The application of High Speed Machining

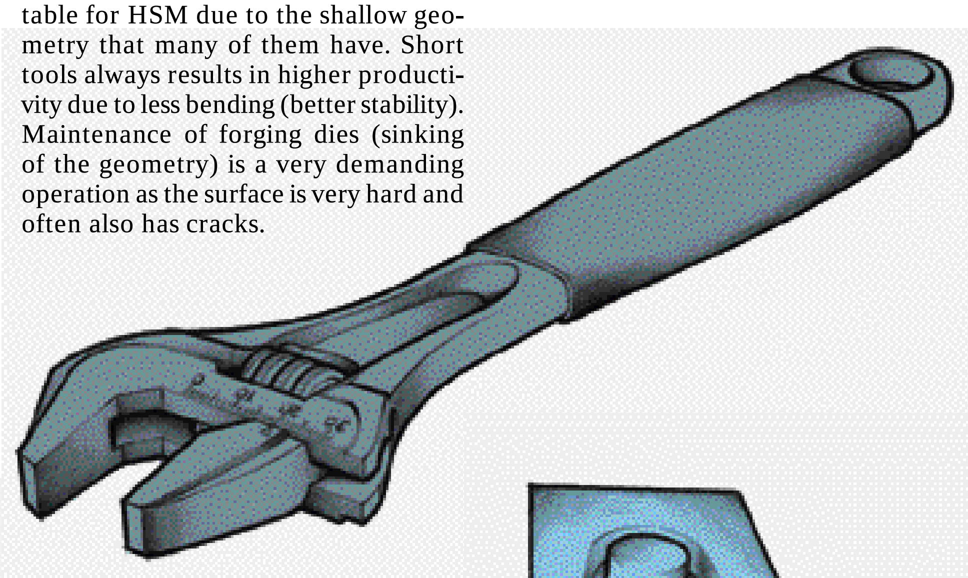
In the article about HSM in the Nov/Dec issue 1998, the focus was on the background, characteristics and definitions of HSM. In this article the discussion will continue with the focus on application areas and the different demands put on machine and cutting tools. We will also shed light on some advantages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

