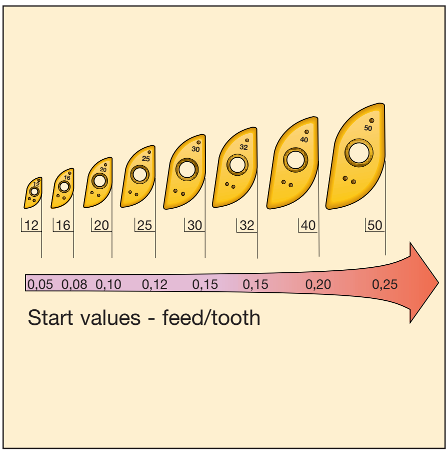
The ball nose endmill 216 is a strong and robust endmill concept. Each cutter body has two effective inserts and each diameter uses the same insert type in both the centre and periphery insert seats. The insert design gives two cutting edges per insert and the concept is aimed at roughing and semi-finishing of curved surfaces in machining centres, CNC milling machines and copy milling machines for die and mould making, aerospace, automotive to name a few. It is possible to feed the tool in all directions.

There are individual insert sizes for each cutter diameter and the insert design permits use of the same insert type in the periphery as well as in the centre insert pocket.

Cutter body and insert details

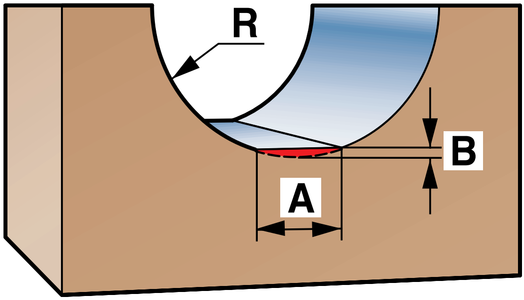
Diameter D_3	**12**	**16**	**20**	**25**	**30**	**32**	**40**	**50**
Max. depth of cut, a_p	10,8	14,4	17,9	22,3	26,9	28,6	35	44
Protection insert a_p	-	-	-	-	42	-	65	74
Radial rake angle	0	0	0	0	0	0	0	0
Axial rake angle	-10	-10	-10	-10	-10	-10	-10	-10

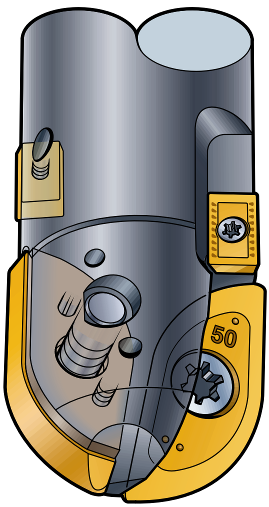




The unique insert geometry makes it possible to drill with the ball nose cutter. However, deviations on the workpiece may occur at the centre of the cutter.

R	A	B
6	1.4	0.07
8	1.7	0.09
10	2.2	0.12
12,5	3.0	0.18
15	3.9	0.20
16	3.5	0.22
20	3.6	0.24
25	3.8	0.26





The larger diameters of the cutter are also available with a shank protection insert for performing heavy duty copy milling. Diameter 30 mm with one insert and diameters 40 and 50 mm with two inserts.

SOLID CARBIDE ENDMILLS

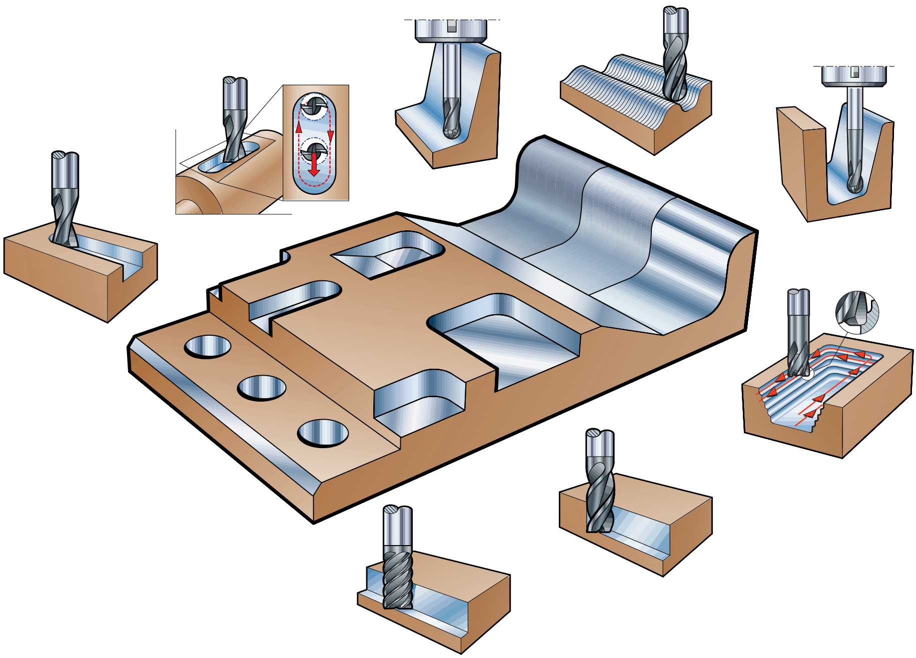
Especially in finishing the demand is high for small high performing solid endmills, both 90 degree and ball nose. Well in line with the modern machining demands there are several different solid carbide endmills for both general and HSM machining available, in several different grades. Some of the cutters are specially designed for HSM and for machining of hardened material up to 63 Hrc.

1020 (TiC,N) is the allround grade most suitable for general endmilling in most materials. It is reliable and the wear process is even and predictable.

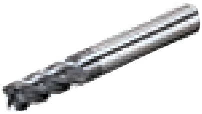
1010 (TiAlN) is very suitable for machining of hardened tool steel. The high heat resistance capability makes it suitable for dry machining and allows short machining times and long toollife, demands when machining big moulds. For best use of grade 1010 special geometries are required.

The substrate for both grades is H10F, a micro grain grade. The toughness behaviour is very good, which is important in order to avoid tool breakage and chipping, particularly on sensitive corners. Most of the multi fluted endmills have strengthened corner geometries for maximum security.

On some of the 1010 cutters there are helix reduction over the corners and this is to strengthen the cutting edge and prevent chipping especially when machining hardened material. The helix reduction also allows tougher cutting data to be used.



OVERVIEW OF TOOLS AND OPERATION GUIDELINES

Recommended endmill	Application
R215.36L 	**Finishing - sidemilling** $a_e < 0.05 - 0.1 \times D_c$ 60 degree helix and six cutting edges. Radial depth of cut max 10% of the diameter D_c. Non drilling version
R215.3x-H 	30 degree helix and the number of edges equal to the diameter in mm. Radial depth of cut max 5% of the diameter. Non drilling version.
R216.34-N R216.33-N 	**Semi - sidemilling** $a_e < 0.25 \times D_c$ Three and four edge endmills with 30- and 45 degree helix.
R216.33-N R216.34-N 	Recommended for side milling in mixed production. Choose 45 degree helix for smoothest action and best surface finish Two edge version recommended for long chipping material. All versions with drilling capability.

R216.32-N

R216.32-N

R216.33-N

R216.33-N

R216.33-P

R216.34-N

R216.34-N

R216.42-N

Roughing - slotting a_p and $a_e < 0.5 \times D_c$

Two and three edge short version provide good stability lowest deflection - also undersized tools are available for key slot milling.

For deep slots two edge versions are recommended or alternatively create several passes.

45° degree helix for smooth action and to help chip evacuation.

Recommended for stainles steel.

For heavier copy milling, two edge ball nose endmills are recommended.

All versions have drilling capability.

R216.44-L

R216.42-L

R216.42-L

R216.42-L
R216.42-L

R216.42-L
R216.42-L

R216.42-L
R216.42-L

Copy milling/High speed milling (HSM)

Two and four edge ball nose endmills for finish milling of deep dies and moulds.

Recommended for all materials.

Two and four edge spherical ball nose endmills for steep die cavities providing accessibility and a reduced contact length.

Recommended for depth of cut in finishing - a_p/a_e = 0.1- 0.2 mm.

Preferably use downmilling and pulling feed direction.

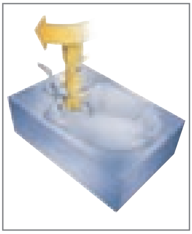
Ball nose and radius versions with rigid shanks, short cutting length and optimised geometry for reliable semi-finishing at high speeds (HSM).

All tools are centre cutting, however, not recommended for drilling. Recommended operations are circular interpolation and light ramping.

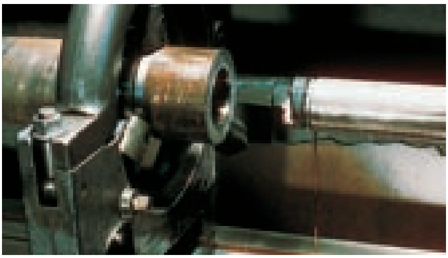
Tools are designed for dry milling.

Always use tool holders with as low radial runout as possible; CoroGrip, shrink fit, Coromant Capto, precision ground collet chucks, precision chucks. The radial runout should not exceed 10 µm. Use the strongest possible tools with high stiffness and big core diameter. Try to minimise the tool overhang.

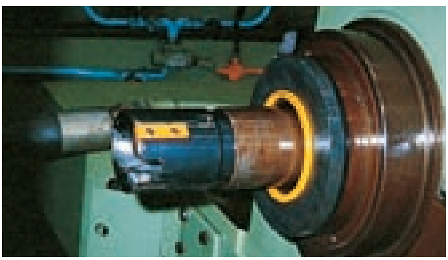
DRILLING TOOLS FOR DIES AND MOULDS



In many dies and moulds there are several holes to be machined such as; hole for location pins, - mounting screws, threaded holes and holes for cooling and warming up. To successfully machine these holes the right kind of drill and drilling method has to be applied, ranging from twisted HSS drills to STS and Ejector drills for deep hole drilling.



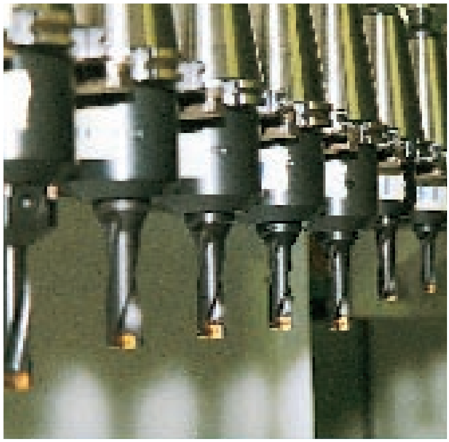
Ejector



STS

Short hole drilling
Generally, short hole drilling is referred to when machining holes with a relatively small hole-depth and drill-diameter ratio. At drill diameters up to 30 mm the hole depth can be 5-6 x D, while up to 60 mm it is 4 x D and above 2,5 x D usually is the limitation. Todays drills can keep close tolerances, down to IT-10.

There are some different alternatives to choose from when deciding what type of drill that should be used for an application; HSS drills, HSS drills tipped with cemented carbide, solid carbide drills, twisted or straight flute drills with indexable inserts.

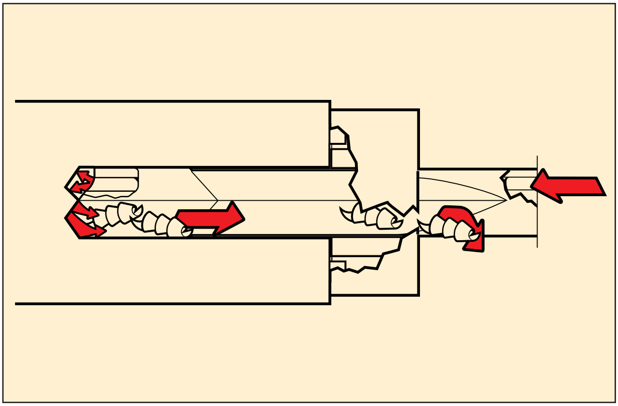


When choosing the type of drill, the first question is whether to use a regrindable- or indexable insert drill? Since indexable insert drills do not cover the smallest diameters it is only regrindable drills which can be used for small hole applications. Under all circumstances it is good to start looking at the hole diameter and required length of the hole, to narrow down the selection of tools.

Small hole diameters
Regrindable drills with Delta geometry is available in two different versions overlapping one another; Delta, and Delta C. In the diameter range covered by Delta C, workpiece material in relation to machine tool capacity will be the determining factor regarding which drill to use.

If the machine tool capacity is good and the possibility to use high spindle speed is available the qualities of the cemented carbide at the Delta C drill should be utilised. However, if the spindle speed is not high enough the capacity of the Delta C drill can not be utilised because cemented Carbide needs sufficient cutting speed to work at its best.

SANDVIK Coromant



Medium sized hole diameters
The diameter range considered as medium sized is where the Coromant U-drill and Delta drill overlap each other. At high demands on close tolerances or when the depth of the hole limits the use of the U-drill (indexable insert drill), the Delta drill is the only alternative.

On the other hand, when the initial penetration surface is not flat, if the hole is pre-drilled, or if cross-drilling must be done, then the indexable insert drill is the only option. These drills will provide the lowest cost per machined component, since they have exchangeable inserts. This cost advantage is particularly beneficial when machining large series.

We will not discuss larger short hole drills such as trepanning drills as these sizes of holes can easily be made by circular interpolation with a round insert milling cutter or an endmill. Using a milling cutter to machine the holes within the medium sized diameter is also an alternative which should be taken in consideration when choosing machining method.

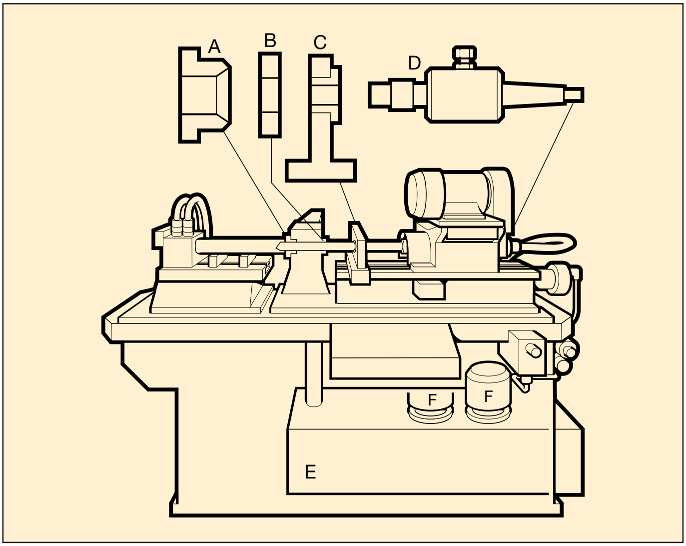
Deep hole drilling

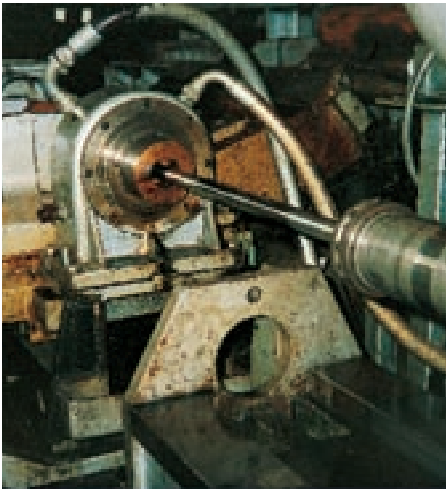
Deep hole drilling is something which is a relatively common operation in die and mould making, for instance to machine holes for cooling or warming of the mould. There are some different ways to drill deep holes and the most common for die and mould making are gundrilling, STS and ejector drilling.

In the gun drilling system, the cutting fluid is supplied internally through the tool and the chips are removed through a V-shaped groove in the shaft. Conventional machines with sufficiently high spindle speed, coolant supply (flow and pressure) and a suitable range of feeds can be modified for gun drilling. However, the best results are obtained with special gun drilling machines.

These should be equipped with a drill bushing (A), a splash guard (B), an oil supply unit (D) and, generally, it is necessary to support the workpiece with a steady rest C. In addition, it is necessary to have a tank which provides efficient filtering and cooling (E), plus a pump (F) with sufficient capacity in terms of both pressure and volume.

The gun drill is the only drill which can be used with the gun drilling system. Gun drilling is primarily used when precision-drilling of small holes has to be performed. It can cope with hole depths up to 100 x the drill diameter. If additional steady rests are used, it is possible to drill up to 200 x the drill diameter, provided the capacity of the pump is sufficient.



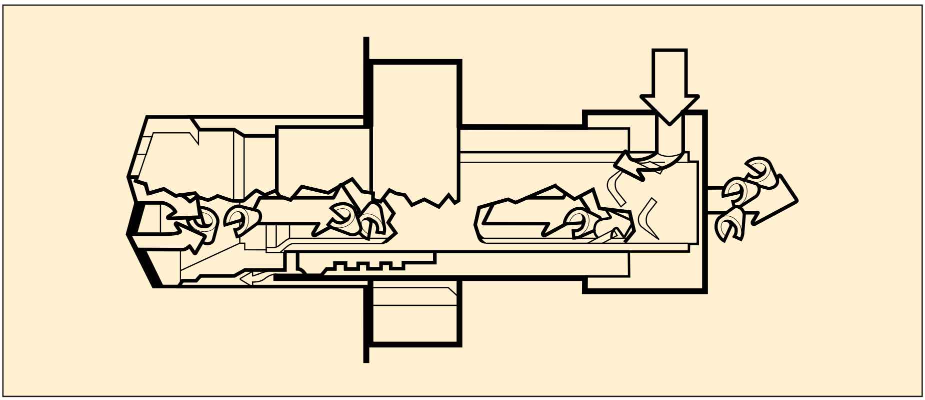


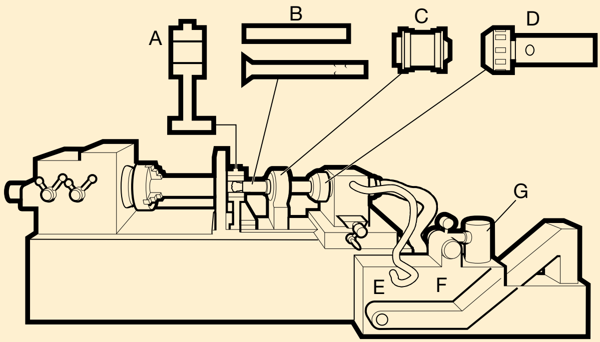
One clear advantage with gun drilling is the high quality of the hole achieved, which is crucial for cooling channels in a tool. If the quality of the cooling channels is not good enough the channels can easily be put out of order due to corrosion.

It is also important that the tolerance of the pre-drilled guidance hole is kept close to get a good result with the gun drill.

The Ejector system has twin drill tubes. The cutting fluid is pumped in between the inner and the outer tubes. The major portion of the cutting fluid is led forward to the drill head while the remainder is forced through a groove in the rear section of the inner tube. The negative pressure which arises in the front section of the inner tube means that the cutting fluid at the drill head is sucked out through the inner tube together with the chips.
The Ejector system is self-contained and can be adapted quite easily for most conventional machines, machining centres, NC and CNC lathes. The machine should be equipped with a drill bushing (A), inner and outer tubes (B), a vibration damper C plus a connector with a collet and a sealing sleeve (D). In addition, it is necessary to have a tank with efficient filtering (E) and cooling (F), plus a pump (G) with sufficient capacity in terms of both pressure and volume.

The drill head is screwed firmly onto the outer tube, which has a four-start square thread. If the support pads are not in contact with the drill bushing another thread start can be used. The thread starts have a 4 x 90 degree positioning.

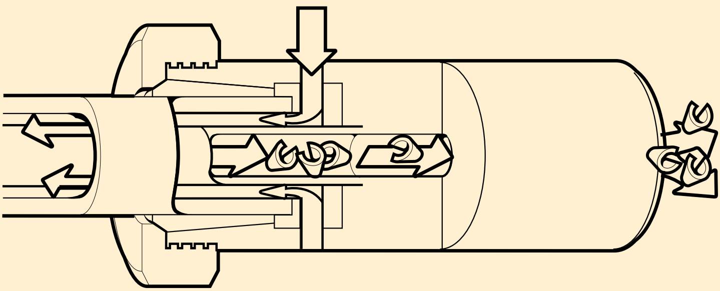




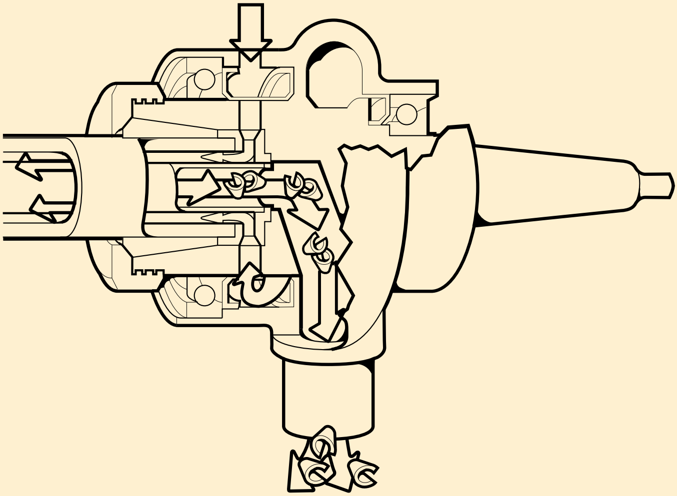
Connectors are available for both rotating and non-rotating drills. The connector for a non-rotating drill (1), which is the most common application, has no rotation parts. For rotation drills, a connector is used with a mounted and sealed housing around the spindle (2). The outer and inner tubes are attached to the connector by a collet. A sealing sleeve is also used here at the entrance for the cutting fluid. The collet and the sealing sleeve must be changed for different diameter ranges.

