High speed machining and conventional die and mould machining
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 molds (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 molds it also includes time consuming hand finishing. Consequently, production costs can be high and lead times excessive.

Typical for the die and mold 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 mold 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 satis-
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 mold 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.

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 molds or components with a complex 3D geometry.

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. A nd 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:

- High Cutting Speed ($v_c$) Machining...
- High Spindle Speed ($n$) Machining...
- High Feed ($v_f$) Machining...
- High Speed and Feed Machining...
- High Productive Machining...

Metalworking World
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.

Application technology
To perform HSM applications it is necessary to use rigid and dedicated machine tools and controls with specific design features and options. All production equipment has to be designed for the specific process of HSM.

It is also necessary to use an advanced programming technique with the most favourable tool paths. The method to ensure constant stock for each operation and tool is a prerequisite for HSM and a basic criteria for high productivity and process security. Specific cutting and holding tools is also a must for this type of machining.

Characteristics of today’s HSM in hardened tool steel
Within the die & mold 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. A nd of course also to the dynamics and size of the machine tool.

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 also give as many practical guidelines for the application of HSM as possible.

True cutting speed
A s 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 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.

Shallow cuts
Very typical and necessary for HSM applications is that the depths of cut, a_e and a_p and the average chip thickness, h_m, 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 and in finishing and superfinishing operations in all types of materials.

Formulas:

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

**Effective cutting speed (\( v_e \))**

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

**Formula for feed speed.**

\[ V_f = f_z \times n \times z_n \text{ [mm/min]} \]

**Formula for material removal rate.**

\[ Q = \frac{a_p \times a_e \times v_f}{1000} \text{ [cm}^3/\text{min]} \]
tact length. A nother 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. E specially for roughing and semi-finishing. These should have maximum shank stability and bending stiffness. A tapered shank improves the rigidity. A nd so does also shanks made of heavy metal.

The geometry of the die or mold 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 (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 mold. A n 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 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: 48-58 HRC.

Roughing
True \( v_c \): 100 m/min, \( a_p \): 6-8% of the cutter diameter, \( f_z \): 0,05-0,1 mm/z

Semi-finishing
True \( v_c \): 150-200 m/min, \( a_p \): 3-4% of the cutter diameter, \( a_c \): 20-40% of the cutter diameter, \( f_z \): 0,05-0,15 mm/z

Finishing and super-finishing
True \( v_c \): 200-250 m/min, \( a_p \): 0,1-0,2 mm, \( a_c \): 0,1-0,2 mm, \( f_z \): 0,02-0,2 mm/z

The values are of course dependent of out-stick, overhang, stability in the application, cutter diameters, material hardness etc. They should be looked upon only 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 \( v_c \) has probably been calculated on the nominal diameter of the cutter. Not the effective diameter in cut.

HSM Cutting Data by Experience

<table>
<thead>
<tr>
<th>Material</th>
<th>Hardness</th>
<th>Conv. ( v_c )</th>
<th>HSM ( v_c, R )</th>
<th>HSM ( v_c, F )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel 01.2</td>
<td>150 HB</td>
<td>&lt;300</td>
<td>&gt;400</td>
<td>&lt;900</td>
</tr>
<tr>
<td>Steel 02.1/2</td>
<td>330 HB</td>
<td>&lt;200</td>
<td>&gt;250</td>
<td>&lt;600</td>
</tr>
<tr>
<td>Steel 03.11</td>
<td>300 HB</td>
<td>&lt;100</td>
<td>&gt;200</td>
<td>&lt;400</td>
</tr>
<tr>
<td>Steel 03.11</td>
<td>39 -48 HRc</td>
<td>&lt;80</td>
<td>&gt;150</td>
<td>&lt;350</td>
</tr>
<tr>
<td>Steel 04</td>
<td>48-58 HRc</td>
<td>&lt;40</td>
<td>&gt;100</td>
<td>&lt;250</td>
</tr>
<tr>
<td>GCI 08.1</td>
<td>180 HB</td>
<td>&lt;300</td>
<td>&gt;500</td>
<td>&lt;3000</td>
</tr>
<tr>
<td>Al/Kirksite</td>
<td>60-75 HB</td>
<td>&lt;1000</td>
<td>&gt;2000</td>
<td>&lt;5000</td>
</tr>
<tr>
<td>Non-ferr</td>
<td>100 HB</td>
<td>&lt;300</td>
<td>&gt;1000</td>
<td>&lt;2000</td>
</tr>
</tbody>
</table>

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.