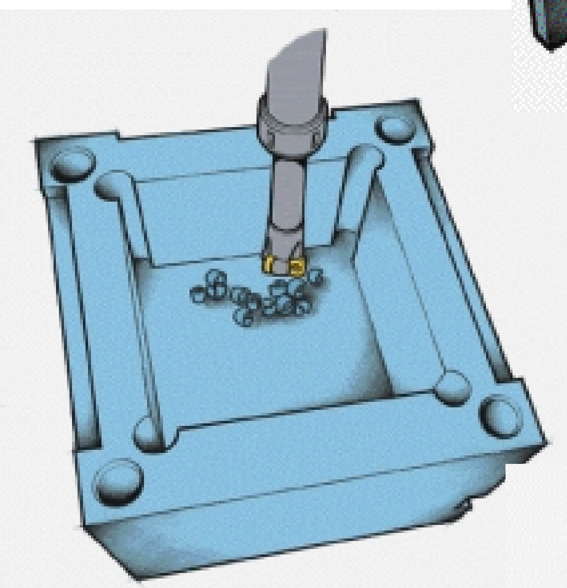
Main application areas for HSM

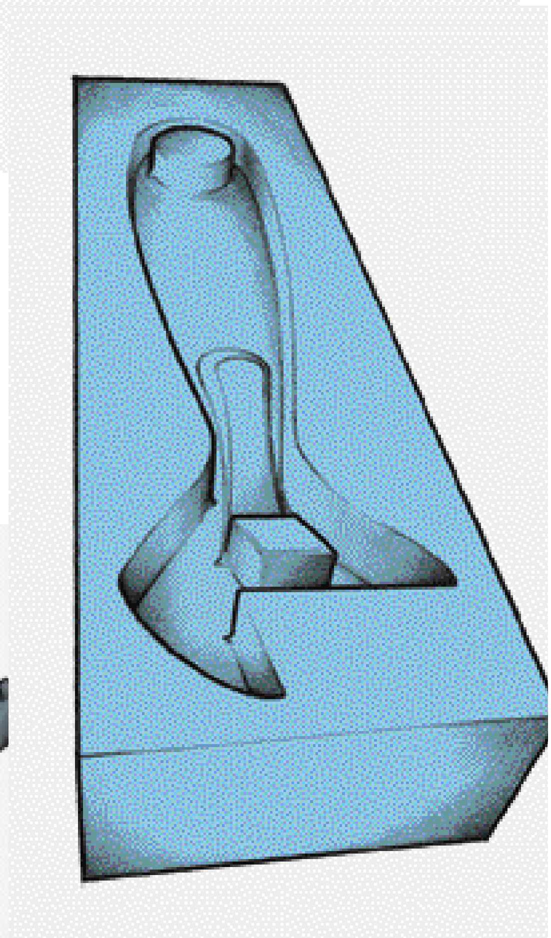
Milling of cavities. As have been discussed in the previous article, it is possible to apply HSM-technology (High Speed Machining) in qualified, high-alloy tool steels up to 60-63 HRc.

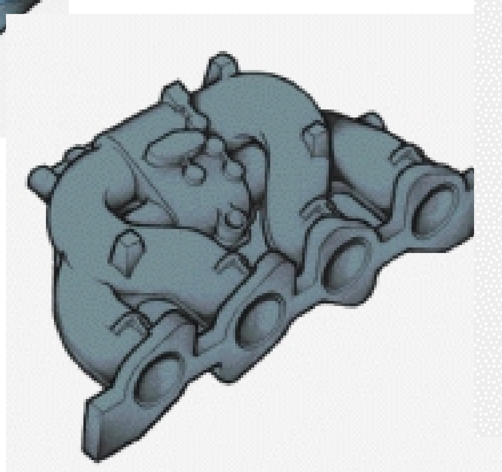
When milling cavities in such hard materials, it is crucial to select adequate cutting and holding tools for each specific operation; roughing, semi-finishing and finishing. To have success, it is also very important to use optimised tool paths, cutting data and cutting strategies. These things will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



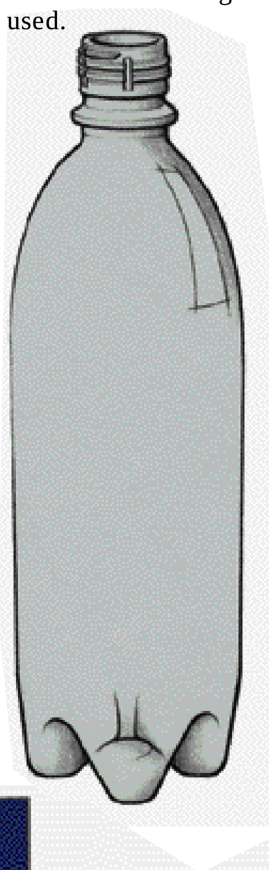




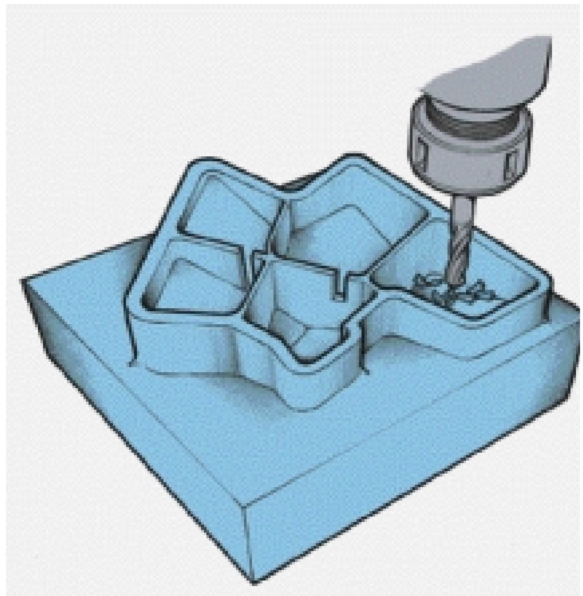


Die casting dies. This is an area where HSM can be utilised in a productive way as most die casting dies are made of demanding tool steels and have a moderate or small size.