Because the cutting fluid is pumped into the space between the inner and outer tubes, no seals are required between the workpiece and the drill bushing. Therefore, the system is often used with workpieces where

1



2



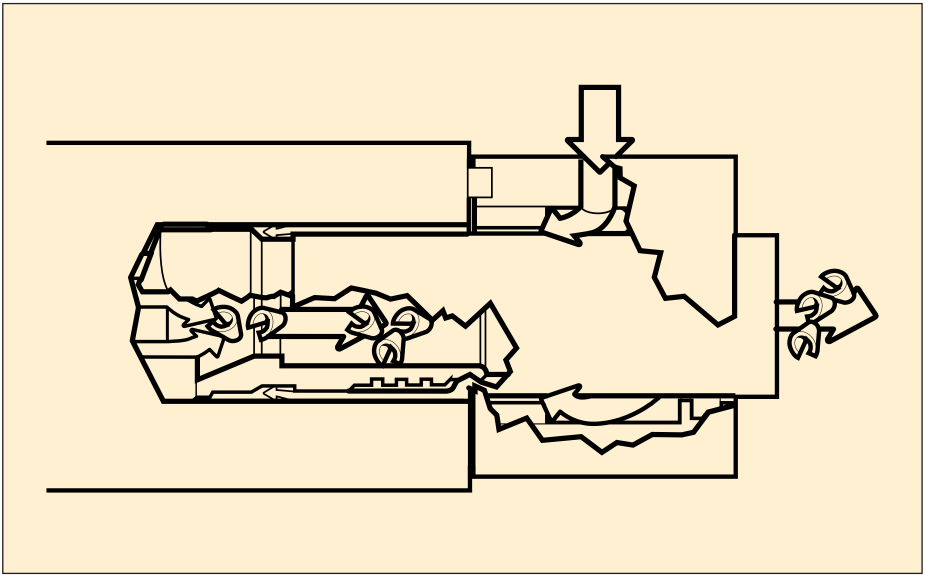
sealing problem could arise. This feature also makes the Ejector system very suitable for use with interrupted drilling.

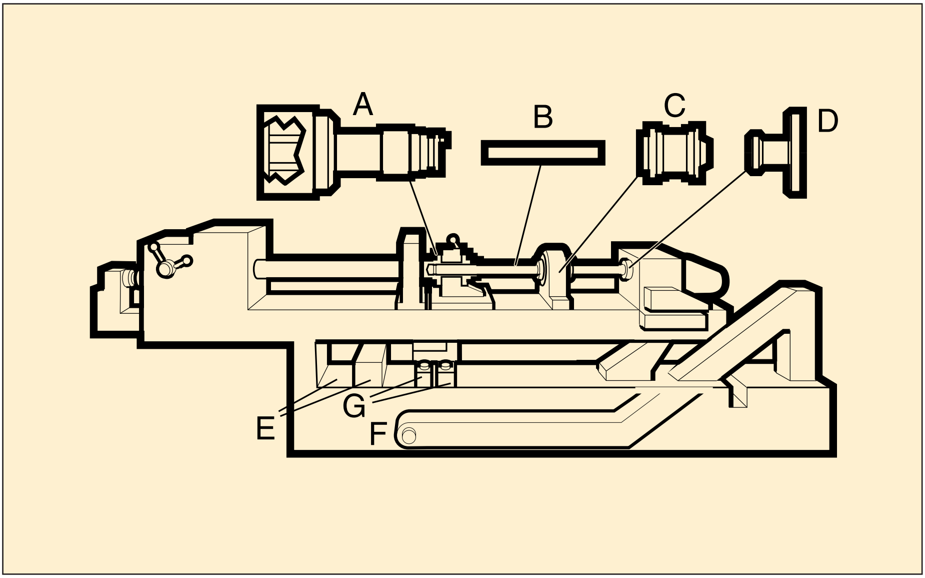
The Ejector system can cope with hole depths of up to 100 x the hole diameter when drilling horizontally and approximately 50 x the hole diameter when drilling vertically.

In the **STS** drilling system, the cutting fluid is forced in between the drill tube and the walls of the hole. The cutting fluid should have sufficient pressure and flow to remove the chips efficiently through the tool and drill tube.

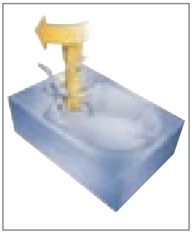
Only special deep hole drilling machines are used for STS drilling. The machine should be equipped with an oil pressure head with a drill bushing and seals (A), a drill tube (B), a vibration damper C and a connecting chuck (D). In addition, it is necessary to have a tank which gives efficient filtering (E) and cooling (F), plus a pump (G) with sufficient capacity in terms of both pressure and volume.

The high cutting fluid pressure and flow makes the system suitable for use where material with poor chip breaking capacity is to be machined. Hole depths of up to 100 x the hole diameter can be drilled with STS.




A
B
C
D
E
G
F

CoroGrip PRECISION-POWER CHUCK

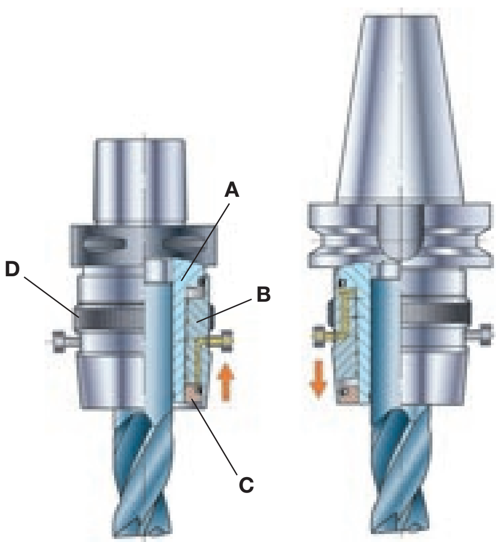


CoroGrip is a holding system that covers all applications from super-finishing to heavy roughing. One holder can clamp all types of tools from facemills to drills with both cylindrical, Whistle notch or Weldon shanks. Standard collets such as collets for HydroGrip, BIG, Nikken, NT etc., can also be used 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 (<40 000 rpm), 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.



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

It takes less than 20 seconds to change a tool.

Two different pumps can be used when changing tools in a CoroGrip holder; one manual hydraulic pump that is portable and one stationary air-driven pump, which is activated by a foot-pedal.

To connect the pump to the holder an ergonomically designed handle is used. Clamping or unclamping the tool is done by a switch without removing the handle from the holder. When using the manual pump one hand is always free for holding the tool and when using the air-driven pump both hands are free.

The same handle is used for all types and sizes of CoroGrip holders.

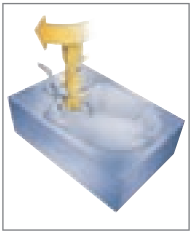
The picture shows how easy it is to pre-set the tool with the CoroGrip concept. Without any setting screw, you can very easy adjust the length of the tool to a rquired measure. Depending on the accuracy of the optical reader, you can set the tool within +/- 3-5 microns.

If the pneumatic pump is used both hands are free to hold and adjust the tool. When it is in exactly right position the foot-pedal is pressed and the tool is fixed. Only the outer sleeve is moving when clamping and unclamping and there are no forces on the pre-setting fixture. This means that all tool changes can be made in the pre-setter and there is no need of a rigid mounting fixture, which is the case if power chucks or outer holders where keys are used for clamping.

Transmission torque comparison [Nm]

Shank diameter	[Torque Nm]	Type of holder
12 mm	72 50 93	Shrink fit HydroGrip **CoroGrip**
20 mm	243 181 440	Shrink fit HydroGrip **CoroGrip**
25 mm	421 365 804	Shrink fit HydroGrip **CoroGrip**
32 mm	--- 651 1512	Shrink fit HydroGrip **CoroGrip**

COROMANT CAPTO



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.

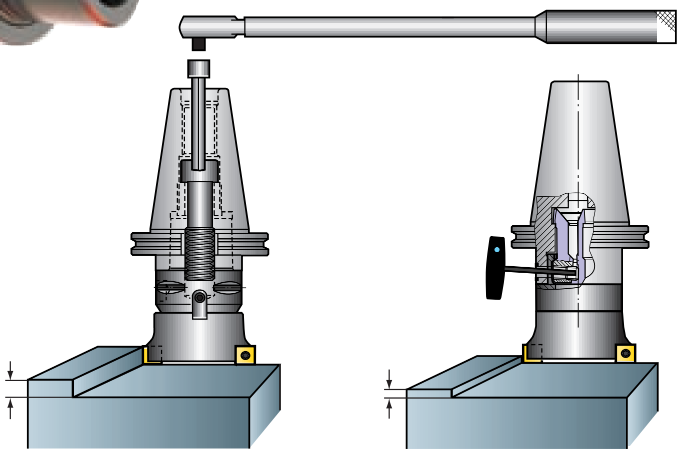
- With modular tools it is always possible to use optimal tool assemblies and cutting data for best productivity!

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.

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 is 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.

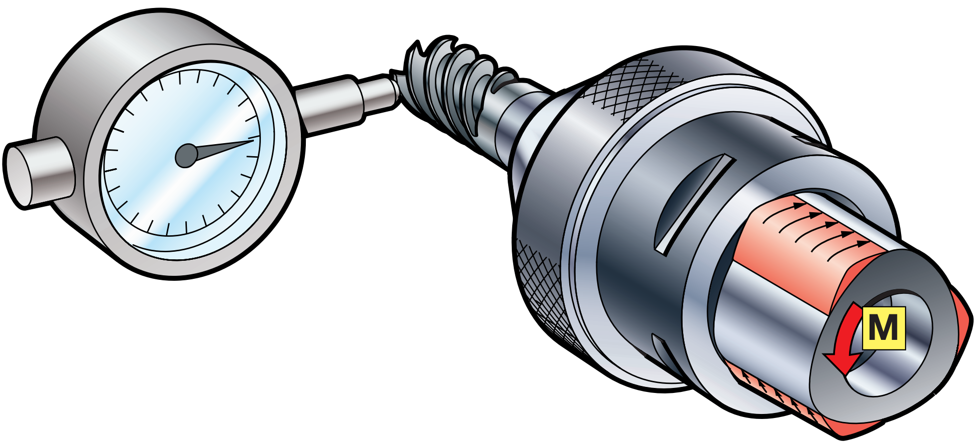


HSK is a spindle interface and not a modular coupling and should not be mistaken for a modular tool system. The clamping force is not as great as that produced by Coromant Capto with centre bolt clamping, which is mainly required for milling and boring operations. HSK extensions can, however, be used for light drilling and reaming operations, where there are low radial forces.

Minimum run-out and guaranteed centre height
Press fit and torque loads spread symmetrically around the coupling, without load peaks, are the reasons for the outstanding minimum run-out and centre-height characteristics.

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 any of the following situations:

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

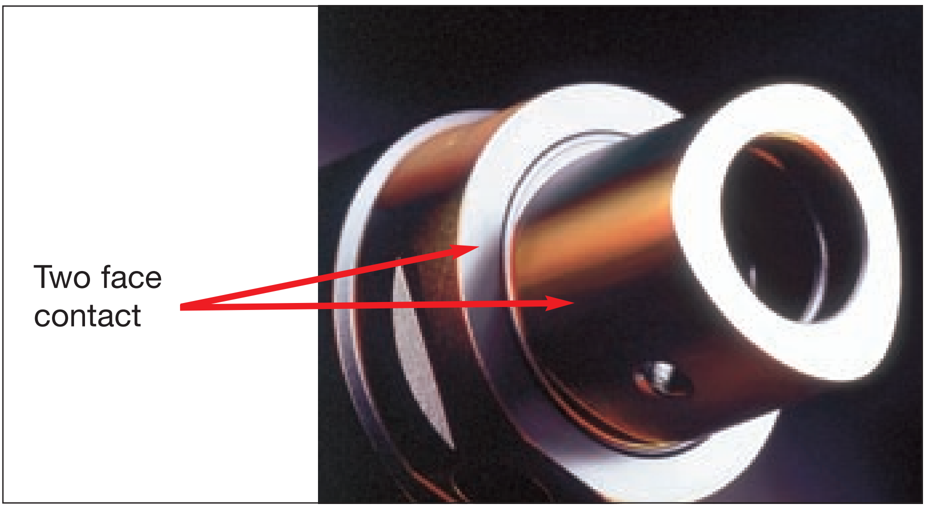


The coupling has the best stability characteristics on the market, as a result of:
No pins, keys etc. The polygon shape transmits torque (T) without any loose parts such as pins or keys.

No play in the coupling. The tight press fit guarantees that there is no play in the coupling.

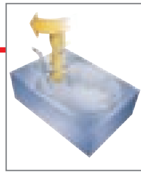
Symmetrical loads. Torque load is spread symmetrically on the polygon without peaks irrespective of rotation. Thereby the coupling is self-centring. It also guarantees long life of the coupling.

Two face contact/high clamping force. 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.



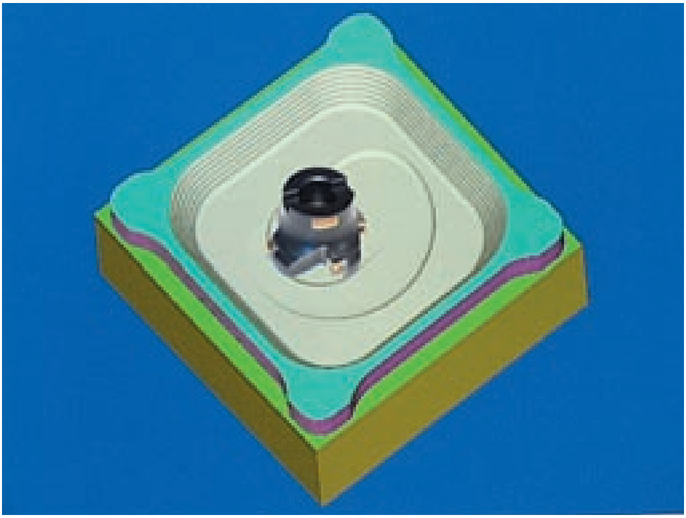
Comparison between holders for clamping of shaft tools

	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 roug-hing-finishing	Finishing	Heavy roug-hing-finishing	Heavy roug-hing -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 speed	+	+	++	++	+++	+++
Maintenance	None required	Cleaning and changing collets	Cleaning and changing spare parts	Non required	None required	None required
Possibility to use collets	No	Yes	Yes	Yes	No	Yes

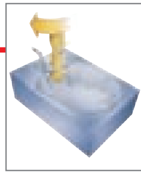


ROUGHING RECOMMENDED METHOD

Roughing of cavity; downmilling, leaves 1 mm in stock

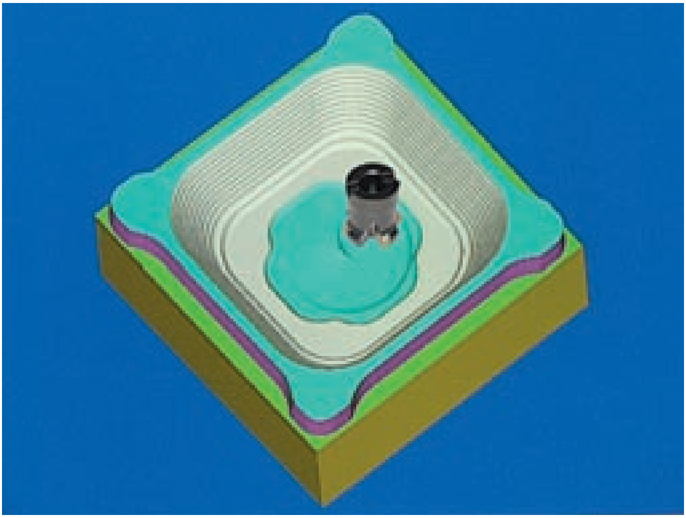


Cutting tool	CoroMill R200-068Q27-12L
Insert	RCKT 1204MO-PM GC4030
Cutter diameter [ø]	80 MM
Number of teeth [*z*]	4
Depth of cut [a_p]	4 mm
Depth of cut [a_e]	65% x D_c
Cutting speed [V_c]	250 m/min
Spindle speed [*n*]	995 rpm
Feed per tooth [f_z]	0.4 mm/tooth
Table feed [V_f]	1592 mm/min
Time [min, sec]	6 00

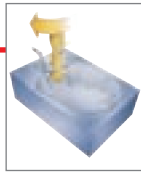


ROUGHING OF CAVITY RECOMMENDED METHOD

3 axis ramping, contouring, downmilling, leaves 1 mm in stock

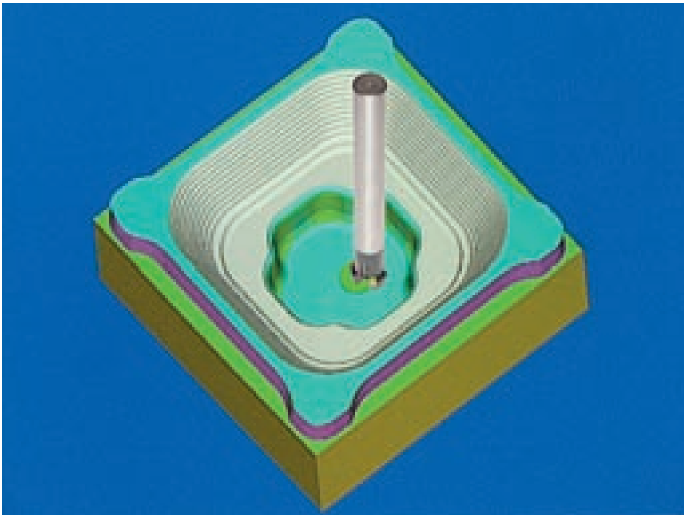


Cutting tool	CoroMill R200-040Q22-12M
Insert	RCKT 1204MO-PM GC4030
Cutter diameter [ø]	52 MM
Number of teeth [z]	4
Depth of cut [a_p]	4 mm
Depth of cut [a_e]	65% x D_c
Cutting speed [V_c]	180 m/min
Spindle speed [n]	1102 rpm
Feed per tooth [f_z]	0.33 mm/tooth
Table feed [V_f]	1469 mm/min
Time [min, sec]	1 57



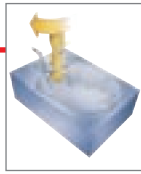
ROUGHING OF CAVITY RECOMMENDED METHOD

Restmilling



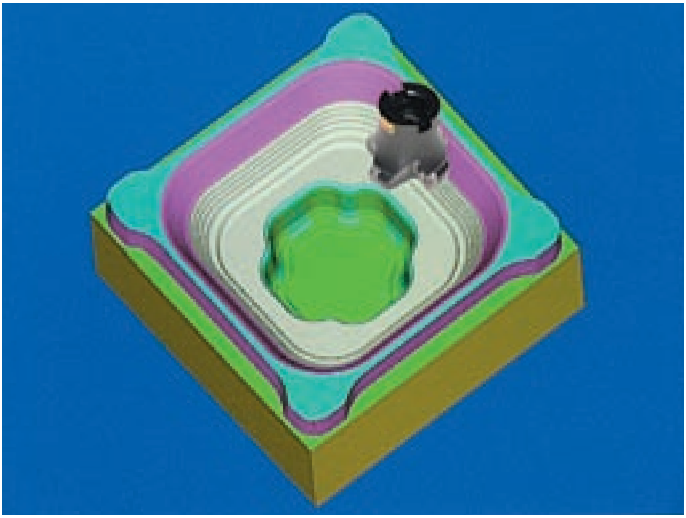
Cutting tool	CoroMill R200-020A25-12H
Insert	RCKT 1204MO-WM CT530
Cutter diameter [ø]	32 mm
Number of teeth [*z*]	3
Depth of cut [a_p]	2-5 mm
Depth of cut [a_e]	<5 mm
Cutting speed [V_c]	500 m/min
Spindle speed [*n*]	4974 rpm
Feed per tooth [f_z]	0.15 mm/tooth
Table feed [V_f]	2238 mm/min
Time [min, sec]	1 35