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

Typical workpieces for HSM, forging die for an automotive component, molds for a plastic bottle and a headphone.
The application of High Speed Machining

In the article about HSM in the Nov/Dec issue 1998, the focus was on the background, characteristics and definitions of HSM. In this article the discussion will continue with the focus on application areas and the different demands put on machine and cutting tools. We will also shed light on some advantages and disadvantages with HSM.

Main application areas for HSM

Milling of cavities. As have been discussed in the previous article, 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 future articles.

Forging dies. Most forging dies are suitable for HSM due to the shallow geometry that many of them have. Short tools always result 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.

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.
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 etcetera. 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
- 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.

**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 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.

---

**The impulse law**

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

Time of engagement is reduced with HSM

- Impulse is reduced
- Deflection is reduced

---

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

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 process steps**
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.

**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...”

---

**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 accessibility (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%.

---

= Manual finishing

---

*Metalworking World*
Some typical demands on the machine tool and the data transfer in HSM (ISO/BT40 or comparable size)

- Spindle speed range \( \leq 40,000 \) 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)
- 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

Block processing speed
- 1-20 ms
- Increments (linear)
- 5-20 microns
- Or circular interpolation via NURBS (no linear increments)
- Data flow via R/S232
- 19.2 Kbit/s (20 ms)
- Data flow via Ethernet
- 250 Kbit/s (1 ms)

Exempel: Two edge cutter. Profile depth \( R_t = \frac{f_z^2}{4 \times D_c} \)

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

Run outs influence on tool-life

Tool life as a funktion of TIR of chipthickness.

Some specific demands on cutting tools made of solid carbide

- High precision grinding giving run-out lower than 3 microns
- As short outstick and overhang as possible, maximum stiff 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. Specific demands on cutting tools will be further discussed in coming articles.

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 work-
A 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). A dicing 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, cermet, 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.

A chieving 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 multilayer 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 has 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. A nether 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. A voiding 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.
Data transfer and tool balance important for HSM

To perform High Speed Machining (HSM) applications it is necessary to use dedicated machine tools. It is of equal importance to have computer software and machine controls with specific design features and options to ensure that correct tool paths can be programmed. In this article the importance of tool holding and balanced tools will be discussed.

This article is the third in a Series of articles dealing with die and mould making techniques from Sandvik Coromant.

CAD/CAM AND CNC STRUCTURES

HSM processes have 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 create 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.

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 bottlenecks 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...
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 imply 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 that the CNC is automatically adapting the specific axis 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.

Conventional CNC-technology does not know anything about cutting conditions. CNCs strictly care only about geometry. Today’s NC-programs contain 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.

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.

**CHOICE OF HOLDING TOOLS**

Just as the CAD/CAM and machine controls, are important to get good machining results and an optimized production, the holding/cutting tools are of equal importance.

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 1.0 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 on a balance machine.
- Reducing the unbalance by altering the tool, machining it to remove mass or by moving counterweights in a balancable toolholder.
- 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 motivated. But, some times there is no alternative to get the required quality.

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 another. 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 down. 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.
The aluminium workpiece on the picture illustrates tool balancing affecting surface finish. The balan-

cetable toolholder used to machine both halves of the surface were set to two unbalance values, 100 gmm and 1.4 g-mm. The more balanced tool produced the smoother surface. Con-

ditions of the two cuts were other-

wise 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 de-

mand holding the force from unba-

lance far less than the cutting force the machine will see anyway. In rea-

lity, an endmill run at 20000 rpm may not need to be balanced to any bet-

ter than 20 gmm, and 5 gmm is gene-

rally appropriate for much higher speeds. The diagram refers to unba-

lance force relating to tool and adap-

ter 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 g mm

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

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

Influence of system accuracy on unbalance for different tool interface.