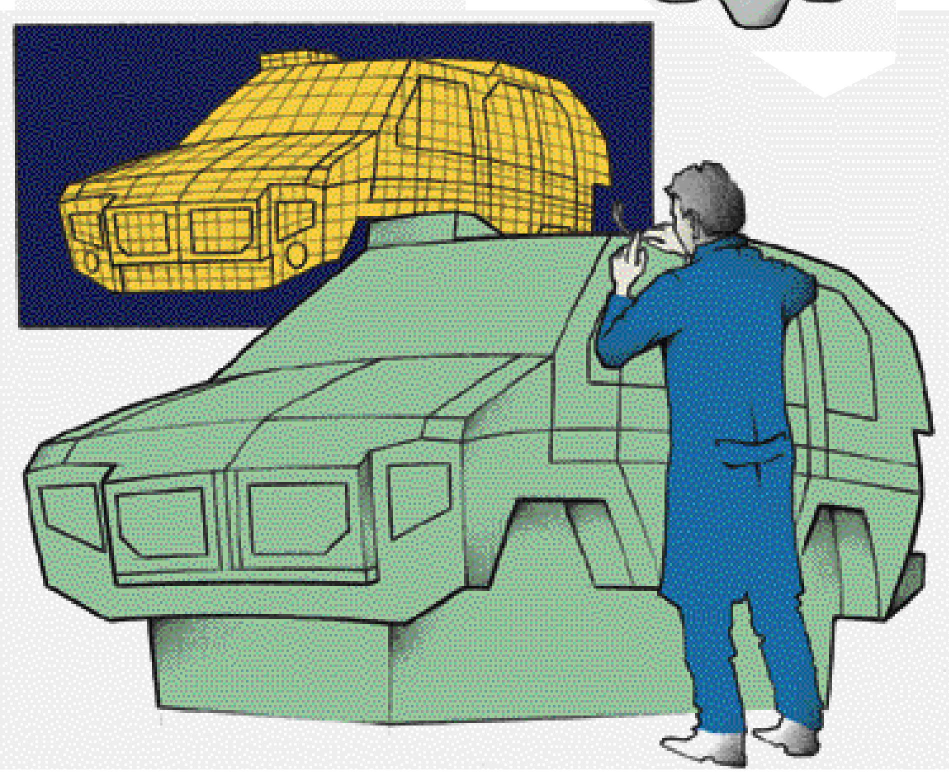
Injection moulds and blow moulds are also suitable for HSM, especially because of their (most often) small size. Which makes it economical to perform all operations (from roughing to finishing) in one set up. Many of these moulds have relatively deep cavities. Which calls for a very good planning of 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 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

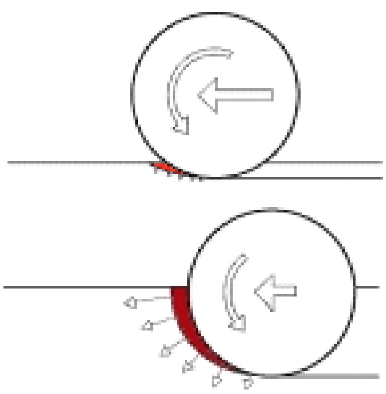
One of the main targets with HSM is to cut production costs via higher productivity. Mainly in finishing operations and often in hardened tool steel.