SANDVIK Coromant

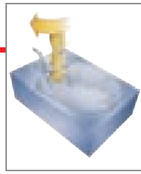


SEMI-FINISHING RECOMMENDED METHOD

Contouring, downmilling, leaving 0.2 mm in stock

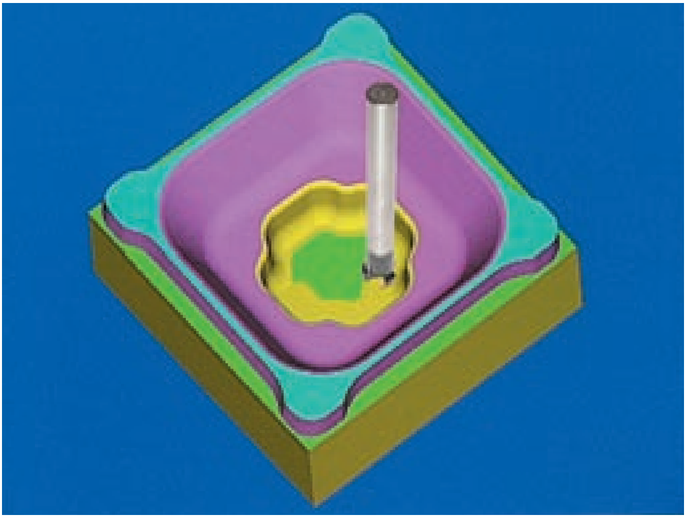


Cutting tool	CoroMill R200-051Q22-12H
Insert	RCKT 1204MO-WM CT530
Cutter diameter [ø]	63 mm
Number of teeth [*z*]	5
Depth of cut [a_p]	1 mm
Depth of cut [a_e]	0.8 mm
Cutting speed [V_c]	600 m/min
Spindle speed [*n*]	3032 rpm
Feed per tooth [f_z]	0.36 mm/tooth
Table feed [V_f]	5457 mm/min
Time [min, sec]	7 30

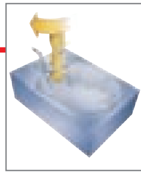


SEMI-FINISHING RECOMMENDED METHOD

Contouring, downmilling, leaving 0.2 mm in stock

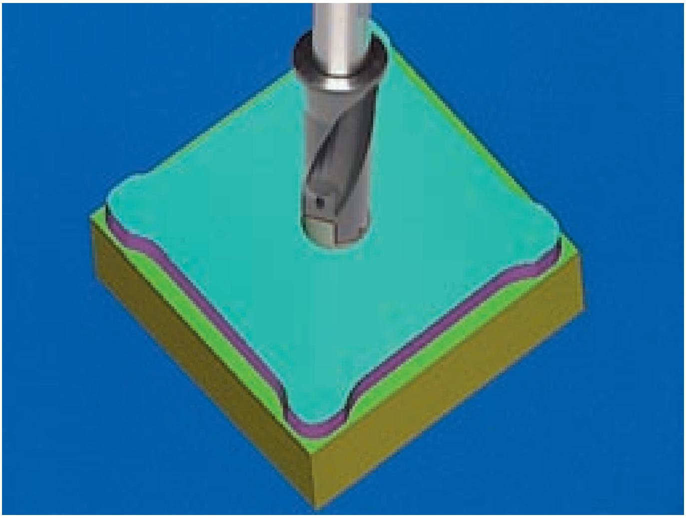


Cutting tool	CoroMill R200-020A25-12H
Insert	RCKT 1204MO-WM CT530
Cutter diameter [ø]	32 mm
Number of teeth [*z*]	3
Depth of cut [a_p]	1 mm
Depth of cut [a_e]	0.8 mm
Cutting speed [V_c]	500 m/min
Spindle speed [*n*]	4974 rpm
Feed per tooth [f_z]	0.16 mm/tooth
Table feed [V_f]	2400 mm/min
Time [min, sec]	1 19

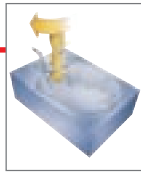


ROUGHING
COMMON METHOD

Roughing of cavity; drilling starting hole

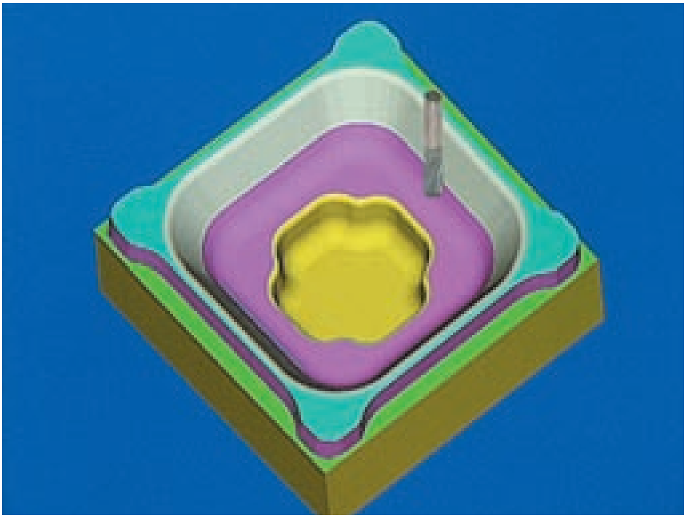


Cutting tool	Coromant U-drill R416.2-0500L40-21
Insert	WCMX 080412R-53 GC4025 (P) WCMX 080412R-53 GC 1020 (C)
Drill diameter [ø]	50 mm
Number of teeth [*z*]	2
Depth of cut [a_p]	84 mm
Depth of cut [a_e]	~ D_c mm
Cutting speed [V_c]	150 m/min
Spindle speed [*n*]	955 rpm
Feed per tooth [f_z]	0.15 mm/tooth
Table feed [V_f]	143 mm/min
Time [min, sec]	0 35

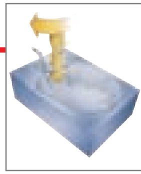


ROUGHING
COMMON METHOD

Roughing of cavity; contouring, constant Z-value, downmilling, leaving 1 mm in stock

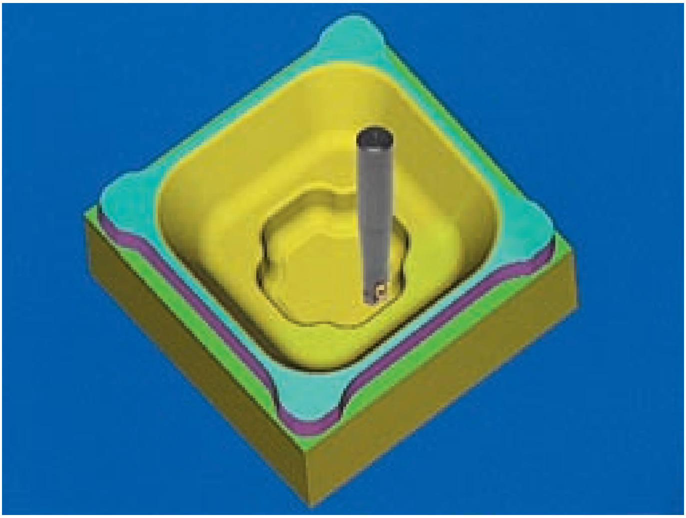


Cutting tool	CoroMill R290-050Q27-12M
Insert	R-290.90-12T320M-PM GC4030
Cutter diameter [ø]	50 mm
Number of teeth [*z*]	4
Depth of cut [a_p]	5 mm
Depth of cut [a_e]	65 % x D_c
Cutting speed [V_c]	200 m/min
Spindle speed [*n*]	1273 rpm
Feed per tooth [f_z]	0.25 mm/tooth
Table feed [V_f]	1273 mm/min
Time [min, sec]	7 43

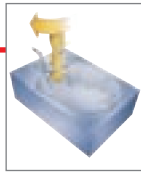


ROUGHING
COMMON METHOD

Roughing of cavity; contouring, downmilling, constant Z-value, leaving 1 mm in stock

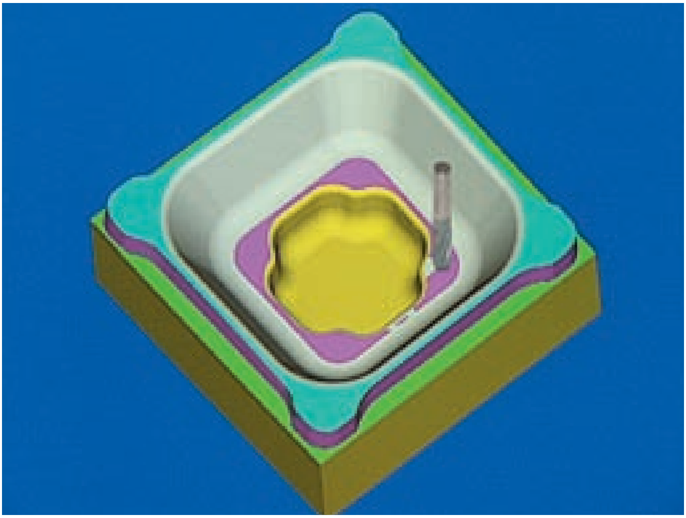


Cutting tool	Ballnose endmill R216-32B32-070
Insert	R-216-32 06 M-M GC1025
Cutter diameter [ø]	32 mm
Number of teeth [*z*]	2
Depth of cut [a_p]	6 mm
Depth of cut [a_e]	6 mm
Cutting speed [V_c]	200 m/min
Spindle speed [*n*]	1989 rpm
Feed per tooth [f_z]	0.25 mm/tooth
Table feed [V_f]	995 mm/min
Time [min, sec]	6 00

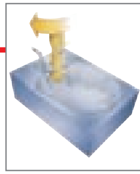


ROUGHING COMMON METHOD

Roughing of cavity; restmilling, copymilling, up- and downmilling (zigzag), leaving 1 mm in stock

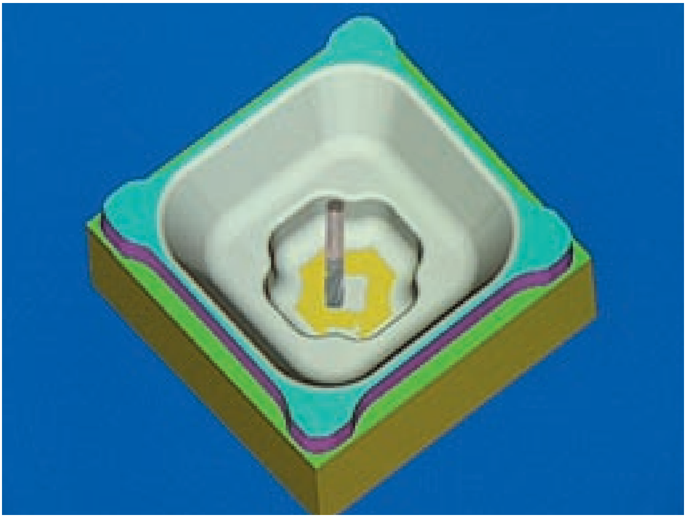


Cutting tool	Ballnose endmill R216-25B325-60
Insert	R-216-25 04 M-M GC1025
Cutter diameter [ø]	25 mm
Number of teeth [*z*]	2
Depth of cut [a_p]	1 mm
Depth of cut [a_e]	1 mm
Cutting speed [V_c]	300 m/min
Spindle speed [*n*]	3820 rpm
Feed per tooth [f_z]	0.20 mm/tooth
Table feed [V_f]	1528 mm/min
Time [min, sec]	3 35



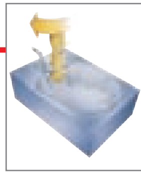
SEMI-FINISHING COMMON METHOD

Contouring, downmilling, leaving 0.2 mm in stock



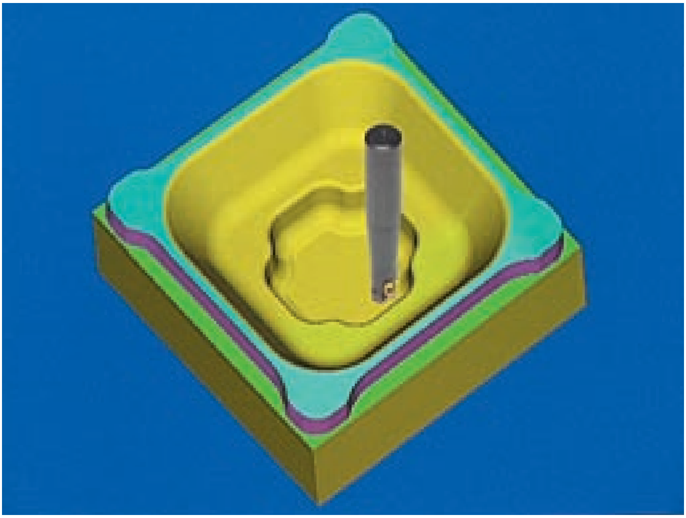
Cutting tool	Ballnose endmill R216-25A25-055
Insert	R-216-20 T3 M-M CT530
Cutter diameter [ø]	20 mm
Number of teeth [*z*]	2
Depth of cut [a_p]	1.5 mm
Depth of cut [a_e]	0.8 mm
Cutting speed [V_c]	400 m/min
Spindle speed [*n*]	6367 rpm
Feed per tooth [f_z]	0.30 mm/tooth
Table feed [V_f]	3820 mm/min
Time [min, sec]	7 46

SANDVIK Coromant

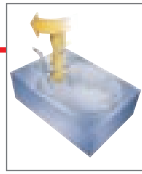


SEMI-FINISHING-
COMMON METHOD

Contouring, downmilling, leaving 0.2 mm in stock

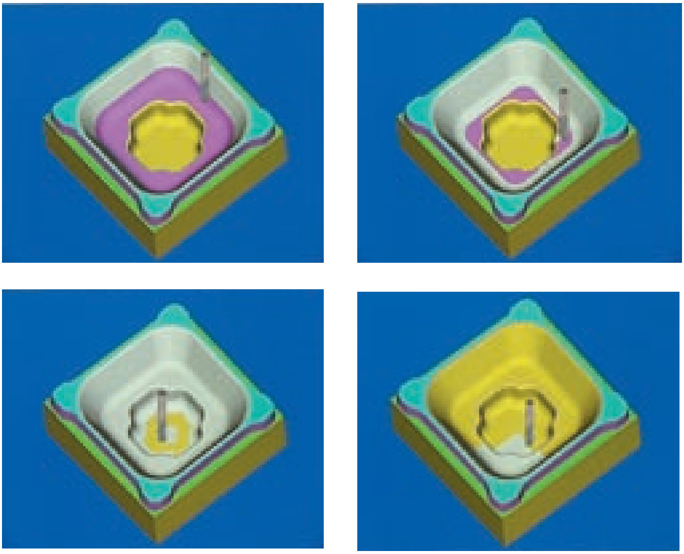


Cutting tool	Ballnose endmill R216-20A25-055
Insert	R216-20 T3 M-M CT530
Cutter diameter [ø]	20 mm
Number of teeth [*z*]	2
Depth of cut [a_p]	1.5 mm
Depth of cut [a_e]	0.8 mm
Cutting speed [V_c]	400 m/min
Spindle speed [*n*]	6367 rpm
Feed per tooth [f_z]	0.30 mm/tooth
Table feed [V_f]	3820 mm/min
Time [min, sec]	6 44



FINISHING BOTH METHODS

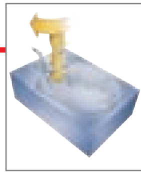
Contouring, up- and downmilling



Cutting tool	Carbide endmill R216.64-12030-AK 13L GC1010
Insert	-------------
Cutter diameter [ø]	12 mm
Number of teeth [*z*]	4
Depth of cut [a_p]	0.2 mm
Depth of cut [a_e]	0.2 mm
Cutting speed [V_c]	302 m/min
Spindle speed [*n*]	8000 rpm
Feed per tooth [f_z]	0.075 mm/tooth
Table feed [V_f]	2400 mm/min
Time [min, sec]	1 08 32

Recommended method	Common method
Roughing: CoroMill 200 - PM 4030 a_p/a_e = 4/26; v_c/v_f = 180/1592 T 6´ 00” CoroMill 200 - PM 4030 a_p/a_e = 4/26; v_c/v_f = 180/1469 T 1´ 57” CoroMill 200 - WM 530 a_p/a_e = 2-5/<5; v_c/v_f = 500/2238 T 1´ 57”	**Roughing:** Coromant U-drill C1020/P4025 a_p = 84; v_c/v_f = 150/143 T 0´ 35” CoroMill 290 - PM 4030 a_p/a_e = 5/33; v_c/v_f = 200/1273 T 7´ 43” Ballnose endmill - MM1025 a_p/a_e = 6/6; v_c/v_f = 200/995 T 6´ 00” Ballnose endmill - MM1025 a_p/a_e = 1/1; v_c/v_f = 300/1528 T 3´ 15”
Semi-finishing: CoroMill 200 - WM 530 a_p/a_e = 1/0,8; v_c/v_f = 600/5457 T 7´ 30” CoroMill 200 - WM 530 a_p/a_e = 1/0,8; v_c/v_f = 500/2400 T 7´ 30”	**Semi-finishing:** Ballnose endmill - MM 530 a_p/a_e = 1,5/0,8; v_c/v_f = 400/3820 T 7´ 46” Ballnose endmill - MM 530 a_p/a_e = 1,5/0,8; v_c/v_f = 400/1910 T 6´ 44”
Finishing: Ballnose solid carbide endmill - 1010 a_p/a_e = 0,2/0,2; v_c/v_f = 302/2400 T 49´ 16” Total machining time: 1h 7´ 37”	**Finishing:** Ballnose solid carbide endmill - 1010 a_p/a_e = 0,2/0,2; v_c/v_f = 302/2400 T 8´ 32” Total machining time: 1h 40´ 55”

The recommended method results in 33% higher productivity and also better surface quality and geometrical accuracy



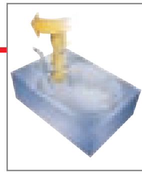
MACHINING EXAMPLE

Machining of thin walls in aluminium

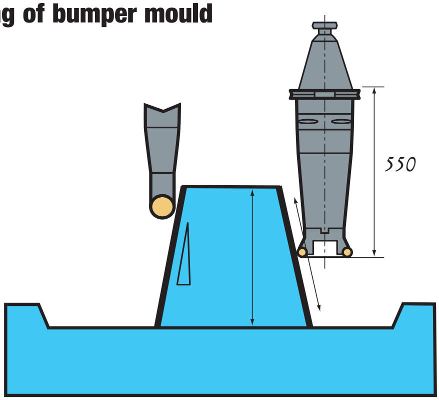
	Roughing	Finishing
Tool:	Solid carbide	Solid carbide
Tool diameter:	10 mm	8 mm
Grade:	GC1020	GC 1020
Number of teeth [*z*]:	2	4
Cutting speed [V_c]:	628 m/min	603 m/min
Spindle speed [*n*]:	20000 rpm	24000 rpm
Table feed [v_f]:	4800 mm/min	9600 mm/min
Feed per tooth [f_z]:	0.12 mm/tooth	0.1 mm/tooth
Depth of cut [a_p]:	5 mm	2 mm
Depth of cut [a_e]:	6-10 mm	0.9 mm

The wall thickness is 0.2, 0.3 and 0.4 mm and the height is 20 mm. The machining is performed by down milling and the first cut on one side is 1 mm in axial depth of cut. The following cuts are 2 mm on each side until the whole 20 mm is machined. The sequence is also explained more in detail in the HSM chapter in this guide.

MACHINING EXAMPLE



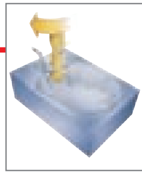
Improved machining of bumper mould



Finishing operation of a bumper mould where the tool length of 550 mm is an instability factor. The machine used is an Okuma V-MC/MCVA-2 with an ISO 50 taper. The material is unalloyed steel.

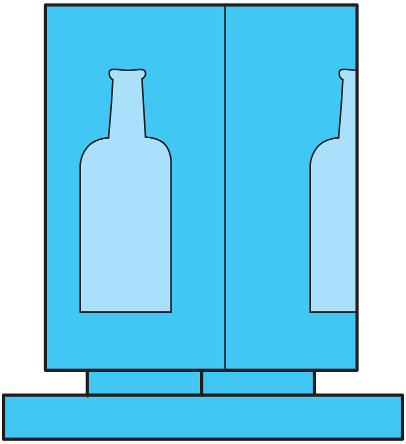
	Old method	**New method**
Tool:	Ballnose endmill	CoroMill 200
Tool diameter:	30 mm	63 mm
Insert:	HC844 (P30+Tin)	GC1025-PL
Number of inserts [z]:	2	4
Cutting speed [V c]:	217m/min	593 m/min
Spindle speed [n]:	2300 rpm	3000 rpm
Table feed [vf]:	2500 mm/min	3000 mm/min
Feed per tooth [fz]:	0.54 mm/tooth	0.25/1.0 mm/tooth
Depth of cut [ap]:	1.0 mm	0.2 mm
Depth of cut [ae]:	1.0 mm	0.2 mm
Machining time:	50 hours/comp	4 hours/comp

Machining time reduced more than 12 times.