<table>
<thead>
<tr>
<th>Angular error</th>
<th>Coromant Capto C5</th>
<th>HSK 50 form A</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unbalance</td>
<td>Up to 0.9 gmm</td>
<td>up to 3.3 gmm</td>
</tr>
<tr>
<td>Balance class</td>
<td>up to G1.5</td>
<td>up to G5.6</td>
</tr>
<tr>
<td>TIR</td>
<td>up to 3.5 $\mu$m</td>
<td>up to 13.4 $\mu$m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parallel error</th>
<th>Coromant Capto C5</th>
<th>HSK 50 form A</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unbalance</td>
<td>Up to 2.6 gmm</td>
<td>up to 9.6 gmm</td>
</tr>
<tr>
<td>Balance class</td>
<td>up to G4.4</td>
<td>up to G16.8</td>
</tr>
<tr>
<td>TIR</td>
<td>up to 4.2 $\mu$m</td>
<td>up to 16 $\mu$m</td>
</tr>
</tbody>
</table>

$n = 20000$ rpm, weight of adapter and tool $m = 1.2$ kg
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 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 is due to its polygon design superior when it comes to high torque and productive machining.

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 fourth, it gives high total stiffness in the assembly.

<table>
<thead>
<tr>
<th>Spindle speed</th>
<th>ISO40</th>
<th>HSK 50A</th>
<th>Coromant Capto C5</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>100%</td>
<td>100%</td>
<td>100%</td>
</tr>
<tr>
<td>20000</td>
<td>100%</td>
<td>95%</td>
<td>100%</td>
</tr>
<tr>
<td>25000</td>
<td>37%</td>
<td>91%</td>
<td>99%</td>
</tr>
<tr>
<td>30000</td>
<td>31%</td>
<td>83%</td>
<td>95%</td>
</tr>
<tr>
<td>35000</td>
<td>26%</td>
<td>72%</td>
<td>91%</td>
</tr>
<tr>
<td>40000</td>
<td>26%</td>
<td>67%</td>
<td>84%</td>
</tr>
</tbody>
</table>

**SURFACE CONTACT OF SPINDLE INTERFACE AT HIGH SPINDLE SPEED**

**COMPARISON BETWEEN HOLDERS FOR CLAMPING OF SHAFT TOOLS**

<table>
<thead>
<tr>
<th></th>
<th>Weldon/whistle-notch holder</th>
<th>Collet chuck Din 6499</th>
<th>Power chuck</th>
<th>HydroGrip Hydraulic chuck</th>
<th>Shrink fit holder</th>
<th>CoroGrip Hydro-mechanical chuck</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of operation</td>
<td>Heavy roughing-semi finishing</td>
<td>Roughing - Semi finishing</td>
<td>Heavy roughing - finishing</td>
<td>Finishing</td>
<td>Heavy roughing-finishing</td>
<td>Heavy roughing - finishing</td>
</tr>
<tr>
<td>Transmission torque</td>
<td>+++</td>
<td>++</td>
<td>++</td>
<td>+</td>
<td>+++</td>
<td>+++</td>
</tr>
<tr>
<td>Accuracy TIR 4 x D [mm]</td>
<td>0.01 - 0.02</td>
<td>0.01 - 0.03</td>
<td>0.003 - 0.010</td>
<td>0.003 - 0.008</td>
<td>0.003 - 0.006</td>
<td>0.003 - 0.006</td>
</tr>
<tr>
<td>Suitable for high speed</td>
<td>+</td>
<td>+</td>
<td>++</td>
<td>++</td>
<td>+++</td>
<td>+++</td>
</tr>
<tr>
<td>Maintenance</td>
<td>None required</td>
<td>Cleaning and changing collets</td>
<td>Cleaning and changing spare parts</td>
<td>None required</td>
<td>None required</td>
<td>None required</td>
</tr>
<tr>
<td>Possibility to use collets</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
</tr>
</tbody>
</table>
The roundness and concentricity are the most crucial factors for toolholders and not the tolerance class (AT).
When machining dies and moulds, and in any machining for that matter, the process has to be carefully planned to utilize the most efficient method possible and achieve the best result. In this fourth article from Sandvik Coromant regarding die and mould machining, the focus will be shifted somewhat from the high speed machining trend to the more basic planning stage of the machining process. Which of course applies to the HSM process as well.