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

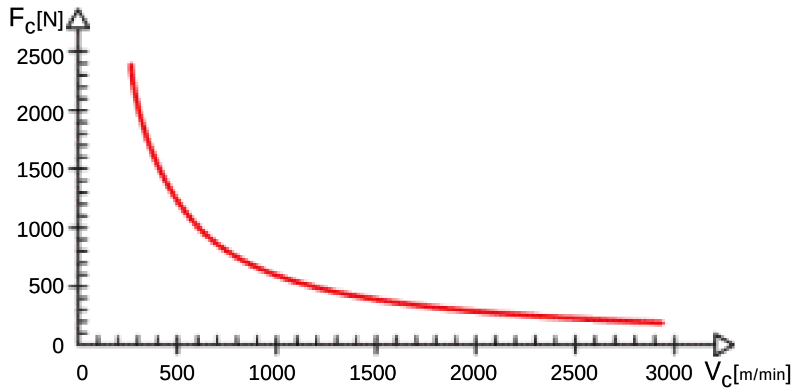
- production of dies or moulds in (a few or) a single set-up
- improvement of the geometrical accuracy of the die or mould via machining, which in turn will decrease the manual labour and try-out time
- increase of the machine tool and workshop utilisation via process planning with the help of a CAM-system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

Advantages with HSM

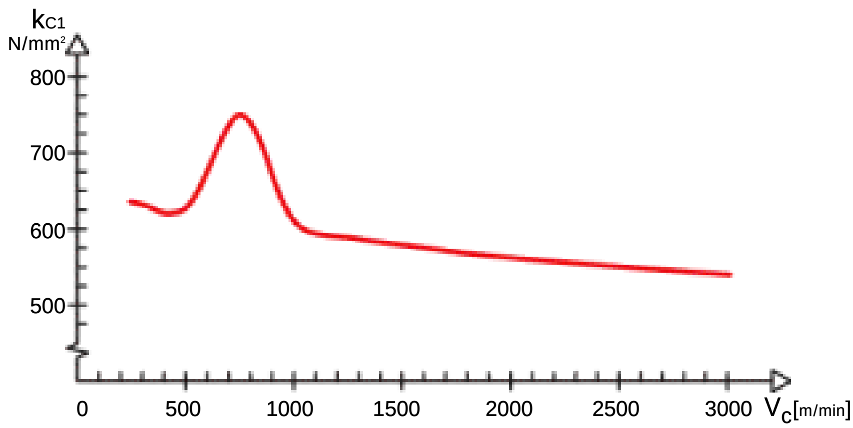
Cutting tool and workpiece temperature are kept low. Which gives a prolonged tool life in many cases. In HSM applications, on the other hand, the cuts are shallow and the engagement time for the cutting edge is extremely short. It can be said that the feed is faster than the time for heat propagation.



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



Cutting force (F_c) vs cutting speed (v_c) for a constant cutting power of 10 kW. $F_c = \frac{P_c}{V_c}$



Cutting speed (v_c) Vs specific cutting force (Mpa) in aluminium 7050.

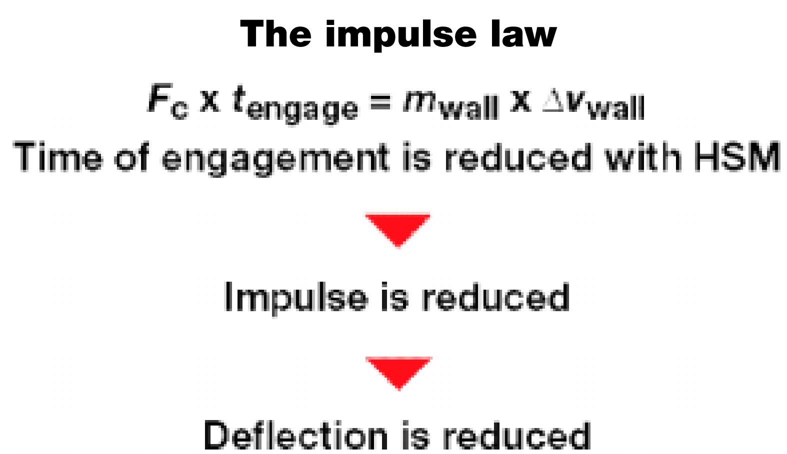
Low cutting force gives a small and consistent tool deflection. This, in combination with a constant stock for each operation and tool, is one of the prerequisites for a highly productive and safe process.