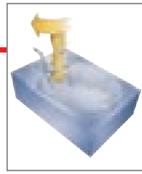


MACHINING EXAMPLE

Roughing of cavity in mould for bottles in high alloyed steel

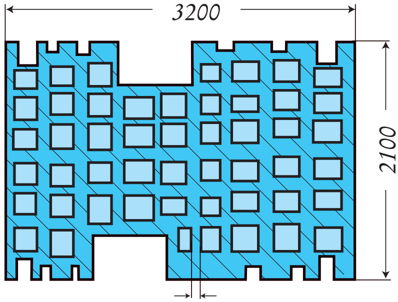


Tool:	CoroMill R200-015A20-10H
Tool diameter:	25 mm
Insert:	RCKT-10T3MO-KH3040
Number of inserts [z]:	3
Cutting speed [V c]:	160 m/min
Spindle speed [n]:	2000
Table feed [vf]:	1500 mm/min
Feed per tooth [fz]:	0.25 mm/tooth
Depth of cut [ap]:	0.8 mm
Depth of cut [ae]:	0.8 mm
Tool-life:	2.5 hours
Machine tool:	YASDA, ISO50



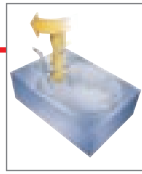
MACHINING EXAMPLE

Improved machining for roughing of grey cast-iron stamping die



	Old method	New method
Tool:	75 facemilling cutter	CoroMill 200,special
Tool diameter:	160 mm	180 mm
Insert:	H 13 A	RCKT-2006-MO-KH,3040
Number of inserts [z]:	10	
Cutting speed [V c]:	100 m/min	102 m/min
Spindle speed [n]:	210 rpm	160 rpm
Table feed [vf]:	800 mm/min	1200 mm/min
Feed per tooth [fz]:	0.35 mm/tooth	0.74 mm/tooth
Tool-life:	Unpredictable due	162 min
to frequent insert failure		

The component machined in the example had a lot of sand inclusions and due to all the cast holes in the component the cutting forces were very unfavourable. The entering and exit angles of the inserts was often close to zero and therefore it was a constant shifting between conventional- and climb-milling where the cut was interrupted. These combined factors call for a rigid milling cutter with strong inserts and as the cutting data in the test shows the round insert cutter is clearly the best choice.

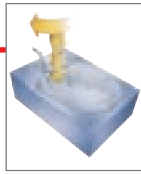


MACHINING EXAMPLE

Roughing of pressing die for trucks in tool steel, IMPAX, machine tool OKUMA MCV-A, 30kW

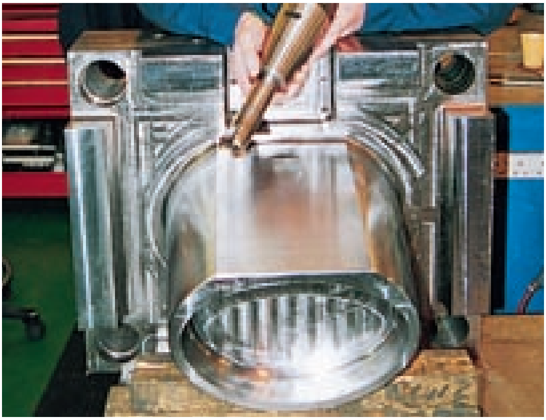
Tool:	CoroMill R200-40Q22-12L
Tool diameter:	44 mm
Insert:	RCKT-1204MO-WM530
Number of inserts [z]:	3
Cutting speed [V c]:	295 m/min
Spindle speed [n]:	1809 rpm
Table feed [vf]:	1628 mm/min
Feed per tooth [fz]:	0.30 mm/tooth
Depth of cut [ap]:	2.5 mm
Depth of cut [ae]:	2-4 mm
Tool-life:	50 min

Rough contouring, downmilling and constant Z-value.



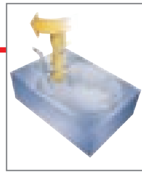
MACHINING EXAMPLE

Finishing of mould for plastic bin in low alloy steel with a hardness of 29 HRC



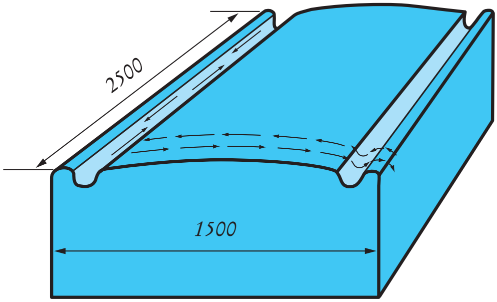
Tool:	CoroMill R200-028A32-12M
Tool diameter:	40 mm
Insert:	RCKT 1204MO-PM,4040
Number of inserts [z]:	3
Cutting speed [V_c]:	250 m/min
Spindle speed [n]:	2000 rpm
Table feed [v_f]:	2000 mm/min
Feed per tooth [f_z]:	0.30 mm/tooth
Depth of cut [a_p]:	0.5 mm
Depth of cut [a_e]:	0.5 mm

The total length of the tool is 400 mm, the component is machined in three passes without the cutting data being changed to get the surface finish required. The two last passes are spring cuts.

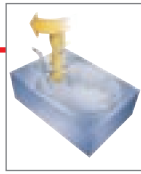


MACHINING EXAMPLE

Roughing of die for outer roof to passenger cars in nodular cast-iron

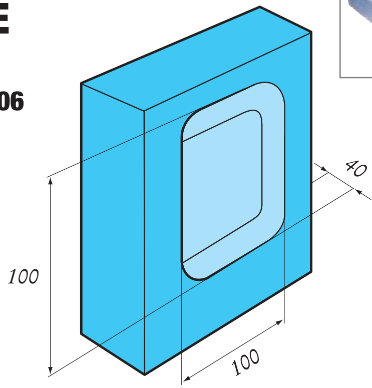


Tool:	BNE R216-32B32-070
Tool diameter:	32 mm
Insert:	R216-3206M-M,4040
Number of inserts [z]:	2
Cutting speed [V c]:	201 m/min
Spindle speed [n]:	20000
Table feed [vf]:	1000-2000 mm/min
Feed per tooth [fz]:	0.25-0.5 mm/tooth
Depth of cut [ap]:	10-18 mm
Depth of cut [ae]:	4-8 mm
Tool-life:	1.5 hours
Machine tool:	SHIN NIPPON KOKI HF-5, ISO50, 40kW



MACHINING EXAMPLE

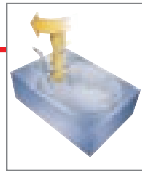
Roughing, semi-finishing and finishing of cavity in material 2606 with a hardness of 46 HRC



	Roughing	Semi-finishing
Tool:	BNE R216-25B25-060	BNE R216-20B25-050
Tool diameter:	25 mm	20 mm
Insert:	BNE R216-2504M-M,1025	R216-20T3M-M,530
Number of inserts [*z*]:	2	2
Cutting speed [V_c]:	120 m/min	200 m/min
Table feed [v_f]:	600 mm/min	2000 mm/min
Depth of cut [a_p]:	3 mm	0.8 mm
Depth of cut [a_e]:	10 mm	0.05 mm
Machining time:	31.5 min	5.1 min

	Finishing
Tool:	BNE R216,44-12030 -AK26L
Tool diameter:	12 mm
Number of inserts [z]	4
Grade:	GC1020
Cutting speed [V_c]:	400 m/min
Table feed [v_f]:	4800 mm/min
Depth of cut [a_p]:	0.3 mm
Depth of cut [a_e]:	0.002 mm
Machining time:	19 min

Previously the machining time for this operation was 2 hours and only solid carbide endmills in small diameters were used. With ball nose endmill with indexable inserts and one solid carbide endmill the operation was performed in 55.6 min. None of the tools showed any significant tool wear.



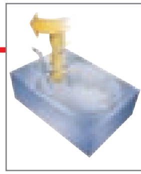
MACHINING EXAMPLE

Roughing of pressing die for trucks in tool steel, IMPAX, machine tool OKUMA MCV-A, 30kW

Tool:	R216-20B25-070
Tool diameter:	20 mm
Insert:	R216-20T3M-M,1025
Number of inserts [z]:	2
Cutting speed [V_c]:	200 m/min
Spindle speed [n]:	3183 rpm
Table feed [v_f]:	1592 mm/min
Feed per tooth [f_z]:	0.25 mm/tooth
Depth of cut [a_p]:	0.5-3.5 mm
Depth of cut [a_e]:	2 mm
Tool-life:	40 min

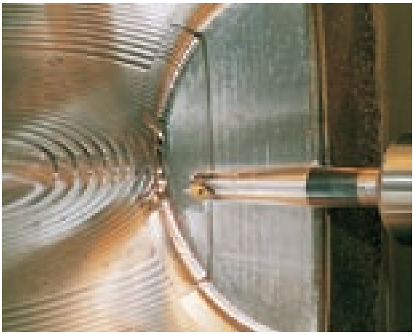
Rough milling of bottom surface, mixed up- and downmilling. Too low cutting speed was used in this case, however no visible wear on the inserts.

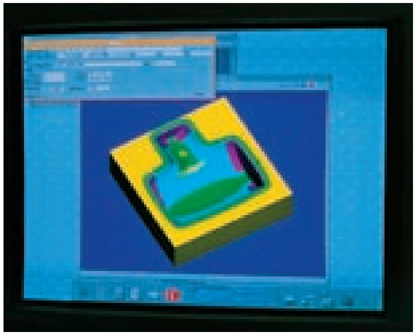
SANDVIK Coromant



MACHINING EXAMPLE

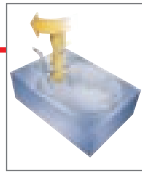
Finishing of forging die in tool steel with a hardness of 35 HRC





Tool:	BNE R216-20B25-50
Tool diameter:	20 mm
Insert:	R216-20T3M-M,1025
Number of inserts [*z*]:	2
Cutting speed [V_c]:	138 m/min
Spindle speed [*n*]:	2200 rpm
Table feed [v_f]:	1550 mm/min
Feed per tooth [f_z]:	0.5 mm/tooth
Depth of cut [a_p]:	2 mm
Depth of cut [a_e]:	0.5 mm

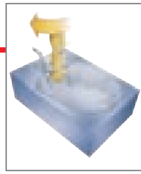
MACHINING EXAMPLE



Semi-finishing of pressing die for trucks in tool steel, IMPAX, machine tool OKUMA MCV-A 30 kW

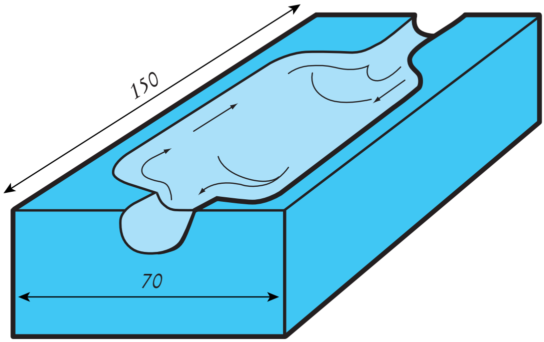
Tool:	BNE R216-25B25-080
Tool diameter:	25 mm
Insert:	R216-2504M-M, 1025
Number of inserts [z]:	2
Cutting speed [V_c]:	300 m/min
Spindle speed [n]:	3820 rpm
Table feed [v_f]:	2292 mm/min
Feed per tooth [f_z]:	0.30 mm/tooth
Depth of cut [a_p]:	0.5-3.5 mm
Depth of cut [a_e]:	1 mm
Tool-life:	2.5 hours

Contouring, up- and downmilling, no visible wear on the inserts.

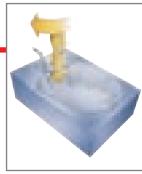


MACHINING EXAMPLE

Finishing of mould for bottles in tool steel (RAMAX)

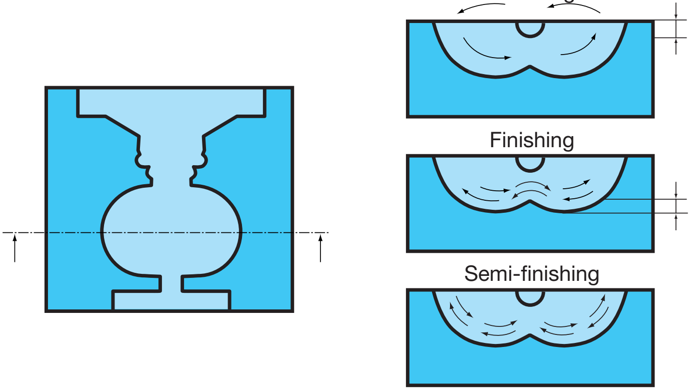


Tool:	BNE R216.42-08030AK16L
Tool diameter:	8 mm
Number of inserts [z]:	2
Grade:	GC1010
Cutting speed [V_c]:	220 m/min
Spindle speed [n]:	11700
Table feed [v_f]:	4200 mm/min
Feed per tooth [f_z]:	0.18 mm/tooth
Depth of cut [a_p]:	0.1 mm
Depth of cut [a_e]:	0.14 mm
Tool-life:	34 pieces
Machine tool:	YASDA, ISO40



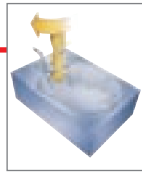
MACHINING EXAMPLE

Semi-finishing and finishing of mould for electro-spindle in high alloy steel



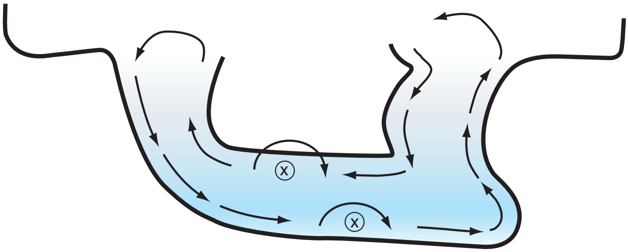
	Semi-finishing	Finishing
Tool:	BNE R216,42-08030AK16L	BNE R216,42-06030AK10L
Tool diameter:	8 mm	6 mm
Number of inerts [z]:	2	2
Grade:	GC1010	GC1010
Cutting speed [V_c]:	276 m/min	370 m/min
Spindle speed [n]:	11000rpm	20000 rpm
Table feed [v_f]:	1400 mm/min	1600 mm/min
Feed per tooth [f_z]:	0.06 mm/tooth	0.04 mm/tooth
Depth of cut [a_p]:	0.3 mm	0.2 mm
Depth of cut [a_e]:	0.5 mm	0.2 mm
Tool-life:	250 min	600 min
Machine tool:	Mori Seiki, ISO30	

** BNE = Ball Nose Endmill*



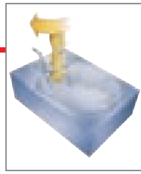
MACHINING EXAMPLE

Semi-finishing and finishing of die for beer mug in high alloy steel



	Semi-finishing	Finishing
Tool:	BNE R216.44-08030-19L	BNE R216.42-06030AK10L
Tool diameter:	8 mm	6 mm
Number of inserts [z]:	4	2
Grade:	GC1010	GC1010
Cutting speed [V_c]:	250 m/min	320 m/min
Spindle speed [n]:	10000rpm	17200 rpm
Table feed [v_f]:	2000-2500 mm/min	3000 mm/min
Feed per tooth [f_z]:	0.05-0.07 mm/tooth	0.07 mm/tooth
Depth of cut [a_p]:	0.2 mm	0.1 mm
Depth of cut [a_e]:	0.2 mm	0.1 mm
Tool-life:	1080 min	600 min
Machine tool:	Mori Seiki, ISO30	

** BNE = Ball Nose Endmill*

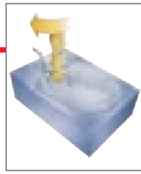


MACHINING EXAMPLE

Slotmilling of inserts for moulds in CALMAX whith a hardness of 56-58 HRC

Tool:	BNE R216,33-02045-ACO6P
Tool diameter:	12 mm
Number of inserts [z]:	3
Grade:	GC1020
Cutting speed [V_c]:	220 m/min
Spindle speed [n]:	15000
Table feed [v_f]:	1500 mm/min
Feed per tooth [f_z]:	0.03 mm/tooth
Depth of cut [a_p]:	0,04 mm
Depth of cut [a_e]:	2 mm
Tool-life:	Not established 160 pieces without wear
Machine tool:	Deckel Maho DMU 50V

Previously the application was performed by spark erosion and the machining time was 1.5 hours per component. With HSM the machining time per component is 5 min. The slot is machined in 48 passes.



MACHINING EXAMPLE

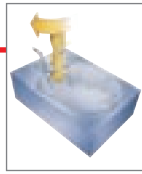
Machining of moulds for ventilation, mounted in the dashboard in cars. The workpiece material is low alloy steel with a hardness of 50-52 HRC

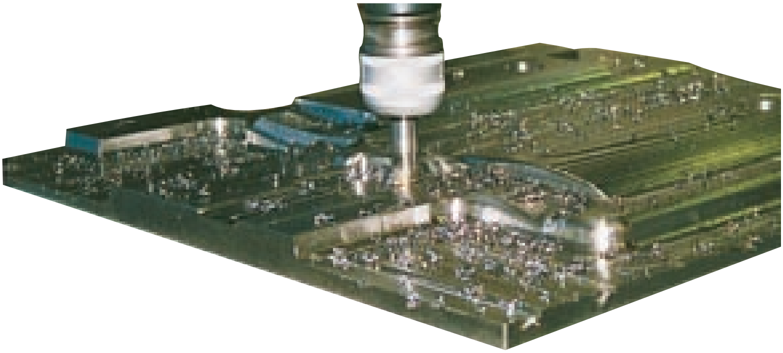
Tool:	BNE R216.42-04030-AKO5L
Tool diameter:	4 mm
Grade:	GC1010
Cutting speed [V_c]:	150 m/min
Spindle speed [n]:	12000
Table feed [v_f]:	1200 mm/min
Feed per tooth [f_z]:	0.05 mm/tooth
Depth of cut [a_p]:	0.1 mm
Depth of cut [a_e]:	0.1 mm
Tool-life:	15-18 hours
Machine tool:	MIKRON VCP 710

There are eight moulds in each set and with the EDM method previously used it took 100 hours to produce a set, with the HSM machine this time has been reduced to 40 hours.

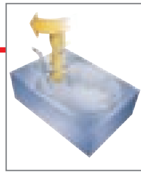
MACHINING EXAMPLE

Semi-finishing of thermoplastic mould





Tool:	CoroMill 390 R390-025B25-11M
Tool diameter:	25 mm
Insert:	R390-11T308M-PM,1025
Number of inserts [*z*]:	3
Cutting speed [V_c]:	162 m/min
Spindle speed [*n*]:	2068 rpm
Table feed [v_f]:	3050 mm/min
Feed per tooth [f_z]:	0,49 mm/tooth
Depth of cut [a_p]:	0.5 mm
Depth of cut [a_e]:	25 mm
Tool-life:	1.5 hours



MACHINING EXAMPLE

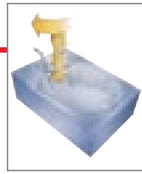
Rough machining of moulds to the rear section of the fenders for trucks

Tool:	CoroMill 390 R390-040B32-11M
Tool diameter:	40 mm
Insert:	R390-11T308M-PM,1025
Number of inserts [z]:	4
Cutting speed [V_c]:	143 m/min
Spindle speed [n]:	1140 rpm
Table feed [v_f]:	780 mm/min
Feed per tooth [f_z]:	0.17 mm/tooth
Depth of cut [a_p]:	16 mm
Depth of cut [a_e]:	3 mm
Tool-life:	45 min/edge

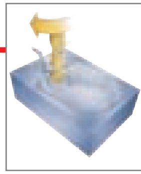
Only downmilling is used in this application, when up-milling was used the tool-life was reduced by 50%.