AN OPEN-MINDED APPROACH

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.

Being a tooling company we are prepared to offer all our expertise in holding and cutting tools as well as in the cutting process in a partnership with the world-wide Die & Mould industry...

AN OPEN-MINDED APPROACH TO THE CHOICE OF METHODS, TOOL PATHS, MILLING AND HOLDING TOOLS

In today’s 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 thin-
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 work will also become more of a routine and 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. The finishing is always more productive with HSM. 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.
• The preparation (milled and parallel surfaces) and the fixturing of the blank 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 bonnet (big sized car) is roughly 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 mould 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, just 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 w. 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**
Ballnose 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 ballnose 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 ae/ap 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.

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. Ball nose endmills, on the other hand can never be replaced in close semi-finishing and finishing of complex 3D (shapes) geometries.

In the next article in the Die & Mould series “Application technologies” will be put in focus.
In this fifth article about die and mould making from Sandvik Coromant, application technology will be in focus. Some basic, but none the less very important parameters, will be discussed. Examples are down milling, copy milling and the importance of as little tool deflection as possible.

**Effective Diameter in Cut**
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, there will be severe miscalculations of the feed rate as it is dependent on the spindle speed 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.

![Diagram](image)

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 well below its capacity and recommended application range. This often leads to premature frittering and chipping of the cutting edge due to too low cutting speed and heat in the cutting zone.

**Avoid Excessive Deflection**
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.

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 (ap/ae). 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.
Down milling is important

Another application parameter of importance is the use of 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. In up milling this is when 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.

Bending

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Downmilling</th>
<th>Upmilling</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roughing</td>
<td>0.06 mm</td>
<td>0.02 mm</td>
</tr>
<tr>
<td>Finishing</td>
<td>0.05 mm</td>
<td>0.00 mm</td>
</tr>
</tbody>
</table>

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.
This is mainly due to the direction of the cutting forces. With a very sharp cutting edge, 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.

COPY MILLING AND PLUNGING
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. This means a risk of frittering at 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 damage 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. But, there will be a big contact length when the cutter hits the wall. This means a risk for vibration, deflection or even tool breakage if the feed speed does not decelerate fast enough. There is also a risk of pulling 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. A void 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.

If the spindle speed is limited in the machine, contouring will help to keep up the cutting speed. This type of tool path 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.
The tool-life will be considerably shorter if the tool has many entries and exits in the material. This 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 having big fluctuations.

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, \( f_z \), identical with the radial depth of cut, \( a_e \) (\( f_z = a_e \)).

This gives the following advantages:

- very smooth surface finish in all directions
- very competitive, short machining time
- very easy to polish, 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 \( f_z/a_e/R \)

For a long 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 interrupted or intermittent character cuts due to the usage of multi-teeth tools.

\[ f_z < a_e \]

\[ f_z = a_e \]
In this article in the series about die and mould making some basic factors of the milling process will be discussed, as well as some trouble shooting hints. It is important to know basic milling factors such as cutter pitch, entrance and exit of cut, positioning of the cutter, extended tools and how these parameters influence the cutting process in order to facilitate the understanding in upcoming articles.

**CUTTER 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 permits 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 and finishing 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 one means unequal spacing of teeth round the cutter and is a very effective means of coming to terms with problems of vibrations.
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 (D) an unfavourable contact between the edge of the insert and the workpiece results. Where the centre of the cutter is positioned inside the workpiece (E) 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.

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 per tooth 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 (a) will be longer if the cutter is moved away from the centre line (B) in either direction.