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

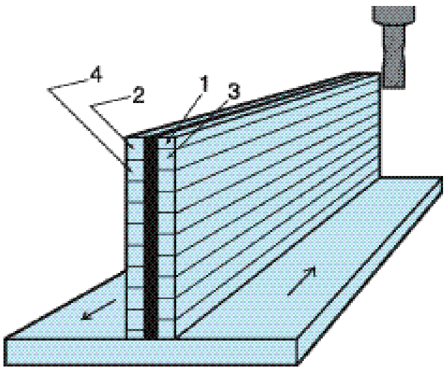
Roughing, semi-finishing and finishing is economical to perform when the total material removal is relatively low.

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



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

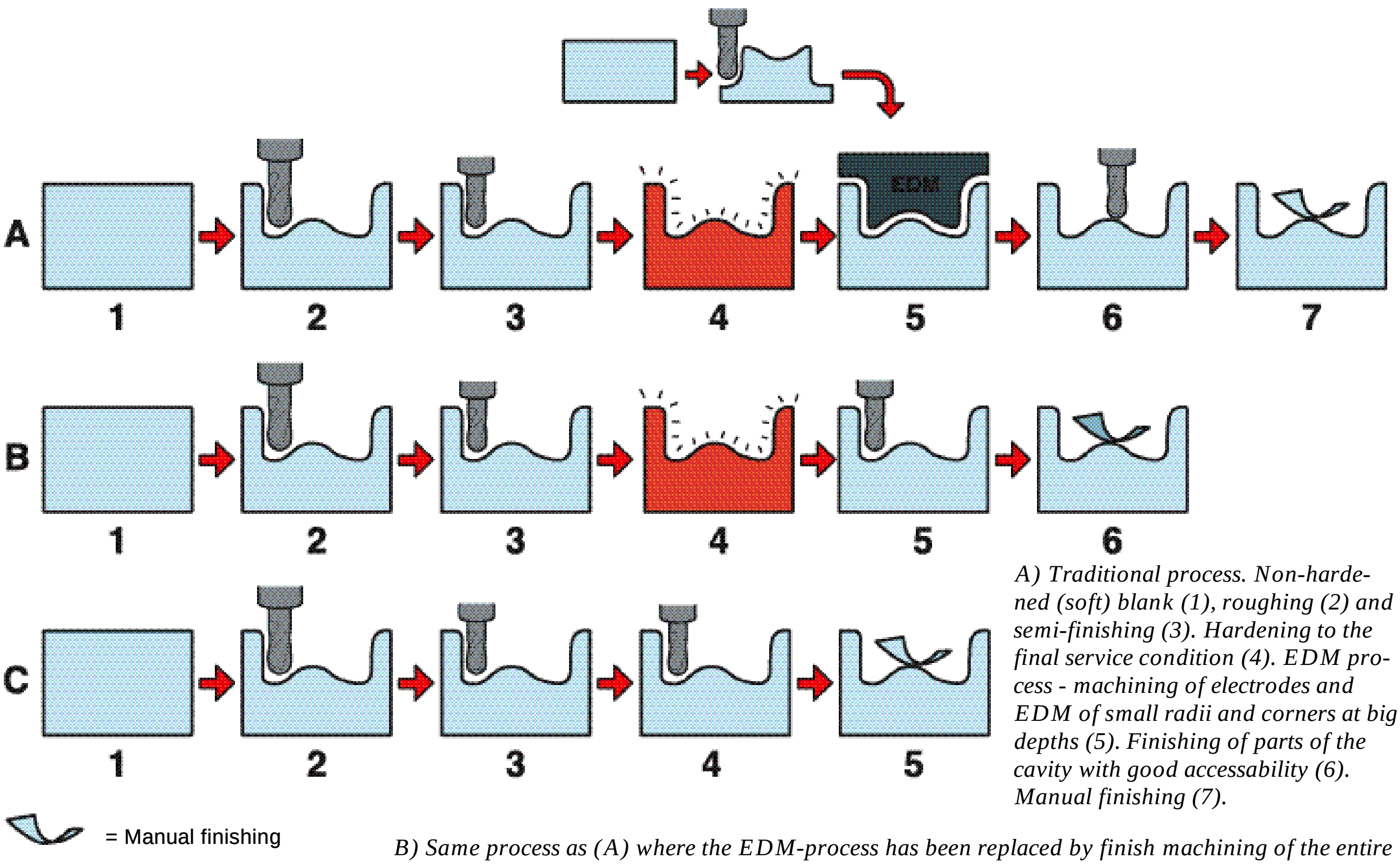
Reduction of production processes as hardening, electrode milling and EDM can be minimised. Which gives lower investment costs and simplifies the logistics. Less floor space is also needed with fewer EDM-equipment. HSM can give a dimensional tolerance of 0,02 mm, while the tolerance with EDM is 0,1-0,2 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...”