MACHINING EXAMPLE

Slotmilling of die in P20 steel

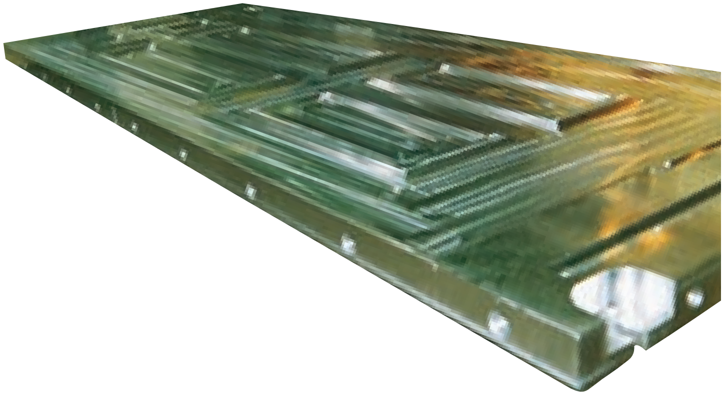


Tool:	CoroMill 390 R390-020B20-11M
Tool diameter:	20 mm
Insert:	R390-11T308M-PM,1025
Number of inserts [*z*]:	3
Cutting speed [V_c]:	185 m/min
Spindle speed [*n*]:	3000 rpm
Table feed [v_f]:	1829 mm/min
Feed per tooth [f_z]:	0,2 mm/tooth
Depth of cut [a_p]:	3 mm
Depth of cut [a_e]:	20 mm

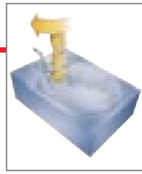


MACHINING EXAMPLE

Facemilling of mould for doors in forged non-alloy steel



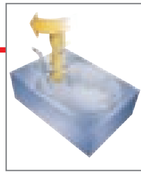
Tool:	CoroMill 245 R245-125Q40-12L
Tool diameter:	125 mm
Insert:	R245-12T3E-PL,4030
Number of inserts [z]:	6
Cutting speed [V_c]:	270 m/min
Table feed [v_f]:	965 mm/min
Feed per tooth [f_z]:	0.19 mm/tooth
Depth of cut [a_p]:	2 mm
Depth of cut [a_e]:	95 mm
Tool-life:	2 components



MACHINING EXAMPLE

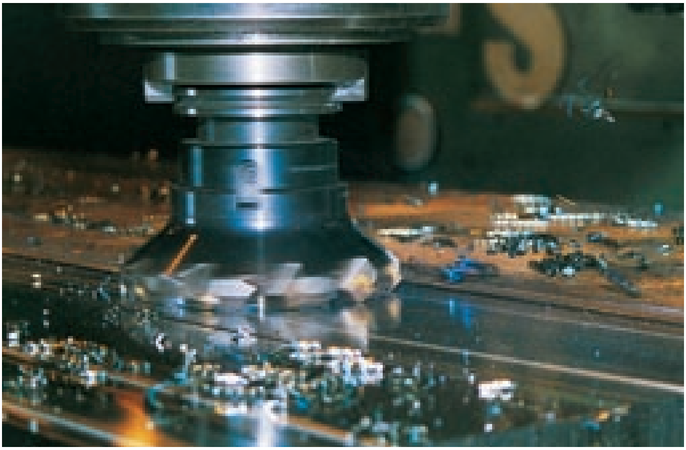
Facemilling of pressing die in cast-iron

Tool:	CoroMill 245 R245-160Q40-12L
Tool diameter:	160 mm
Insert:	R245-12T3M-KM,3020
Number of inserts [z]:	7
Cutting speed [V_c]:	221 m/min
Spindle speed [n]:	440 rpm
Table feed [v_f]:	700 mm/min
Feed per tooth [f_z]:	0,23 mm/tooth
Depth of cut [a_p]:	5 mm
Depth of cut [a_e]:	120 mm

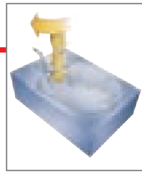


MACHINING EXAMPLE

Rough facemilling of die in P20 steel

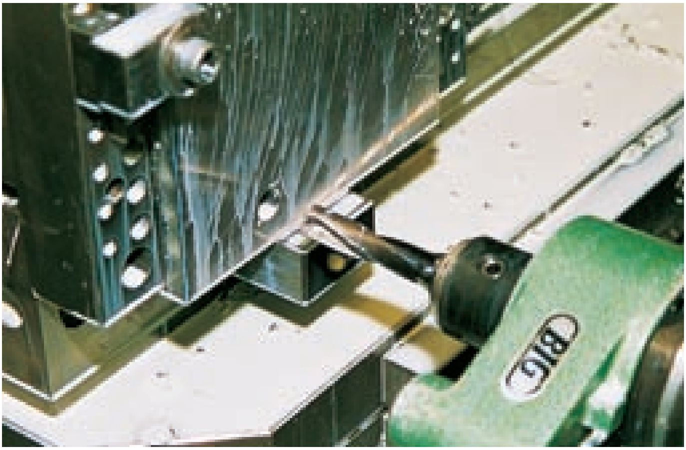


Tool:	CoroMill 245 R245-160Q40-12M
Tool diameter:	152 mm
Insert:	R245-12T3M-PH,4040
Number of inserts [z]:	10
Cutting speed [V_c]:	110 m/min
Spindle speed [n]:	500 rpm
Table feed [v_f]:	914 mm/min
Feed per tooth [f_z]:	0,18 mm/tooth
Depth of cut [a_p]:	5 mm
Depth of cut [a_e]:	106 mm

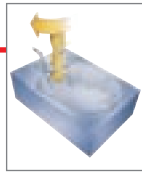


MACHINING EXAMPLE

Drilling through holes for guide pins in 30 mm thick plates for mould bases in low carbon steel



Tool:	Coromant U R416.2-0165L20-31
Tool diameter:	16.5 mm
Insert:	LCMX 020204C-53
Cutting speed [V_c]:	62 m/min
Spindle speed [n]:	1200 rpm
Feed [v_n]:	0.1 mm/rev
Depth of cut [a_p]:	30 mm
Depth of cut [a_e]:	16,5 mm

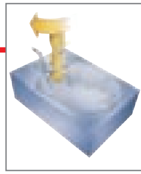


MACHINING EXAMPLE

Drilling holes in mould bases in non-alloy steel

Tool:	Coromant U R416.2-0260L32-41
Tool diameter:	26 mm
Insert:	WCMX 05 0308R-53,1020 (centre) WCMX 05 0308R-53,3040 (periphery)
Cutting speed [V_c]:	150 m/min
Spindle speed [n]:	1850 rpm
Feed [f_n]:	0.10 mm/rev
Depth of cut [a_p]:	102 mm
Depth of cut [a_e]:	26 mm

The reason for using two different insert grades is the difference in cutting speed between the centre and the periphery of the drill. The GC 1020 grade have good wear resistance and toughness at low to moderate cutting speeds. While the GC4025 grade has good wear resistance and edge security at high cutting speed.

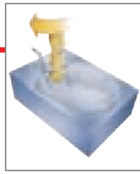


MACHINING EXAMPLE

Drilling of holes for guide pins in mould bases in non-alloy steel

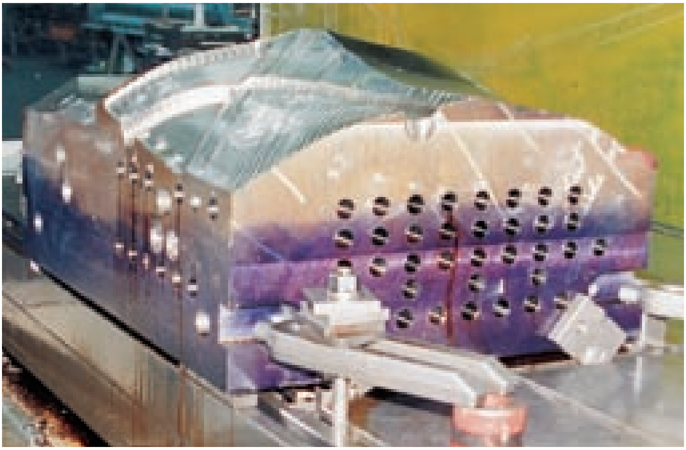
Tool:	Coromant U R416.0240L 25-41
Tool diameter:	24 mm
Insert:	LCMX 04 0308-53.1020
Cutting speed [V_c]:	113 m/min
Spindle speed [n]:	1499 rpm
Feed [f_n]:	0,12 mm/rev

Previously high speed steel drills were used for this application and when changed to the Coromant U drill the feed has been increased by 360% and the cutting speed by 514 %. The U drill has also been equipped with an adaptor to facilitate internal cutting fluid. Which is of great importance for chip evacuation when drilling.



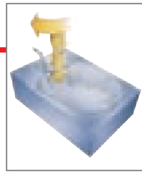
MACHINING EXAMPLE

Deep hole drilling of mould base, change of method from Gun drilling to STS, resulting in significant savings. The workpiece material is low alloy steel with a hardness of 300 HB



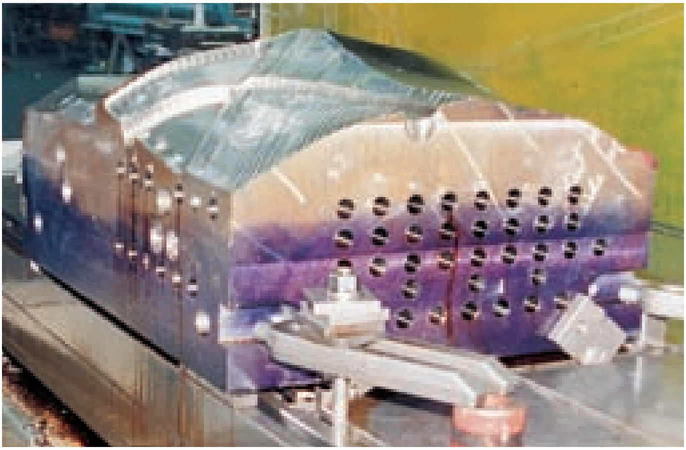
	New method	Old method
Tool:	Sandvik Coromant Single Tube System 420.6-0224 DO.093	Gun drill
Tool diameter:	23.8 mm	23.8 mm
Insert:	67 TIN/TIC,N	C4
Cutting speed [V c]:	60 m/min	62 m/min
Table feed [vf]:	165 mm/min	38 mm/min
Depth of hole:	401.6 mm	401.6 mm
Machining time/comp:	5.25 hours	22.75 hours

The change to STS drilling in this case resulted in a saving of 1050 US dollar per componet.



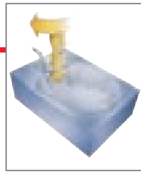
MACHINING EXAMPLE

Deep hole drilling (cross hole drilling) change of drilling method from Gun drilling to STS-drilling



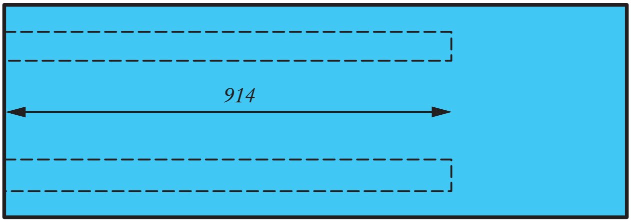
	New method	Old method
Tool:	Sandvik Coromant Single Tube System 420.6-0513 D1.156	Gun drill
Tool diameter:	29.4 mm	29.4 mm
Insert:	67 TIN/TIC,N (cross hole adapter)	C4
Cutting speed [V_c]:	55 m/min	62 m/min
Table feed [v_f]:	122 mm/min	31.8 mm/min
Depth of hole:	762 mm (1 cross)	762 mm (1 cross)
Machining time/comp:	8.1 hours	31.18 hours

The change to STS drilling in this case resulted in a saving of 1269 US dollar per componet. The saving is calculated on cross hole only.



MACHINING EXAMPLE

Deep hole drilling of waterlines in moulds, change of drilling method from Gun drilling to STS drilling. Workpiece material; low alloy steel with a hardness of 350 HB



	New method	Old method
Tool:	Sandvik Coromant Single Tube System 420.6-0514	Gun drill
Tool diameter:	29.4 mm	29.4 mm
Insert:	70	70
Cutting speed [V_c]:	62 m/min	62 m/min
Table feed [v_f]:	139.7 mm/min	25.4 mm/min
Depth of hole:	914 mm (1 cross)	914 mm (1 cross)
Machining time/comp:	3.3 hours	18 hours

The change to STS drilling in this case resulted in a saving of 882 US dollar per component.

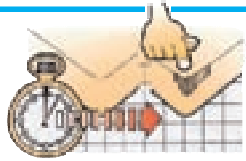
TROUBLE SHOOTING

Problem: The machining results are affected by vibration	
Cause	**Action**
Weak clamping	Establish the direction of the cutting forces and position the material support accordingly. Try to improve the clamping generally. Reduce the cutting forces by reducing the radial and axial cutting depth. Choose a milling cutter with a coarse pitch and positive design. Choose positive inserts with a small corner radius and small parallel flats. Where possible, choose an insert grade with a thin coating and a sharp cutting edge. If necessary, choose an uncoated insert grade. Avoid machining where the work-piece has poor support against cut-ting forces.
Axially weak workpiece	The first choice is a square shoulder cutter with positive inserts. Choose an L-geometry with a sharp cutting edge and a large clearance angle which produces low cutting forces. Try to reduce the axial cutting forces by reducing the axial cutting depth, as well as using positive inserts with a small corner radius, small parallel lands and sharp cutting edges.

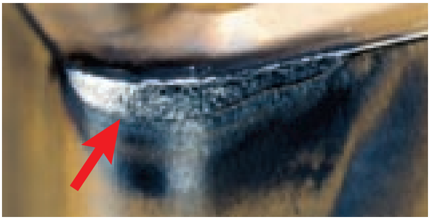
Problem: The machining results are affected by vibration	
Cause	**Action**
Large overhang either on the spindle or the tool	Always use a coarse and a differentially pitched milling cutter. Balance the cutting forces axially and radially. Use a 45 degree entering angle, large corner radius or round inserts. Use maximum sized extensions, tapered if possible and/or tuned. Use heavy metal shanks. Use inserts with a light cutting geometry. Try to reduce the overhang, every millimetre counts! Reduce rpm but not feed/tooth. Reduce a_p/a_e if necessary.
Square shoulder milling with a radially weak spindle	Choose the smallest possible milling cutter diameter in order to obtain the most favourable entering angle. The smaller diameter the milling cutter has the smaller the radial cutting forces will be. Choose positive and light cutting geometries. Try up milling.
Uneven table feed.	Try up milling. Look at the possibility of adjusting the feed screw on conventional machines. Adjust the lock screw or replace the ball screw on CNC-machines.

TROUBLE SHOOTING - SOLID CARBIDE ENDMILLS

Problem	Possible solution
Vibrations The most common problems • High noise • Uneven wear and chipping • Corner damage • Short toollife • Bad surface finish	- Reduce tool run-out max 0,02mm (HSM max. 0.01) - Minimize tool outstick!! 20% outstick reduction reduces the tool deflection by 50%!! - Improve stability in clamping - If possible use larger tool diameter - Use a tool with less teeth - Divide cutting depth in several steps and increase f_z - Use a tool with high helix - Increase f_z - Reduce V_c
Chip jamming Critical in full slotting especially in long chipping materials. Can cause corner damage, edge chipping and breakage. Recutting of chips reduces toollife.	- Use 2 or maximum 3-fluted tools - Divide cutting depth into several passes - Try high helix tools - Use well directed compressed air to evacuate chips - Reduce f_z
Unacceptable workpiece/surface Geometrical and dimensional tolerances Surface finish and burr generation	- Use upmilling in finishing - Reduce radial depth of cut - Reduce tool outstick - In finish-copying in steep pockets use BNE with spherical design to reduce deflection. - Check runout in holder - Avoid ***Vibrations*** - Reduce fz and/or increase v_c - Check wear on the tool
Tool breakage or chipping Often caused by overloading of tool or chip jamming.	- Reduce cutting depth - Reduce tool outstick - Use largest possible tool diameter - Avoid ***Chip jamming*** - Reduce f_z - Evacuate chips to avoid recutting!
Smearing and build up edge Especially austenitic materials, titanium and aluminum	- Use coolant or oilmist - Increase v_c
High wear Mostly due to unfavourable cutting conditions or very abrasive materials	- Use downmilling - Increase fz - Check recommended cutting data - Avoid ***Vibrations*** - Avoid recutting of chips



TROUBLE SHOOTING
Tool Wear

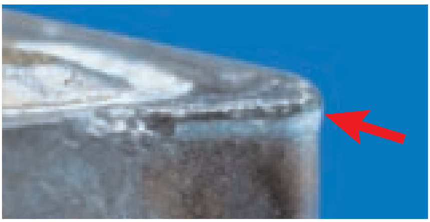


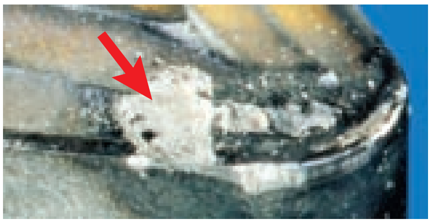
FLANK WEAR

Tool wear	Possible cause	Possible remedy
Rapid flank wear causing poor suface texture or inconsistency in tolerance.	Cutting speed too high or insufficient wear resistance.	Select a more wear resistant grade. For work-hardening materials, select a smaller entering angle. Reduce cutting speed when machining heat resistant material.

CRATER WEAR

Tool wear	Possible cause	Possible remedy
Excessive crater wear causing a weakened edge and poor surface finish.	Excessive cutting temperatures and pressure on the top face of inserts.	First, reduce cutting speed to obtain a lower temperature and secondly, the feed. Select a more wear resistant grade. Select a positive insert geometry



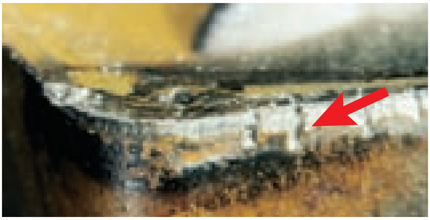


PLASTIC DEFORMATION

Tool wear	Possible cause	Possible remedy
Plastic deformation of edge, depression or flank impression, leading to poor chip control poor surface finish and insert breakage.	Cutting temperature and pressure too high.	Select a more wear resistant grade, which is harder. • Reduce cutting speed. • Reduce feed.

BUILT-UP EDGE

Tool wear	Possible cause	Possible remedy
Built-up edge causing poor surface finish and cutting edge chipping, when the BUE is torn away.	Cutting zone temperature is too low. Negative cutting geometry. Very sticky material, such as low-carbon steel, stainless steels and aluminium.	Increase cutting speed. Change to a more suitable coated grade. Select a positive geometry insert.

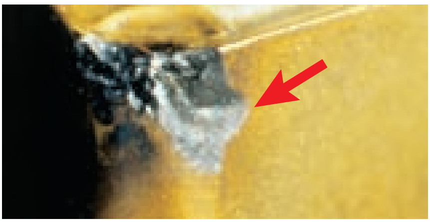


THERMAL CRACKS

Tool wear	Possible cause	Possible remedy
Small cracks perpendicular to the cutting edge causing chipping and poor surface finish.	Excessive temperature variations. Intermittent machining. Varying coolant supply.	Select a tougher insert grade. Coolant should be applied copiously or not at all.

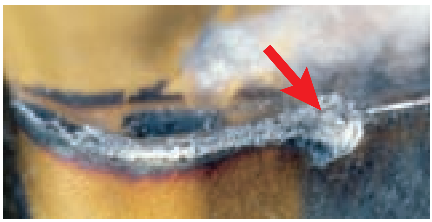
CHIPPING

Tool wear	Possible cause	Possible remedy
Small cutting edge chipping leading to poor surface texture and excessive flank wear.	Cutting edge too brittle. Insert edge too weak. Built-up edge has been formed.	Select tougher grade. Select an insert with a stronger cutting edge. Increase cutting speed. Select a positive geometry. Reduce feed at beginning of cut. Improve stability.



EDGE FRACTURE

Tool wear	Possible cause	Possible remedy
Damages not only the insert but can also ruin the shim and workpiece.	Excessive tool wear. Grade and geometry too weak. Excessive load on the insert. Built-up edge has been formed.	Reduce feed and/or depth of cut. Select a stronger geometry, preferably a single sided insert. Select a thicker/larger insert and tougher grade. Improve stability.



NOTCH WEAR

Tool wear	Possible cause	Possible remedy
Notch wear causing poor surface texture and risk of edge breakage.	Cutting speed too high or insufficient wear resistance.	Select a more wear resistant grade. For work-hardening materials, select a smaller entering angle. Reduce cutting speed when machining heat resistant material.