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 (D) an unfavourable contact between the edge of the insert and the workpiece results.

Where the centre of the cutter is positioned inside the workpiece (E) 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.
Cutting data should also be adapted to each tool length to keep up maximum productivity.

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

The basic action to be taken 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 coarse and differential pitch.

Use positive insert geometries.

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

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 per tooth). Or maintain the rotational speed and increase the table feed (= larger feed per tooth). Do not reduce the feed per tooth!

Reduce the radial and axial cutting depths.

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

With long overhangs, use tuned adapters in combination with coarse and differential pitched 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 speed. If vibrations 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
<table>
<thead>
<tr>
<th>Cause</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Poor clamping of the workpiece</strong></td>
<td>Establish the direction of the cutting forces and position the material accordingly.</td>
</tr>
<tr>
<td></td>
<td>Try to improve the clamping generally.</td>
</tr>
<tr>
<td></td>
<td>Reduce the cutting forces by reducing the radial and axial cutting depth.</td>
</tr>
<tr>
<td></td>
<td>Choose a milling cutter with a coarse pitch and positive design.</td>
</tr>
<tr>
<td></td>
<td>Choose positive inserts with small corner radius and small parallel lands.</td>
</tr>
<tr>
<td></td>
<td>Where possible, choose an insert grade with a thin coating and sharp cutting edge. If, necessary, choose an uncoated insert grade.</td>
</tr>
<tr>
<td></td>
<td>Avoid machining where the workpiece has poor support against cutting forces.</td>
</tr>
<tr>
<td><strong>Axially weak workpiece</strong></td>
<td>The first choice is a square shoulder facemill with positive insets.</td>
</tr>
<tr>
<td></td>
<td>Choose an insert geometry with sharp cutting edge and a large clearance angle, which produces low cutting forces.</td>
</tr>
<tr>
<td></td>
<td>Try to reduce the axial cutting forces by reducing the axial depth of cut, as well as using positive inserts with a small corner radius, small parallel lands and sharp cutting edges.</td>
</tr>
<tr>
<td><strong>Large overhang either on the machine spindle or the tool</strong></td>
<td>Always use a coarse and differentially pitched milling cutter.</td>
</tr>
<tr>
<td></td>
<td>Balance the cutting forces axially and radially. Use a 45-degree entering angle, large corner radius or round inserts.</td>
</tr>
<tr>
<td></td>
<td>Use inserts with a light cutting geometry.</td>
</tr>
<tr>
<td></td>
<td>Try to reduce the overhang, every millimetre counts.</td>
</tr>
<tr>
<td><strong>Square shoulder milling with a radially weak machine spindle</strong></td>
<td>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.</td>
</tr>
<tr>
<td></td>
<td>Choose positive and light cutting geometries.</td>
</tr>
<tr>
<td></td>
<td>Try up milling.</td>
</tr>
<tr>
<td><strong>Uneven table feed</strong></td>
<td>Try up milling.</td>
</tr>
<tr>
<td></td>
<td>Look at the possibility of adjusting the prestress of the washer to the ball-screw (CNC). Adjust the lock nut or exchange the screw on conventional machines.</td>
</tr>
</tbody>
</table>
Effective machining of corners & cavities

This is the last article in this series about die and mould making from Sandvik Coromant. In this article the most efficient way to machine corners are discussed as well as different methods for machining of cavities. Finally the advantages of machining in segments is also discussed.

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.

- A nother solution is to produce a bigger corner radius, via circular interpolation, than stated in the drawing. This can be favourable sometimes as it allows 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 restmilling of corners can also be performed by axial milling. It is im-
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 $a_p/a_e$ should be kept low to avoid deflection and vibration ($a_p/a_e$ 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 continuous 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%!

**RAMPING AND CIRCULAR INTERPOLATION**

Axial feed capability is an advantage in many operations. Holes, cavities as well as contours can be efficiently machined. Face milling 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 are 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. A milling cutters has limitations in the axial depth of cut and varies depending on the diameter, 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.
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, or several 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