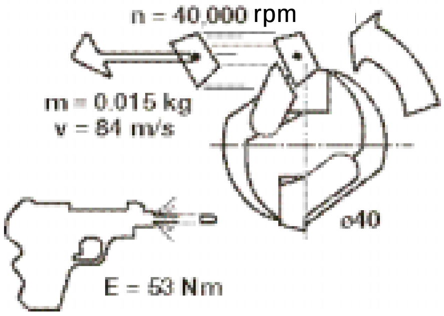


A) Traditional process. Non-hardened (soft) blank (1), roughing (2) and semi-finishing (3). Hardening to the final service condition (4). EDM process - machining of electrodes and EDM of small radii and corners at big depths (5). Finishing of parts of the cavity with good accessability (6). Manual finishing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

• Safety precautions are necessary: Use machines with safety enclosing - bullet proof covers! Avoid long overhangs on tools. Do not use "heavy" tools and adapters. Check tools, adapters and screws regularly for fatigue cracks. Use only tools with posted maximum spindle speed. Do not use solid tools of HSS!

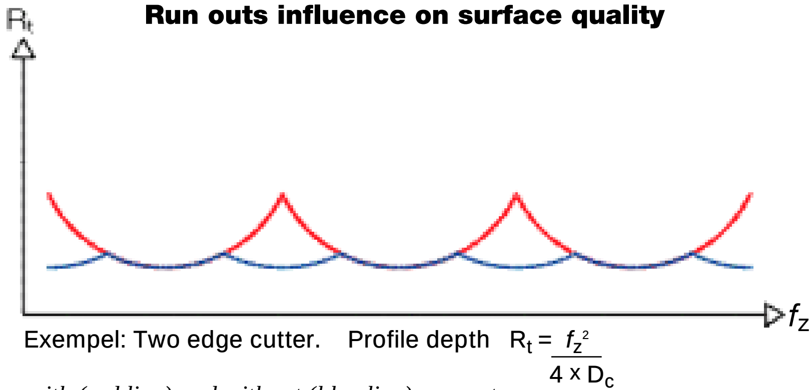


An example of the consequences of breakage at high speed machining is that of an insert breaking loose from a 40 mm diameter endmill at a spindle speed of 40.000 rpm. The ejected insert, with a mass of 0.015 kg, will fly off at a speed of 84 m/s, which is an energy level of 53 nM - equivalent to the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