TECHNICAL DATA

Metal Cutting terminology and units

Designations according to ISO Unit

a_e	=	Radial depth of cut	mm
a_p	=	Axial depth of cut	mm
a_r	=	Radial depth of cut for side and facemills	mm
D_e	=	Diameter of tool at a given depth of cut	mm
D_c	=	Diameter of tool	mm
D_m	=	Machined diameter - turning	mm
f_n	=	Feed per revolution	mm/rev
f_z	=	Feed per tooth	mm/z
h_m	=	Average chip thickness	mm
k_c	=	Specific cutting force	N/mm^2
l_m	=	Machined length	mm
n	=	Spindle speed (Revolutions per minute)	RPM
M_c	=	Torqe	Nm
P_c	=	Net power	kW
Q	=	Metal removal rate	cm^3/min
V_c	=	Cutting speed	m/min
V_f	=	Feed per minute	mm/min
Z_c	=	Number of effective teeth in tool	pieces
Z_n	=	Number of teeth in tool	pieces
η_{mt}	=	Efficiency	

Metal Cutting formulas

Effective cutting speed V_e (m/min)

$$V_e = \frac{\pi \times D_e \times n}{1000}$$

Cutting speed, V_c (m/min)

$$V_c = \frac{\pi \times D_c \times n}{1000}$$

Revolutions per minute, n (rev/min)

$$n = \frac{1000 \times V_c}{\pi \times D_c}$$

Feed per minute, v_f (mm/min)

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

Feed per tooth, f_z (mm/tooth)

$$f_z = \frac{V_f}{(z_n \times n)}$$

Cutting time, T_c (min)

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

Metal removal rate, Q (cm^3/min)

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

Net power at spindle, P_c (kW)

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

Torque at spindle, M_c (Nm)

$$M_c = \frac{f_n \times \pi \;\; D_c^2 \times k_c}{4000}$$

Average chip thickness (mm) when, $a_e/D_c \geq 0{,}1$

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

Specific cutting force (N/mm^2)

$$k_c = k_{c1} \times h_m^{-mc}$$

Approximate power consumption (for K-value see page 189)

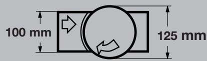
$$P_c = \frac{a_p \times a_e \times V_f \times K}{100\ 000}$$

CONSTANT K FOR USE IN POWER REQUIREMENT CALCULATION 1)

ISO	CMC No.	Description		a_e/D_c=0,8			a_e/D_c=0,4			a_e/D_c=0,2		
				f_z=0,1	f_z=0,2	f_z=0,4	f_z=0,1	f_z=0,2	f_z=0,4	f_z=0,1	f_z=0,2	f_z=0,4
P		**Steel**										
	01.1	**Unalloyed**	C = 0,10–0,25%	5.7	4.8	4.0	6.2	5.2	4.4	6.8	5.7	4.8
	01.2		C = 0,25–0,55%	6.1	5.1	4.3	6.6	5.6	4.7	7.2	6.1	5.1
	01.3		C = 0,55–0,80%	6.5	5.4	4.6	7.1	5.9	5.0	7.7	6.5	5.4
	01.4			6.9	5.8	4.8	7.5	6.3	5.3	8.2	6.9	5.8
	01.5			7.6	6.4	5.4	8.3	7.0	5.9	9.1	7.6	6.4
	02.1	**Low-alloy**	Non-hardened	6.5	5.4	4.6	7.1	5.9	5.0	7.7	6.5	5.4
	02.2	(alloying elements ≤5%)	Hardened and tempered	7.6	6.4	5.4	8.3	7.0	5.9	9.1	7.6	6.4
	03.11	**High-alloy**	Annealed	7.4	6.2	5.3	8.1	6.8	5.7	8.8	7.4	6.2
	03.13	(alloying elements ≤5%)	Hardened tool steel	8.2	6.9	5.8	8.9	7.5	6.3	9.7	8.2	6.9
	03.21			11.0	9.3	7.8	12.0	10.1	8.5	13.1	11.0	9.3
	03.22			11.8	9.9	8.4	12.9	10.8	9.1	14.0	11.8	9.9
	06.1	**Castings**	Unalloyed	5.3	4.5	3.8	5.8	4.9	4.1	6.3	5.3	4.5
	06.2		Low-alloy, alloying elements ≤5%	6.1	5.1	4.3	6.6	5.6	4.7	7.2	6.1	5.1
	06.3		High-alloy, alloying elements >5%	7.4	6.2	5.3	8.1	6.8	5.7	8.8	7.4	6.2
M		**Stainless steel**										
	05.11	Ferritic/Martensitic	Non-hardened	6.2	5.4	4.7	6.7	5.8	5.0	7.2	6.2	5.4
	05.12		PH-hardened	9.7	8.4	7.2	10.4	9.0	7.8	11.2	9.7	8.4
	05.13		Hardened	8.0	6.9	5.9	8.6	7.4	6.4	9.2	8.0	6.9
	05.21	Austenitic	Non-hardened	6.9	6.0	5.2	7.4	6.4	5.6	8.0	6.9	6.0
	05.22		PH-hardened	9.7	8.4	7.2	10.4	9.0	7.8	11.2	9.7	8.4
	05.51	Austenitic-Ferritic	Non-weldable ≥0,05%C	6.9	6.0	5.2	7.4	6.4	5.6	8.0	6.9	6.0
	05.52	(Duplex)	Weldable <0,05%C	8.3	7.2	6.2	8.9	7.7	6.7	9.6	8.3	7.2
		Stainless steel – cast										
	15.11	Ferritic/Martensitic	Non-hardened	6.5	5.4	4.6	7.1	5.9	5.0	7.7	6.5	5.4
	15.12		PH-hardened	9.5	8.0	6.7	10.4	8.7	7.3	11.3	9.5	8.0
	15.13		Hardened	8.0	6.7	5.7	8.7	7.3	6.2	9.5	8.0	6.7
	15.21	Austenitic	Non-hardened	6.9	5.8	4.8	7.5	6.3	5.3	8.2	6.9	5.8
	15.22		PH-hardened	9.5	8.0	6.7	10.4	8.7	7.3	11.3	9.5	8.0
	15.51	Austenitic-Ferritic	Non-weldable ≥0,05%C	6.9	5.8	4.8	7.5	6.3	5.3	8.2	6.9	5.8
	15.52	(Duplex)	Weldable <0,05%C	8.4	7.0		9.1	7.7		10.0	8.4	
S		**Heat resistant super alloys**										
	20.11	**Heat resistant super alloys**	Annealed or solution treated	9.1	7.7		10.0	8.4		10.9	9.1	
	20.12		Aged or solution treated and aged	9.5	8.0		10.4	8.7		11.3	9.5	
	20.21	NIckel base	Annealed or solution treated	10.1	8.5		11.0	9.3		12.0	10.1	
	20.22		Aged or solution treated and aged	11.0	9.3		12.0	10.1		13.1	11.0	
	20.24		Cast or cast and aged	11.4	9.6		12.5	10.5		13.6	11.4	
	20.31	Cobalt base	Annealed or solution treated	10.3	8.6		11.2	9.4		12.2	10.3	
	20.32		Solution treated and aged	11.4	9.6		12.5	10.5		13.6	11.4	
	20.33		Cast or cast and aged	11.8	9.9		12.9	10.8		14.0	11.8	
		Titanium alloys										
	23.1		Commercial pure (99,5% Ti)	4.7	4.0		5.1	4.4		5.5	4.7	
	23.21		α, near α and α+β alloys, annealed	5.1	4.3		5.5	4.7		6.0	5.1	
	23.22		α+β alloys in aged cond. β alloys, annealed or aged	5.1	4.3		5.5	4.7		6.0	5.1	
H	**04.1**	**Extra hard steel Hard steel**	Hardened and tempered	16.0	13.5		17.4	14.7		19.0	16.0	
	10.1	**Chilled cast iron**	Cast or cast and aged	9.0	7.4		9.9	8.2		10.9	9.0	
K	**07.1**	**Malleable cast iron**	Ferritic (short chipping)	3.3	2.7	2.2	3.6	3.0	2.4	4.0	3.3	2.7
	07.2		Pearlitic (long chipping)	3.7	3.0	2.5	4.1	3.3	2.8	4.5	3.7	3.0
	08.1	**Grey cast iron**	Low tensile strength	3.7	3.0	2.5	4.1	3.3	2.8	4.5	3.7	3.0
	08.2		High tensile strength	4.5	3.7	3.1	5.0	4.1	3.4	5.5	4.5	3.7
	09.1	**Nodular SG iron**	Ferritic	3.7	3.0	2.5	4.1	3.3	2.8	4.5	3.7	3.0
	09.2		Pearlitic	5.5	4.6		6.1	5.0		6.7	5.5	
N	**30.11**	**Aluminium alloys**	Wrought or wrought and coldworked, non-aging	1.5	1.3		1.7	1.4		1.8	1.5	
	30.12		Wrought or wrought and aged	2.5	2.1		2.7	2.3		2.9	2.5	
	30.21	**Aluminium alloys**	Cast, non-aging	2.3	1.9		2.5	2.1		2.7	2.3	
	30.22		Cast or cast and aged	2.7	2.2		2.9	2.4		3.2	2.7	
	30.3			1.3	1.1		1.5	1.2		1.6	1.3	
	30.41	**Aluminium alloys**	Cast, 13–15% Si	2.7	2.2		2.9	2.4		3.2	2.7	
	30.42		Cast, 16–22% Si	2.7	2.2		2.9	2.4		3.2	2.7	
	33.1	**Copper and copper alloys**	Free cutting alloys, ≥1% Pb	2.1	1.8		2.3	1.9		2.5	2.1	
	33.2		Brass, leaded bronzes, ≤ 1% Pb	2.1	1.8		2.3	1.9		2.5	2.1	
	33.3		Bronze and non-leaded copper incl. electrolytic copper	5.1	4.3		5.6	4.7		6.1	5.1	

1)Calculated with an efficiency η_{mt} = 0,8

FACEMILLING, SQUARE SHOULDER FACEMILLING AND SLOTTING WITH SIDE AND FACEMILLING



ISO	CMC No.	Material		Specific cutting force k_c 1 (N/mm²)	Hardness Brinell (HB)	mc	Basic grades 4020 — Feed, f_z or h_{ex}: 0,1– 0,2– 0,3 — Cutting speed, v_c m/min	4030 — Feed, f_z or h_{ex}: 0,1– 0,2– 0,3 — Cutting speed, v_c m/min
P		**Steel**						
	01.1	**Unalloyed**		1500	125	0,25	490– 405– 330	365– 295– 245
	01.2		C = 0,10 – 0,25 %	1600	150	0,25	440– 360– 295	325– 265– 220
	01.3		C = 0,25 – 0,55 %	1700	170	0,25	415– 340– 280	305– 255– 205
	01.4		C = 0,55 – 0,80 %	1800	210	0,25	365– 295– 245	265– 220– 180
	01.5			2000	300	0,25	265– 220– 180	195– 165– 135
	02.1	**Low-alloy** (alloying elements ≤ 5 %)	Non-hardened	1700	175	0,25	345– 285– 230	255– 210– 170
	02.2		Hardened and tempered	2000	275	0,25	245– 195– 165	180– 145– 120
	02.2			2300	350	0,25	195– 160– 130	145– 115– 95
	03.11	**High-alloy** (alloying elements > 5 %)	Annealed	1950	200	0,25	295– 245– 200	195– 155– 130
	03.13		Hardened tool steel	2150	200	0,25	215– 175– 145	160– 130– 105
	03.21			2900	300	0,25	185– 155– 125	140– 115– 95
	03.22			3100	380	0,25	115– 95– 75	85– 70– 55
	06.1	**Castings**	Unalloyed	1400	150	0,25	350– 285– 235	260– 215– 175
	06.2		Low-alloy (alloying elements ≤ 5 %)	1600	200	0,25	275– 225– 185	205– 170– 140
	06.3		High-alloy, alloying elements > 5 %)	1950	200	0,25	205– 165– 135	150– 125– 100

ISO	CMC No.	Material		Specific cutting force k_c 1 (N/mm²)	Hardness Brinell (HB)	mc	Basic grades 2030 — Feed, f_z or h_{ex}: 0,05– 0,15– 0,25 — Cutting speed, v_c m/min	2040 — Feed, f_z or h_{ex}: 0,1– 0,2– 0,3 — Cutting speed, v_c m/min
M		**Stainless steel**						
	05.11	Ferritic/martensitic	Non-hardened	1800	200	0,21	240– 190– 155	235– 190– 155
	05.12		PH-hardened	2800	330	0,21	170– 135– 105	165– 130– 105
	05.13		Hardened	2300	330	0,21	175– 140– 155	175– 135– 110
	05.21	Austenitic	Non-hardened	2000	200	0,21	235– 185– 150	200– 160– 125
	05.22		PH-hardened	2800	330	0,21	165– 130– 105	155– 125– 100
	05.51	Austenitic-ferritic (Duplex)	Non-weldable ≥ 0,05%C	2000	230	0,21	195– 155– 125	165– 135– 105
	05.52		Weldable < 0,05%C	2400	260	0,21	165– 130– 105	135– 105– 85
		Stainless steel – Cast						
	15.11	Ferritic/martensitic	Non-hardened	1700	200	0,25	215– 170– 135	210– 170– 135
	15.12		PH-hardened	2500	330	0,25	145– 115– 95	145– 115– 90
	15.13		Hardened	2100	330	0,25	160– 125– 105	160– 125– 100
	15.21	Austenitic	Austenitic	1800	200	0,25	225– 175– 145	190– 155– 125
	15.22		PH-hardened	2500	330	0,25	145– 115– 95	145– 85– 90
	15.51	Austenitic-ferritic (Duplex)	Non-weldable ≥ 0,05%C	1800	230	0,25	185– 145– 115	155– 125– 100
	15.52		Weldable < 0,05%C	2200	260	0,25	150– 120– 95	125– 100– 80

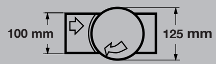
ISO	CMC No.	Material		Specific cutting force k_c 1 (N/mm²)	Hardness Brinell (HB)	mc	Basic grades 1025 — Feed, f_z or h_{ex}: 0,1– 0,15– 0,2 — Cutting speed, v_c m/min	H10F — Feed, f_z or h_{ex}: 0,1– 0,15– 0,2 — Cutting speed, v_c m/min
-S		**Heat resistant super alloys**						
	20.11	Iron base	Annealed or solution treated	2400	200	0,25	65– 55– 55	55– 50– 47
	20.12		Aged or solution treated and aged	2500	280	0,25	46– 43– 40	40– 37– 35
	20.21	Nickel base	Annealed or solution treated	2650	250	0,25	60– 55– 50	50– 48– 45
	20.22		Aged or solution treated and aged	2900	350	0,25	37– 34– 32	32– 30– 27
	20.24		Cast or cast and aged	3000	320	0,25	46– 42– 39	40– 37– 34
	20.31	Cobalt base	Annealed or solution treated	2700	200	0,25	25– 22– 20	22– 19– 17
	20.32		Solution treated and aged	3000	300	0,25	18– 16– 14	15– 14– 12
	20.33		Cast or cast and aged	3100	320	0,25	16– 14– 13	14– 13– 12
		Titanium alloys[1]			Rm[2]			
	23.1		Commercial pure (99,5% Ti)	1300	400	0,23	130– 120– 110	115– 105– 95
	23.21		α, near α and α+β alloys, annealed	1400	950	0,23	65– 65– 55	55– 55– 50
	23.22		α+β alloys in aged cond., β alloys, annealed or aged	1400	1050	0,23	55– 50– 48	49– 45– 42

[1] 45–60° entering angle, positive cutting geometry and coolant should be used.
[2] Rm = ultimate tensile strength measured in MPa.

Basic grades 4040	1025	3040	530	SM30
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)				
0,1– 0,2– 0,4	0,05– 0,1– 0,2	0,05– 0,1– 0,2	0,1– 0,2– 0,4	0,1– 0,2– 0,4
Cutting speed, v_c m/min				
305– 225– 170	340– 305– 255	385– 315– 260	425– 385– 350	265– 225– 165
275– 225– 155	305– 275– 225	350– 285– 235	385– 350– 315	235– 205– 150
260– 215– 145	285– 260– 215	330– 270– 220	365– 325– 295	225– 195– 145
225– 185– 125	250– 225– 185	285– 235– 195	315– 285– 260	195– 165– 125
165– 135– 95	185– 165– 135	215– 175– 145	235– 210– 195	145– 125– 90
215– 175– 120	235– 215– 175	275– 225– 185	300– 275– 245	185– 160– 115
155– 125– 85	165– 155– 125	190– 155– 130	210– 190– 175	130– 115– 85
125– 100– 65	135– 125– 100	155– 125– 105	170– 155– 140	105– 90– 65
165– 135– 90	180– 165– 135	205– 170– 145	225– 205– 185	140– 120– 85
135– 110– 75	150– 135– 110	170– 140– 115	185– 170– 155	115– 100– 75
115– 95– 65	130– 115– 95	145– 120– 100	165– 150– 135	105– 85– 65
75– 60– 41	80– 75– 60	95– 75– 65	105– 95– 85	65– 55– 41
220– 180– 120	245– 220– 180	275– 225– 185	305– 275– 250	190– 165– 120
175– 145– 95	195– 175– 145	220– 180– 145	245– 220– 200	150– 130– 95
125– 105– 70	140– 125– 105	160– 135– 110	175– 160– 145	110– 95– 70

Basic grades 4040	1025	530	4030
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)			
0,1– 0,2– 0,4	0,1– 0,15– 0,2	0,1– 0,15– 0,2	0,1– 0,2– 0,3
Cutting speed, v_c m/min			
200– 160– 105	210– 185– 165	285– 255– 225	245– 195– 115
115– 90– 55	120– 105– 95	165– 145– 130	140– 115– 90
115– 95– 60	125– 110– 100	170– 155– 135	145– 115– 95
185– 145– 95	195– 175– 155	265– 235– 215	– – –
110– 85– 55	115– 105– 90	155– 140– 125	– –
180– 145– 95	190– 170– 155	260– 235– 205	– – –
160– 125– 85	170– 150– 135	230– 205– 185	– – –
180– 145– 95	190– 170– 150	255– 230– 205	255– 180– 145
90– 70– 46	95– 85– 75	125– 115– 105	110– 90– 70
100– 80– 50	105– 95– 85	145– 125– 115	125– 100– 80
180– 145– 95	190– 170– 150	255– 230– 205	– – –
90– 75– 46	95– 85– 75	125– 115– 105	– – –
135– 105– 65	135– 125– 110	190– 170– 150	– – –
135– 105– 65	140– 125– 115	195– 175– 155	– – –

Basic grades H13A
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)
0,1– 0,15– 0,2
Cutting speed, v_c m/min
60– 55– 50
44– 41– 38
55– 55– 49
35– 33– 30
44– 41– 38
23– 21– 18
17– 15– 13
16– 14– 13
125– 115– 105
65– 60– 55
55– 49– 46



ISO		CMC No.	Material		Specific cutting force k_c 1 N/mm²	Hardness Brinell HB	mc	BASIC GRADES 3020	3040
								Feed, f_z or h_{ex} 0,1– 0,2– 0,3	0,1– 0,2– 0,4
								Cutting speed, v_c m/min	
K		07.1 07.2	**Malleable cast iron**	Ferritic (short chipping) Pearlitic (long chipping)	800 900	130 230	0,28 0,28	265– 215– 180 220– 180– 145	240– 195– 135 195– 165– 110
		08.1 08.2	**Grey cast iron**	Low tensile strength High tensile strength	900 1100	180 245	0,28 0,28	290– 235– 195 235– 190– 155	260– 215– 145 210– 170– 115
		09.1 09.2	**Nodular cast iron**	Ferritic Pearlitic	900 1350	160 250	0,28 0,28	180– 145– 125 165– 135– 115	165– 135– 90 150– 125– 85
		CMC No.	**Material**		**Specific cutting force k_c 1 N/mm²**	**Hardness Brinell HB**	**mc**	**BASIC GRADES 4020**	**3040**
								Feed, f_z or h_{ex} 0,07– 0,1– 0,12	0,07– 0,12– 0,2
								Cutting speed, v_c m/min	
	-H	04.1	**Extra hard steel**	Hardened and tempered	4200	59 HRC	0,25	65– 55– 48	47– 41– 33
		10.1	**Chilled cast iron**	Cast or cast and aged	2200	400	0,28	125– 110– 90	90– 75– 65
		CMC No	**Material**		**Specific cutting force k_c 1 N/mm²**	**Hardness Brinell HB**	**mc**	**BASIC GRADES H10**	**CD10**
								Feed, f_z or h_{ex} 0,1– 0,15– 0,2	0,1– 0,15– 0,2
								Cutting speed, v_c m/min	
	-N	30.11 30.12			400 650	60 100		935– 870– 805 845– 785– 725	1875–1740–1615 1695–1565–1455
		30.21 30.22			600 700	75 90	0,25 0,25	940– 870– 805 845– 785– 725	1880–1745–1615 1695–1570–1455
		30.3			350	30		945– 875– 810	1890–1755–1625
		30.41 30.42			700 700	130 130		375– 350– 325 285– 265– 245	755– 700– 645 565– 525– 485
		33.1 33.2 33.3			550 550 1350	110 90 100	0,25 0,25	470– 435– 405 470– 435– 405 325– 305– 285	945– 875– 810 940– 875– 805 655– 610– 565

SANDVIK Coromant

BASIC GRADES

4030	4040	H13A	690	CB50			
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,1– 0,2– 0,4	0,1– 0,2– 0,4	0,1– 0,2– 0,4	0,1– 0,2– 0,3	0,1– 0,15– 0,2			
Cutting speed, v_c m/min							
215– 175– 115 175– 145– 95	195– 160– 105 160– 130– 85	120– 105– 75 95– 85– 65	545– 445– 365 455– 370– 305	– – – –			
230– 190– 125 185– 155– 105	215– 175– 115 170– 135– 95	130– 110– 85 105– 90– 65	595– 485– 400 475– 395– 325	845– 725– 365 675– 575– 305			
145– 120– 80 135– 110– 75	135– 105– 75 125– 100– 65	80– 70– 50 75– 65– 48	375– 305– 255 345– 285– 235	– – 495– 420– 360			

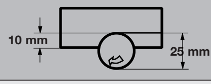
BASIC GRADES

1025	530	CB50					
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,07– 0,1– 0,12	0,07– 0,1– 0,12	0,07– 0,12– 0,2					
Cutting speed, v_c m/min							
41– 37– 35	80– 55– 70	160– 140– 115					
75– 70– 65	155– 140– 135	305– 265– 215					