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

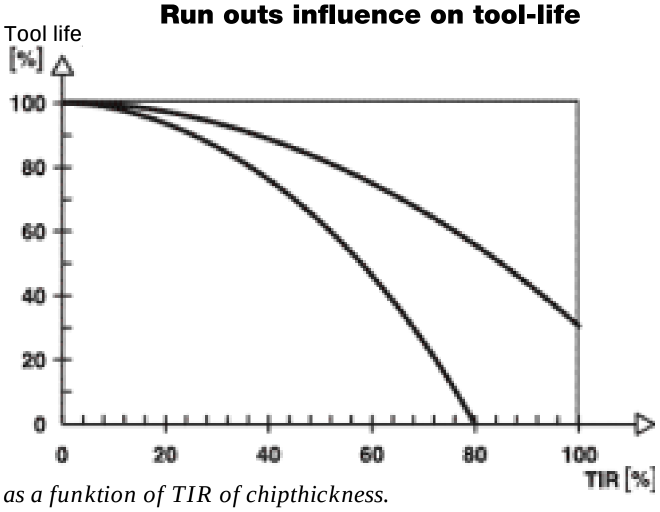
- Spindle speed range
  </ = 40 000 rpm
- Spindle power
  > 22 kW
- Programmable feed rate
  40-60 m/min
- Rapid traverse
  < 90 m/min
- Axis dec./acceleration
  > 1 g (faster w. linear motors)
- Block processing speed
  1-20 ms
- Increments (linear)
  5-20 microns
- Or circular interpolation via NURBS (no linear increments)
- Data flow via RS232
  19,2 Kbit/s (20 ms)
- Data flow via Ethernet
  250 Kbit/s (1 ms)
- 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

Run outs influence on surface quality



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 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 multilayer coatings. Or a somewhat harder, micro-grain carbide with a thin PVD coated TiN layer.

Essential savings can be done via dry machining:

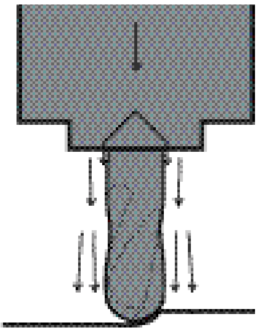
- Increases in productivity as per above.
- Production costs lowered. The cost of coolant and the disposal of it represent 15-20% of the total production costs! This could be compared to that of cutting tools, amounting to 4-6% of the production costs.
- Environmental and health aspects. A cleaner and 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

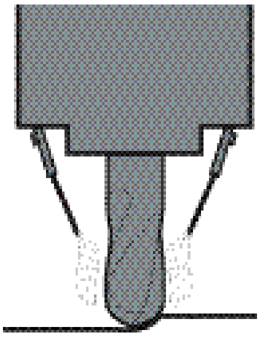
In conventional machining, when there is much time for heat propagation, it can sometimes be necessary to use coolant to prevent excessive heat from being conducted into; the workpiece, cutting and holding tool and eventually into the machine spindle. The effects on the application may be that the tool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

This problem can be solved in different ways. As 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.

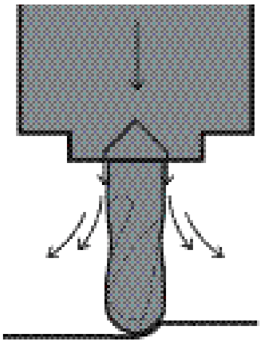
It may sound trivial, but **one of the main factors for success in HSM applications is the total evacuation of chips from the cutting zone**. Avoiding recutting of chips when working in hardened steel is absolutely essential for a predictable tool life of the cutting edges and for a good process security.



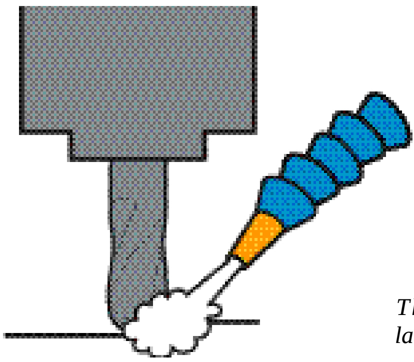
The best way to ensure a perfect chip evacuation is to use compressed air. It should be well directed to the cutting zone. Absolutely best is if the machine tool has an option for air through the spindle.



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.