BASIC GRADES

530	H13A	1025					
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,1– 0,15– 0,2	0,1– 0,15– 0,2	0,1– 0,15– 0,2					
Cutting speed, v_c m/min							
1035– 955– 885 930– 865– 800	750– 695– 645 675– 625– 580	985– 915– 845 885– 825– 765					
1035– 955– 885 930– 865– 800	750– 695– 645 675– 625– 580	985– 915– 845 890– 825– 765					
1040– 965– 895	755– 700– 650	995– 920– 855					
415– 385– 355 310– 285– 265	300– 280– 260 225– 210– 195	395– 365– 340 295– 275– 255					
515– 480– 445 515– 480– 445 365– 335– 310	375– 350– 325 375– 345– 325 265– 245– 225	5495– 455– 425 495– 455– 425 345– 320– 295					

ENDMILLING



ISO	CMC No.	Material		Specific cutting force k_c 1 (N/mm²)	Hardness Brinell (HB)	mc	Basic grades 4020 – Feed, f_z or h_{ex} 0,1–	4020 0,2–	4020 0,3	4030 0,05–	4030 0,1–	4030 0,15
							Cutting speed, v_c m/min					
P		**Steel**										
	01.1	**Unalloyed**	C = 0,10 – 0,25 %	1500	125	0,25	490–	405–	330	430–	425–	415
	01.2		C = 0,25 – 0,55 %	1600	150	0,25	440–	360–	295	385–	380–	375
	01.3		C = 0,55 – 0,80 %	1700	170	0,25	415–	340–	280	365–	355–	350
	01.4			1800	210	0,25	365–	295–	245	320–	315–	305
	01.5			2000	300	0,25	265–	220–	180	235–	230–	225
	02.1	**Low-alloy** (alloying elements ≤ 5 %)	Non-hardened	1700	175	0,25	345–	285–	230	305–	295–	290
	02.2		Hardened and tempered	2000	275	0,25	245–	195–	165	215–	215–	205
	02.2			2300	350	0,25	195–	160–	130	175–	165–	165
	03.11	**High-alloy** (alloying elements > 5 %)	Annealed	1950	200	0,25	295–	245–	200	230–	225–	220
	03.13		Hardened tool steel	2150	200	0,25	245–	175–	145	190–	185–	185
	03.21			2900	300	0,25	185–	155–	125	165–	165–	155
	03.22			3100	380	0,25	115–	95–	75	105–	100–	100
	06.1	**Castings**	Unalloyed	1400	150	0,25	350–	285–	235	310–	305–	295
	06.2		Low-alloy (alloying elements ≤ 5 %)	1600	200	0,25	275–	225–	185	245–	240–	235
	06.3		High-alloy, alloying elements > 5 %)	1950	200	0,25	205–	165–	135	180–	175–	175

ISO	CMC No.	Material		Specific cutting force k_c 1 (N/mm²)	Hardness Brinell (HB)	mc	Basic grades 2030 – Feed, f_z or h_{ex} 0,05–	2030 0,15–	2030 0,25	2040 0,1–	2040 0,2–	2040 0,3
							Cutting speed, v_c m/min					
M		**Stainless steel**										
	05.11	Ferritic/martensitic	Non-hardened	1800	200	0,21	240–	190–	155	235–	190–	155
	05.12		PH-hardened	2800	330	0,21	170–	135–	105	165–	130–	105
	05.13		Hardened	2300	330	0,21	175–	140–	115	175–	135–	110
	05.21	Austenitic	Non-hardened	2000	200	0,21	235–	185–	150	200–	160–	125
	05.22		PH-hardened	2800	330	0,21	165–	130–	105	155–	125–	100
	05.51	Austenitic-ferritic (Duplex)	Non-weldable ≥ 0,05%C	2000	230	0,21	195–	155–	125	165–	135–	105
	05.52		Weldable < 0,05%C	2400	260	0,21	165–	130–	105	135–	105–	85
		Stainless steel – Cast										
	15.11		Non-hardened	1700	200	0,25	215–	170–	135	210–	170–	135
	15.12		PH-hardened	2500	330	0,25	145–	115–	95	145–	115–	90
	15.13		Hardened	2100	330	0,25	160–	125–	105	160–	125–	100
	15.21	Austenitic	Austenitic	1800	200	0,25	225–	175–	145	190–	155–	125
	15.22		PH-hardened	2500	330	0,25	145–	115–	95	145–	115–	90
	15.51	Austenitic-ferritic (Duplex)	Non-weldable ≥ 0,05%C	1800	230	0,25	185–	145–	115	155–	125–	100
	15.52		Weldable < 0,05%C	2200	260	0,25	150–	120–	95	125–	100–	80

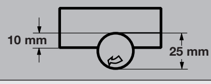
ISO	CMC No.	Material		Specific cutting force k_c 1 (N/mm²)	Hardness Brinell (HB)	mc	Basic grades 1025 – Feed, f_z or h_{ex} 0,05–	1025 0,1–	1025 0,15	H10F 0,05–	H10F 0,1–	H10F 0,15
							Cutting speed, v_c m/min					
-S		**Heat resistant super alloys**										
	20.11	Iron base	Annealed or solution treated	2400	200	0,25	70–	70–	65	65–	60–	60
	20.12		Aged or solution treated and aged	2500	280	0,25	55–	50–	50	46–	45–	45
	20.21	Nickel base	Annealed or solution treated	2650	250	0,25	65–	65–	65	60–	55–	55
	20.22		Aged or solution treated and aged	2900	350	0,25	42–	42–	41	36–	36–	35
	20.24		Cast or cast and aged	3000	320	0,25	50–	50–	50	45–	44–	44
	20.31	Cobalt base	Annealed or solution treated	2700	200	0,25	30–	29–	29	26–	26–	25
	20.32		Solution treated and aged	3000	300	0,25	21–	21–	20	19–	18–	18
	20.33		Cast or cast and aged	3100	320	0,25	20–	19–	19	18–	17–	17
		Titanium alloys[1)]			Rm[2)]							
	23.1		Commercial pure (99,5% Ti)	1300	400	0,23	145–	145–	145	130–	125–	125
	23.21		α, near α and α+β alloys, annealed	1400	950	0,23	75–	75–	75	65–	65–	65
	23.22		α+β alloys in aged cond., β alloys, annealed or aged	1400	1050	0,23	65–	65–	60	55–	55–	55

[1)] 45–60° entering angle, positive cutting geometry and coolant should be used.
[2)] Rm = ultimate tensile strength measured in MPa.

Basic grades							
4040	1025	3040	530	SM30			
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)Feed, f_n mm/r							
0,05– 0,1– 0,15	0,05– 0,15– 0,25	0,05– 0,1– 0,15	0,05– 0,11– 0,2	0,08–0,15–0,25			
Cutting speed, v_c m/min							
365– 350– 335	365– 350– 335	465– 350– 335	510– 495– 475	300– 295– 285			
330– 315– 305	330– 315– 305	415– 315– 305	460– 450– 430	270– 265– 255			
310– 295– 285	310– 295– 285	395– 295– 285	435– 425– 405	255– 245– 240			
275– 260– 250	275– 260– 250	345– 260– 245	375– 370– 355	225– 215– 210			
200– 195– 185	200– 195– 185	255– 195– 185	275– 275– 260	165– 160– 155			
255– 245– 235	255– 245– 235	325– 245– 235	355– 350– 335	210– 205– 195			
180– 175– 165	180– 175– 165	225– 175– 165	250– 255– 235	145– 145– 140			
145– 140– 135	145– 140– 135	185– 140– 135	205– 195– 190	120– 115– 115			
195– 185– 175	195– 185– 175	245– 185– 175	270– 265– 255	160– 155– 150			
160– 155– 145	160– 155– 145	205– 155– 145	225– 220– 210	135– 130– 125			
140– 135– 130	140– 135– 130	175– 135– 130	195– 190– 185	115– 115– 110			
85– 85– 80	85– 85– 80	110– 85– 80	125– 120– 115	75– 70– 65			
265– 250– 240	265– 250– 240	330– 250– 240	365– 355– 340	215– 210– 205			
205– 200– 190	205– 200– 190	265– 200– 190	290– 285– 275	170– 165– 160			
155– 145– 140	155– 145– 140	195– 145– 140	215– 205– 195	125– 125– 115			

Basic grades							
4040	1025	530	4030				
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,05– 0,12– 0,2	0,05– 0,12– 0,2	0,05– 0,1– 0,15	0,05– 0,1– 0,15				
Cutting speed, v_c m/min							
245– 235– 225	255– 245– 235	350– 340– 335	305– 295– 285				
135– 135– 125	145– 140– 135	195– 195– 185	170– 165– 165				
145– 140– 135	150– 145– 140	210– 200– 195	175– 175– 170				
225– 220– 210	235– 230– 220	325– 315– 310	– –				
135– 125– 125	140– 135– 130	190– 185– 180	– –				
220– 215– 205	235– 225– 215	315– 310– 300	– –				
195– 190– 185	205– 200– 190	280– 275– 265	– –				
220– 215– 205	230– 225– 215	315– 305– 300	275– 265– 260				
110– 105– 100	115– 110– 105	155– 155– 145	135– 345– 130				
125– 115– 115	125– 125– 115	175– 170– 165	150– 145– 145				
220– 215– 205	230– 225– 215	315– 305– 300	–				
110– 105– 105	115– 110– 105	155– 155– 150	– –				
165– 155– 150	170– 165– 155	230– 225– 220	– –				
165– 160– 155	175– 165– 160	235– 230– 225	– –				

Basic grades							
H13A							
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,05– 0,1– 0,15							
Cutting speed, v_c m/min							
65– 65– 65							
50– 49– 48							
65– 65– 65							
40– 40– 39							
50– 49– 49							
28– 28– 27							
20– 20– 19							
20– 19– 19							
145– 140– 135							
75– 75– 70							
60– 55– 55							

ISO		CMC No.	Material	 	Specific cutting force k_c 1 N/mm²	Hardness Brinell HB	mc	BASIC GRADES 3040	4030
								Feed, f_z or h_{ex} 0,1– 0,2– 0,3	0,1– 0,2– 0,3
								Cutting speed, v_c m/min	
K		07.1 07.2	Malleable cast iron	Ferritic (short chipping) Pearlitic (long chipping)	800 900	130 230	0,28 0,28	280– 265– 255 230– 220– 210	245– 235– 225 205– 195– 185
		08.1 08.2	Grey cast iron	Low tensile strength High tensile strength	900 1100	180 245	0,28 0,28	305– 290– 275 245– 235– 225	270– 255– 245 215– 205– 195
		09.1 09.2	Nodular cst iron	Ferritic Pearlitic	900 1350	160 250	0,28 0,28	190– 185– 175 175– 170– 160	170– 165– 155 155– 150– 145
		CMC No.	Material		Specific cutting force k_c 1 N/mm²	Hardness Brinell HB	mc	BASIC GRADES 4020	3040
								Feed, f_z or h_{ex} 0,07– 0,1– 0,12	0,07– 0,1– 0,12
								Cutting speed, v_c m/min	
	-H	04.1	Extra hard steel Hard steel	Hardened and tempered Hardened and tempered	4200	59 HRC	0,25	95– 90– 90	55– 55– 55
		10.1	Chilled cast iron	Cast or cast and aged	2200	400	0,28	175– 175– 175	105– 100– 100
		CMC No	Material		Specific cutting force k_c 1 N/mm²	Hardness Brinell HB	mc	BASIC GRADES H10F	H13A
								Feed, f_z or h_{ex} 0,05– 0,1– 0,15	0,05– 0,1– 0,15
								Cutting speed, v_c m/min	
	-N	30.11 30.12	Aluminium alloys	Wrought or wrought and coldworked, non-aging Wrought or wrought and aged	400 650	60 100		1075– 1055– 1035 965– 955– 935	860– 845– 830 775– 760– 745
		30.21 30.22	Aluminium alloys	Cast, non-aging Cast or cast and aged	600 700	75 90	0,25 0,25	1075– 1065– 1040 970– 955– 935	860– 845– 830 775– 765– 750
		30.3			350	30		1085– 1065– 1045	865– 850– 835
		30.41 30.42	Aluminium alloys	Cast, 13–15% Si Cast, 16–22% Si	700 700	130 130		435– 425– 415 325– 315– 315	345– 340– 335 260– 255– 250
		33.1 33.2 33.3	Copper and copper alloys	Free cutting alloys, ≥1% Pb Brass, leaded bronzes, ≤1% Pb Bronze and non-leadad copper incl. electrolytic copper	550 550 1350	110 90 100	0,25 0,25	540– 530– 520 535– 530– 520 375– 370– 365	430– 425– 415 430– 425– 415 300– 295– 290

BASIC GRADES							
4040	H13A						
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,1– 0,2– 0,3	0,05– 0,15– 0,25						
Cutting speed, v_c m/min							
225– 215– 205 185– 175– 170	135– 135– 125 115– 110– 105						
245– 235– 225 195– 185– 180	150– 145– 140 120– 115– 110						
155– 145– 140 145– 135– 130	95– 90– 85 85– 85– 80						

BASIC GRADES							
1025							
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,07- 0,1- 0,12							
Cutting speed, v_c m/min							
47– 46– 46							
90– 85– 85							

BASIC GRADES							
530	1025						
Feed/tooth (f_z, mm/tooth) or max chip thickness (h_{ex}, mm)							
0,05– 0,1– 0,15	0,05– 0,1– 0,15						
Cutting speed, v_c m/min							
1185– 1165– 1145 1165– 1045– 1030	1125– 1110– 1090 1025– 1000– 985						
1185– 1165– 1145 1165– 1045– 1030	1130– 1110– 1090 1015– 1000– 985						
1190– 1170– 1150	1135– 1115– 1095						
475– 465– 460 355– 350– 345	455– 445– 435 340– 335– 330						
595– 585– 575 595– 585– 575 415– 405– 400	565– 555– 545 565– 555– 545 395– 385– 380						

SOLID ENDMILLS

General purpose milling with grades GC1010 and GC1020


R216.32-...N 1020


R216.33-...P 1020


R216.33-...N 1020


R216.35-...N 1020

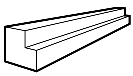

R216.34-...N 1020


R215.36-...L 1020

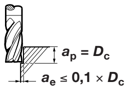

R215.3X-...H 1010

Only for hardened steel 55–63 HRC.
Max a_e = 0,05 x D_c.
Reduce f_z to 40%.

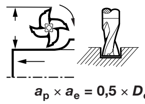
Materials



Finishing



Roughing – slotting



ISO	CMC No	Material	HB	Finishing: Cutting speed v_c m/min	Roughing – slotting: Cutting speed v_c m/min
P	01.1	Unalloyed steel	125	200–350	125–220
	01.2		150	180–250	120–190
	02.1	Low alloy steel	175	140–240	90–160
	02.2		330	120–200	80–120
	03.11	High alloy steel	200	140–190	90–130
M	05.11[1]	Stainless steel	200	90–160	40– 90
	05.21[1]		200	80–120	50– 90
	05.51[1]		230	60– 90	40– 60
	20.22	Heat resistant alloys	350	40– 50	20– 30
	23.22	Titanium alloys	350	50– 80	50– 60
K	04	Hard steel	HRC55	40– 70	30– 50
	04		HRC63	30– 50	-
	07.1	Malleable cast iron	130	170–250	130–190
	07.2		230	130–190	100–130
	09.1	Nodular SG iron	160	200–300	130–190
	09.2		250	150–200	100–140
	08.1	Cast iron	180	150–220	100–150
	30.22	Aluminium alloys (cast)	90	1000	1000

Finishing:

D_c mm	Feed/tooth f_z mm/z
2	0,01–0,02
3	0,01–0,02
4	0,02–0,04
5	0,03–0,06
6	0,03–0,07
7	0,04–0,08
8	0,05–0,09
9	0,07–0,10
10	0,07–0,12
12	0,08–0,13
14	0,08–0,14
16	0,09–0,15
18	0,10–0,16
20	0,10–0,16
25	0,10–0,16

Roughing – slotting:

D_c mm	Feed/tooth f_z mm/z
2	0,005–0,015
3	0,01–0,02
4	0,015–0,03
5	0,02–0,03
6	0,02–0,04
7	0,02–0,04
8	0,03–0,045
9	0,03–0,045
10	0,035–0,05
12	0,035–0,06
14	0,04–0,07
16	0,05–0,08
18	0,06–0,08
20	0,06–0,08
25	0,06–0,09

Calculations

[1] For ramping and drilling in stainless steel

$0,8 \times v_c$
$0,15 \times v_f$

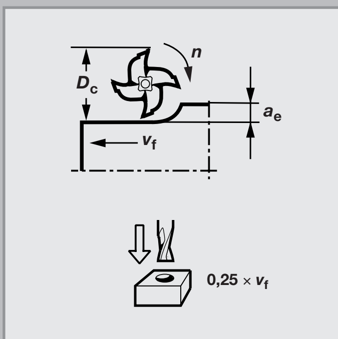


Table feed

$v_f = f_z \times n \times z_n$ mm/min

Cutting speed

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

Spindle speed

$$n = \frac{v_c \times 1000}{\pi \times D_c} \text{ rpm}$$

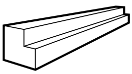
SOLID ENDMILLS

High speed machining (HSM) with grade GC1010

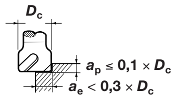
R216.22-...L 1010

R216.24-...L 1010

Materials



ISO	CMC No		HB	Cutting speed v_c m/min
P	01.1	Unalloyed steel	125	300–500
	01.2		150	250–450
	02.1	Low alloy steel	175	200–400
	02.2		330	180–330
	03.11	High alloy steel	200	200–330
M	05.11	Stainless steel	200	150–200
	05.21		200	120–170
	05.51		230	100–150
	20.22	Heat resistant alloys	350	40– 70
	23.22	Titanium alloys	350	70–120
K	04	Hard steel	HRC55	150–250
	04		HRC63	90–150
	07.1	Malleable cast iron	130	200–450
	07.2		230	300–450
	09.1	Nodular SG iron	160	400–500
	09.2		250	200–350
	08.1	Cast iron	180	300–500
	30.22	Aluminium alloys (cast)	90	1000



D_c mm	Feed/tooth f_z mm/z
3	0,03–0,04
4	0,04–0,07
5	0,05–0,09
6	0,05–0,10
8	0,06–0,11
10	0,07–0,12
12	0,08–0,13
16	0,09–0,16

Calculations

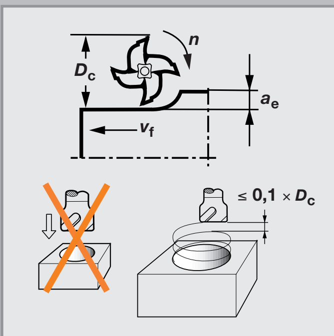


Table feed

$v_f = f_z \times n \times z_n$ mm/min

Cutting speed

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

Spindle speed

$$n = \frac{v_c \times 1000}{\pi \times D_c} \text{ rpm}$$

SOLID ENDMILLS

Copy milling – high speed machining (HSM), grades GC1010 and GC1020


R216.42-...L 1010


R216.64-...L 100


R216.42-...N 1020


R216.42-...H 1010

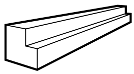

R216.42-...L 1010


R216.44-...L 1020

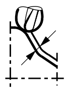

R216.62-...L 1010


R216.44-...L 1010

Materials



ISO	CMC No		HB	Coromant grade GC1010 Effective cutting speed v_c m/min	Coromant grade GC1020 Effective cutting speed v_c m/min
P	01.1	Unalloyed steel	125	300–500	240–400
	01.2		150	250–450	200–360
	02.1	Low alloy steel	175	200–400	160–320
	02.2		330	180–330	140–260
	03.11	High alloy steel	200	200–330	160–260
M	05.11	Stainless steel	200	150–200	120–160
	05.21		200	120–170	100–140
	05.51		230	100–150	80–120
	20.22	Heat resistant alloys	350	40– 70	30– 60
	23.22	Titanium alloys	350	70–120	50– 90
K	04	Hard steel	HRC55	150–250	120–200
	04		HRC63	90–150	70–120
	07.1	Malleable cast iron	130	200–450	160–360
	07.2		230	300–450	240–360
	09.1	Nodular SG iron	160	400–500	320–400
	09.2		250	200–350	160–280
	08.1	Cast iron	180	300–500	240–400
	30.22	Aluminium alloys (cast)	90	1000	800



$a_p \leq 0{,}05 \times D_c$

D_c mm	Feed per tooth f_z mm/z
2	0,015–0,020
3	0,03 –0,04
4	0,04 –0,07
5	0,05 –0,09
6	0,05 –0,10
7	0,06 –0,10
8	0,06 –0,11
9	0,06 –0,12
10	0,07 –0,12
12	0,08 –0,13
14	0,08 –0,15
16	0,09 –0,16
18	0,09 –0,16
20	0,09 –0,16

Calculations

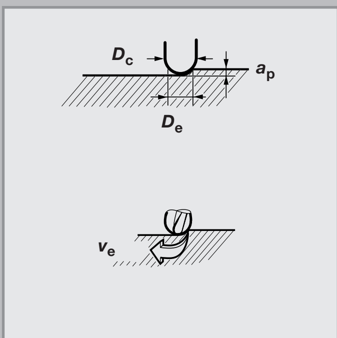


Table feed

$v_f = f_z \times n \times z_n$ mm/min

Resultant cutting speed

$$v_e = \frac{n \times \pi \times D_e}{1000} \text{ m/min}$$

Spindle speed

$$n = \frac{v_e \times 1000}{\pi \times D_c} \text{ rpm}$$

Effective cutting diameter

$$D_e = 2\sqrt{a_p(D_c - a_p)}$$

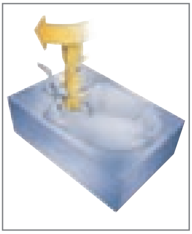
MATERIAL CROSS REFERENCE LIST

ISO	Coromant Material Classification (CMC)	Country: Great Britain, Standard: BS	Great Britain: EN	Sweden: SS	USA: AISI/SAE	Germany: W.-nr.	Germany: DIN	France: AFNOR	Italy: UNI	Spain: UNF	Japan: JIS
P	**Structural and constructional steel**										
	01.1	080M15	-	1350	1015	1.0401	C15	CC12	C15C16	F.111	-
	01.1	050A20	2C	1450	1020	1.0402	C22	CC20	C20C21	F.112	-
	01.2	060A35	-	1550	1035	1.0501	C35	CC35	C35	F.113	-
	01.2	080M46	-	1650	1045	1.0503	C45	CC45	C45	F.114	-
	01.3	070M55	-	1655	1055	1.0535	C55	-	C55	-	-
	01.3	080A62	43D	-	1060	1.0601	C60	CC55	C60	-	-
	01.1	230M07	-	1912	1213	1.0715	9SMn28	S250	CF9SMn28	11SMn28	SUM22
	01.1	-	-	1914	12L13	1.0718	9SMnPb28	S250Pb	CF9SMnPb28	11SMnPb28	SUM22L
	01.1	-	-	-	-	1.0722	10SPb20	10PbF2	CF10SPb20	10SPb20	-
	01.2	212M36	8M	1957	1140	1.0726	35S20	35MF4	-	F210G	-
	01.1	240M07	1B	-	1215	1.0736	9SMn36	S 300	CF9SMn36	12SMn35	-
	01.1	-	-	1926	12L14	1.0737	9SMnPb36	S300Pb	CF9SMnPb36	12SMnP35	-
	02.1/02.2	250A53	45	2085	9255	1.0904	55Si7	55S7	55Si8	56Si7	-
	02.1/02.2	-	-	-	9262	1.0961	60SiCr7	60SC7	60SiCr8	60SiCr8	-
	01.1	080M15	32C	1370	1015	1.1141	Ck15	XC12	C16	C15K	S15C
	01.2	150M36	15	-	1039	1.1157	40Mn4	35M5	-	-	-
	01.1	-	-	–	1025	1.1158	Ck25	-	-	-	S25C
	01.2	–	–	2120	1335	1.1167	36MN5	40M5	–	36Mn5	SMn438(H)
	01.2	150M28	14A	-	1330	1.1170	28Mn6	20M5	C28Mn	-	SCMn1
	01.2	060A35	-	1572	1035	1.1183	Cf35	XC38TS	C36	-	S35C
	01.2	080M46	-	1672	1045	1.1191	Ck45	XC42	C45	C45K	S45C
	01.3	070M55	-	-	1055	1.1203	Ck55	XC55	C50	C55K	S55C
	01.2	060A52	-	1674	1050	1.1213	Cf53	XC48TS	C53	-	S50C
	01.3	080A62	43D	1678	1060	1.1221	Ck60	XC60	C60	-	S58C
	06.33	Z120M12	-	-	-	1.3401	G-X120Mn12	Z120M12	XG120Mn12	X120Mn12	SCMnH/1
	02.1/02.2	534A99	31	2258	52100	1.3505	100Cr6	100C6	100Cr6	F.131	SUJ2
	02.1/02.2	1501-240	-	2912	ASTM A204Gr.A	1.5415	15Mo3	15D3	16Mo3KW	16Mo3	-
	02.1/02.2	1503-245-420	-	-	4520	1.5423	16Mo5	-	16Mo5	16Mo5	-
	02.1/02.2	-	-	-	ASTM A350LF5	1.5622	14Ni6	16N6	14Ni6	15Ni6	-
	03.11	1501-509;510	-	-	ASTM A353	1.5662	X8Ni9	-	X10Ni9	XBNi09	-
	03.11	-	-	-	2515	1.5680	12Ni19	Z18N5	-	-	-
	02.2	640A35	111A	-	3135	1.5710	36NiCr6	35NC6	-	-	SNC236
	02.1/02.2	-	-	-	3415	1.5732	14NiCr10	14NC11	16NiCr11	15NiCr11	SNC415(H)
	02.1/02.2	655M13; 655A12	36A	-	3415;3310	1.5752	14NiCr14	12NC15	-	-	SNC815(H)
	02.1/02.2	816M40	110	-	9840	1.6511	36CrNiMo4	40NCD3	38NiCrMo4(KB)	35NiCrMo4	-
	02.1/02.2	805M20	362	2506	8620	1.6523	21NiCrMo2	20NCD2	20NiCrMo2	20NiCrMo2	SNCM220(H)
	02.1/02.2	311-Type 7	-	-	8740	1.6546	40NiCrMo22	-	40NiCrMo2(KB)	40NiCrMo2	SNCM240
	02.1/02.2	817M40	24	2541	4340	1.6582	35CrNiMo6	35NCD6	35NiCrMo6(KB)	-	-
	02.1/02.2	820A16	-	-	-	1.6587	17CrNiMo6	18NCD6	-	14NiCrMo13	-
	03.11	832M13	36C	-	-	1.6657	14NiCrMo134	-	15NiCrMo13	14NiCrMo131	-
	02.1/02.2	523M15	-	-	5015	1.7015	15Cr3	12C3	-	-	SCr415(H)
	02.1/02.2	530A32	18B	-	5132	1.7033	34Cr4	32C4	34Cr4(KB)	35Cr4	SCr430(H)
	02.1/02.2	530M40	18	-	5140	1.7035	41Cr4	42C4	41Cr4	42Cr4	SCr440(H)
	02.1/02.2	-	-	2245	5140	1.7045	42Cr4	-	-	42Cr4	SCr440
	02.1/02.2	(527M20)	-	2511	5115	1.7131	16MnCr5	16MC5	16MnCr5	16MnCr5	-
	02.1/02.2	527A60	48	-	5155	1.7176	55Cr3	55C3	-	-	SUP9(A)
	02.1/02.2	1717CDS110	-	2225	4130	1.7218	25CrMo4	25CD4	25CrMo4(KB)	55Cr3 AM26CrMo4	SCM420;SCM430
	02.1/02.2	708A37	19B	2234	4137;4135	1.7220	34CrMo4	35CD4	35CrMo4	34CrMo4	SCM432;SCCRM3
	02.1/02.2	708M40	19A	2244	4140;4142	1.7223	41CrMo4	42CD4TS	41CrMo4	42CrMo4	SCM 440
	02.1/02.2	708M40	19A	2244	4140	1.7225	42CrMo4	42CD4	42CrMo4	42CrMo4	SCM440(H)
	02.1/02.2	-	-	2216	-	1.7262	15CrMo5	12CD4	-	12CrMo4	SCM415(H)
	02.1/02.2	1501-620Gr27	-	-	ASTM A182 F11;F12	1.7335	13CrMo4 4	15CD3.5 15CD4.5	14CrMo4 5	14CrMo45	-
	02.1/02.2	722M24	40B	2240	-	1.7361	32CrMo12	30CD12	32CrMo12	F.124.A	-
	02.1/02.2	1501-622 Gr.31;45	- -	2218	ASTM A182 F.22	1.7380	10CrMo9 10	12CD9, 10 -	12CrMo9, 10	TU.H -	- -
	02.1/02.2	1503-660-440	-	-	-	1.7715	14MoV6 3	-	-	13MoCrV6	-
	02.1/02.2	735A50	47	2230	6150	1.8159	50CrV4	50CV4	50CrV4	51CrV4	SUP10
	02.1/02.2	905M39	41B	2940	-	1.8509	41CrAlMo7	40CAD6, 12	41CrAlMo7	41CrAlMo7	-
	02.1/02.2	897M39	40C	-	-	1.8523	39CrMoV13 9	-	36CrMoV12	-	-
	Tool steel										
	02.1/02.2	BL3	-	-	L3	1.2067	100Cr6	Y100C6	-	100Cr6	-
	03.11	BD3	-	-	D3	1.2080	X210Cr12	Z200C12	X210Cr13KU X250Cr12KU	X210Cr12	SKD1
	03.11	BH13	-	2242	H13	1.2344	X40CrMoV5 1	Z40CDV5	X35CrMoV05KU X40CrMoV511KU	X40CrMoV5	SKD61
	03.11	BA2	-	2260	A2	1.2363	X100CrMoV5 1	Z100CDV5	X100CrMoV51KU	X100CrMoV5	SKD12
	02.1/02.2	-	-	2140	-	1.2419	105WCr6	105WC13	10WCr6 107WCr5KU	105WCr5	SKS31 SKS2, SKS3

ISO	Coromant Material Classification (CMC)	Country									
		Great Britain		Sweden	USA	Germany		France	Italy	Spain	Japan
		Standard									
		BS	EN	SS	AISI/SAE	W.-nr.	DIN	AFNOR	UNI	UNF	JIS
P	03.11	-	-	2312	-	1.2436	X210CrW12	-	X215CrW12 1KU	X210CrW12	SKD2
	03.11	BS1	-	2710	S1	1.2542	45WCrV7	-	45WCrV8KU	45WCrSi8	-
	03.11	BH21	-	-	H21	1.2581	X30WCrV9 3 X30WCrV9 3KU	Z30WCV9	X28W09KU X30WCrV9 3KU	X30WCrV9	SKD5
	03.11	-	-	2310	-	1.2601	X165CrMo V 12	-	X165CrMoW12KU	X160CrMoV12	-
	03.11	401S45	52	-	HW3	1.4718	X45GrSi93	Z45CS9	X45GrSi8	F322	SUH1
	02.1/02.2	-	-	-	L6	1.2713	55NiCrMoV6	55NCDV7	-	F.520.S	SKT4
M	**Stainless and heat resistant materials**										
	05.11/15.11	403S17	-	2301	403	1.4000	X7Cr13	Z6C13	X6Cr13	F.3110	SUS403
						1.4001	X7Cr14	-	-	F.8401	-
	05.11/15.11	430S15	60	2320	430	1.4016	X8Cr17	Z8C17	X8Cr17	F3113	SUS430
	05.11/15.11	410S21	56A	2302	410	1.4006	X10Cr13	Z10C14	X12Cr13	F.3401	SUS410
	05.11/15.11	430S17	60	2320	430	-	X8Cr17	Z8C17	X8Cr17	F.3113	SUS430
	05.11/15.11	420S45	56D	2304	-	1.4034	X46Cr13	Z40CM Z38C13M	X40Cr14	F.3405	SUS420J2
	05.11/15.11	405S17	-	-	405	1.4002	-	Z8CA12	X6CrAl13	-	-
	05.11/15.11	420S37	-	2303	420	1.4021	-	Z20C13	X20Cr13	-	-
	05.11/15.11	431S29	57	2321	431	1.4057	X22CrNi17	Z15CNi6.02	X16CrNi16	F.3427	SUS431
	05.11/15.11	-	-	2383	430F	1.4104	X12CrMoS17	Z10CF17	X10CrS17	F.3117	SUS430F
	05.11/15.11	434S17	-	2325	434	1.4113	X6CrMo17	Z8CD17.01	X8CrMo17	-	SUS434
	05.11/15.11	425C11	-	-	-	1.4313	X5CrNi13 4	Z4CND13.4M	-	-	SCS5
	05.11/15.11	403S17	-	-	405	1.4724	X10CrA113	Z10C13	X10CrA112	F.311	SUS405
	05.11/15.11	430S15	60	-	430	1.4742	X10CrA118	Z10CAS18	X8Cr17	F.3113	SUS430
	05.11/15.11	443S65	59	-	HNV6	1.4747	X80CrNiSi20	Z80CSN20.02	X80CrSiNi20	F.320B	SUH4
	05.11/15.11	-	-	2322	446	1.4762	X10CrA124	Z10CAS24	X16Cr26	-	SUH446
	05.11/15.11	349S54	-	-	EV8	1.4871	X53CrMnNiN 21 9	Z52CMN21.09	X53CrMnNiN 21 9	-	SUH35, SUH36
	05.12/15.12	-	-	-	630	1.4542/1.4548	-	Z7CNU17-04	-	-	-
	05.21/15.21	304S11	-	2352	304L	1.4306	-	Z2CN18-10	X2CrNi18 11	-	-
	05.21/15.21	304S31	58E	2332/2333	304	1.4350	X5CrNi189	Z6CN18.09	X5CrNi18 10	F.3551 F.3541 F.3504	SUS304
	05.21/15.21	303S21	58M	2346	303	1.4305	X12CrNiS18 8	Z10CNF 18.09	X10CrNiS 18.09	F.3508	SUS303
	05.21/15.21	304S15	58E	2332	304	1.4301	X5CrNi189	Z6CN18.09	X5CrNi18 10	F.3551	SUS304
		304C12		2333				Z3CN19.10	-	-	SUS304L
	05.21/15.21	304S12	-	2352	304L	1.4306	X2CrNi18 9	Z2CrNi18 10	X2CrNi18 11	F.3503	SCS19
	05.21/15.21	-	-	2331	301	1.4310	X12CrNi17 7	Z12CN17.07	X12CrNi17 07	F.3517	SUS301
	05.21/15.21	304S62	-	2371	304LN	1.4311	X2CrNiN 18 10	Z2CN18.10	-	-	SUS304LN
	05.21/15.21	316S16	58J	2347	316	1.4401	X5CrNiMo 18 10	Z6CND17.11	X5CrNiMo17 12	F.3543	SUS316
	05.21/15.21	-	-	2375	316LN	1.4429	X2CrNiMoN 18 13	Z2CND17.13	-	-	SUS316LN
	05.21/15.21	316S13		2348	316L	1.4404	-	Z2CND17-12	X2CrNiMo1712	-	-
	05.21/15.21	316S13	-	2353	316L	1.4435	X2CrNiMo 18 12	Z2CND17.12 -	X2CrNiMo17 12 -	-	SCS16 SUS316L
	05.21/15.21	316S33	-	2343/2347	316	1.4436	-	Z6CND18-12-03	X8CrNiMo1713	-	-
	05.21/15.21	317S12	-	2367	317L	1.4438	X2CrNiMo 18 16	Z2CND19.15	X2CrNiMo18 16	-	SUS317L
	05.51/15.51*)	-	-	-	S31500	1.4417	X2CrNiMoSi 19 5	-	-	-	-
	05.51/15.51*)	-	-	2324	S32900	-	X8CrNiMo 27 5	-	-	-	-
	05.52/15.52*)	-	-	2327	S32304	-	X2CrNiN 23 4	Z2CN23-04AZ	-	-	-
	05.52/15.52*)	-	-	2328	-	-	-	-	-	-	-
	05.52/15.52*)	-	-	2377	S31803	-	X2CrNiMoN 22 53	Z2CND22-05-03	-	-	-
	05.21/15.21	321S12	58B	2337	321	1.4541	X10CrNiTi 18 9	Z6CNT18.10	X6CrNiTi18 11	F.3553 F.3523	SUS321
	05.21/15.21	347S17	58F	2338	347	1.4550	X10CrNiNb 18 9	Z6CNNb18.10	X6CrNiNb18 11	F.3552 F.3524	SUS347
	05.21/15.21	320S17	58J	2350	316Ti	1.4571	X10CrNiMoTi 18 10	Z6NDT17.12	X6CrNiMoTi17 12 17 12	F.3535	-
	05.21/15.21	-	-	-	318	1.4583	X10CrNi MoNb 18 12	Z6CNDNb 17 13B	X6CrNiMoNb	- 17 13	-
	05.21/15.21	309S24	-	-	309	1.4828	X15CrNiSi 20 12	Z15CNS20.12	-	-	SUH309
	05.21/15.21	310S24	-	2361	310S	1.4845	X12CrNi25 21	Z12CN25 20	X6CrNi25 20	F.331	SUH310
	05.22/15.22	316S111	-	-	17-7PH	1.4568/1.4504	-	Z8CNA17-07	X2CrNiMo1712	-	-
	05.23/15.23	-	-	2584	NO8028	1.4563	-	Z1NCDU31-27-03	-	-	-
	05.23/15.23	-	-	2378	S31254	-	-	Z1CNDU20-18-06AZ	-	-	-

*) Duplex stainless steel.

ISO	Coromant Material Classification (CMC)	Country: Great Britain		Sweden	USA	Germany		France	Italy	Spain	Japan
		Standard									
		BS	**EN**	**SS**	**AISI/SAE**	**W.-nr.**	**DIN**	**AFNOR**	**UNI**	**UNF**	**JIS**
M -S	**20.11**	-	-	-	330	1.4864	X12NiCrSi 36 16	Z12NCS35.16 -	-	-	SUH330
	20.11	330C11	-	-	-	1.4865	G-X40NiCrSi 38 18	-	XG50NiCr 39 19	-	SCH15
	20.21	-	-	-	5390A	2.4603	-	NC22FeD	-	-	
	20.21	-	-	-	5666	2.4856	NiCr22Mo9Nb	NC22FeDNB	-	-	
	20.21	HR5,203-4	-	-	-	2.4630	NiCr20Ti	NC20T	-	-	
	20.22	-	-	-	5660	LW2.4662	NiFe35Cr14MoTi	ZSNCDT42	-	-	
	20.22	3146-3	-	-	5391	LW2 4670	S-NiCr13A16MoNb	NC12AD	-	-	
	20.22	HR8	-	-	5383	LW2.4668	NiCr19Fe19NbMo	NC19eNB	-	-	
	20.22	3072-76	-	-	4676	2.4375	NiCu30Al	-	-	-	
	20.22	Hr401,601	-	-	-	2.4631	NiCr20TiAk	NC20TA	-	-	
	20.22	-	-	-	AMS 5399	2.4973	NiCr19Co11MoTi	NC19KDT	-	-	
	20.22	-	-	-	AMS 5544	LW2.4668	NiCr19Fe19NbMo	NC20K14	-	-	
	20.24	-	-	-	AMS 5397	LW2 4674	NiCo15Cr10MoAlTi	-	-	-	
	20.32	-	-	-	5537C	LW2.4964	CoCr20W15Ni	KC20WN	-	-	
		-	-	-	AMS 5772	-	CoCr22W14Ni	KC22WN	-	-	
	23.22	TA14/17	-	-	AMS R54520	-	TiAl5Sn2.5	T-A5E	-	-	
	23.22	TA10-13/TA28	-	-	AMS R56400	-	TiAl6V4	T-A6V	-	-	
	23.22	TA11	-	-	AMS R56401	-	TiAl6V4ELI	-	-	-	
	23.22	-	-	-	-	-	TiAl4Mo4Sn4Si0.5	-	-	-	
K	**Grey cast iron**										
					ASTM A48-76						
	08.1			01 00							
	08.1			01 10	No 20 B		GG 10	Ft 10 D			
	08.1	Grade 150		01 15	No 25 B		GG 15	Ft 15 D			
	08.1	Grade 220		01 20	No 30 B		GG 20	Ft 20 D			
	08.2	Grade 260		01 25	No 35 B		GG 25	Ft 25 D			
					No 40 B						
	08.2	Grade 300		01 30	No 45 B		GG 30	Ft 30 D			
	08.2	Grade 350		01 35	No 50 B		GG 35	Ft 35 D			
	08.2	Grade 400		01 40	No 55 B		GG 40	Ft 40 D			
	Nodular cast iron										
		2789;1973			A536-72			NF A32-201			
	09.1	SNG 420/12		07 17-02	60-40-18		GGG 40	FCS 400-12			
	09.1	SNG 370/17		07 17-12	-		GGG 40.3	FGS 370-17			
	09.1	-		07 17-15	-		GGG 35.3	-			
	09.1	SNG 500/7		07 27-02	80-55-06		GGG 50	FGS 500-7			
	09.2	SNG 600/3		07 32-03	-		GGG 60	FGS 600-3			
	09.2	SNG 700/2		07 37-01	100-70-03		GGG 70	FGS 700-2			
	Malleable cast iron										
					ASTM A47-74 A 220-76 2)						
	07.1	8 290/6		08 14			-	MN 32-8			
	07.1	B 340/12		08 15	32510		GTS-35	MN 35-10			
	07.2	P 440/7		08 52	40010		GTS-45				
	07.2	P 510/4		08 54	50005		GTS-55	MP 50-5			
		P 570/3		08 58	70003		GTS-65	MP 60-3			
	Aluminium alloys, cast										
	30.21/30.22	LM25		4244	356.1						
				4247	A413.0		GD-AlSi12				
		LM24		4250	A380.1		GD-AlSi8Cu3				
		LM20		4260	A413.1		G-AlSi12(Cu)				
		LM6		4261	A413.2		G-AlSi12				
		LM9		4253	A360.2		G-AlSi10Mg(Cu)				



NOTES