Chip formation through cutter positioning

Milling is a cyclic process with the cutting edge entering and exiting the workpiece. The thickness of the chip being generated is constantly changing when feeding the cutter in a radial direction (feeding axially/plunging has a constant chip thickness).

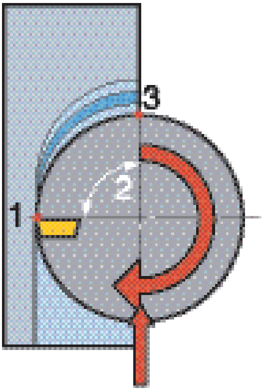
The effect of the cycle on the cutting edge is divided into three clear cutting zones which are determined by the postion and feed direction of the cutter.

1) Entrance into cut

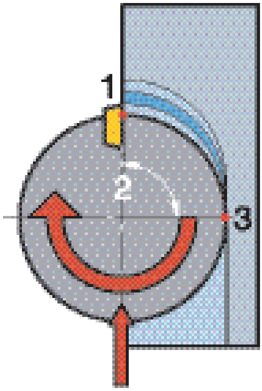
2) Arc of engagement in cut

3) Exit from cut

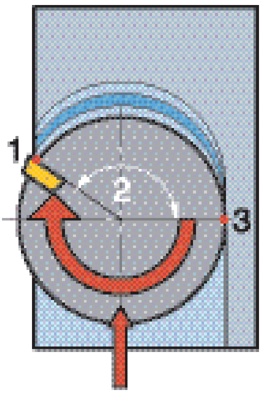
Conventional/up milling



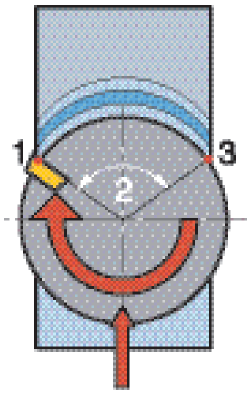
Climb/down milling



Climb milling – zero chip thickness on exit



Milling cutter in component center



1) Entrance into cut

This is the least sensitive of the three cutting zones. Carbide copes well with the compressive stresses on impact of entering with a thicker chip.

– Climb/down milling is always recommended.

– Optimized entering angles to allow high feed rates with softest entry are:

➡ $a_e \geq 70\%\ D_c$ -φ30°
➡ $a_e \leq 25\%\ D_c$ +φ30°

2) Arc of engagement in cut

The maximum arc of engagement possible is 180° when slotting. For finish profile milling the arc can be very small.

The higher the arc of engagement the greater the heat transferred into the cutting edge. Therefore the grade requirements are quite different depending upon the percentage of radial immersion – a_e/D_c.

– High arc of engagement – CVD coated grades provide the best heat barrier.

– Low arc of engagement – here the chip thickness is normally lower and the sharper edge on PVD coated grades generates less heat and cutting pressure.

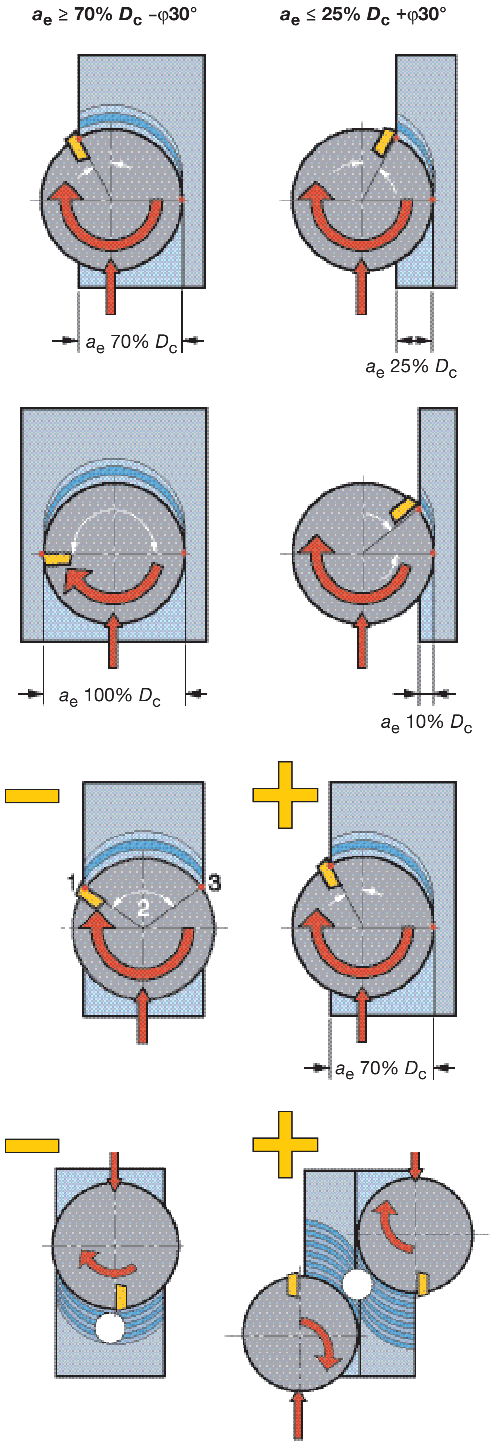
3) Exit from cut

Exiting from the workpiece with a thick chip can cause a drastic reduction in tool life when using carbide. The chip being formed lacks support at the final point of cut. It will try to bend rather than be cut and as it changes direction, the force applied (compressive to tensile) on the carbide geometry fractures the last point of the edge to leave.

– Always climb mill.

– Position cutter off-center, leaving smallest chip thickness on exit.

– Program around interruptions in the workpiece (if this is not possible reduce feed rate).



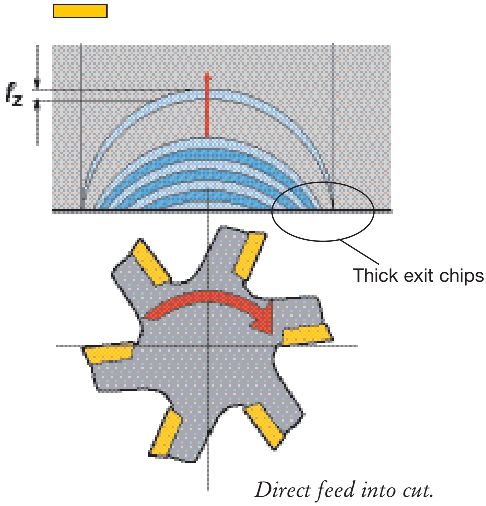
Programming methods

Entering the component

Thick chips on exit from cut will reduce tool life and can cause catastrophic failure.

It can be seen that when the cutter is programmed to enter straight into the workpiece, thick exit chips will be produced until the cutter is fully engaged into the workpiece.

The result of this is that the tool life will be dramatically reduced. To achieve acceptable tool life, the feed for the whole process must be reduced.



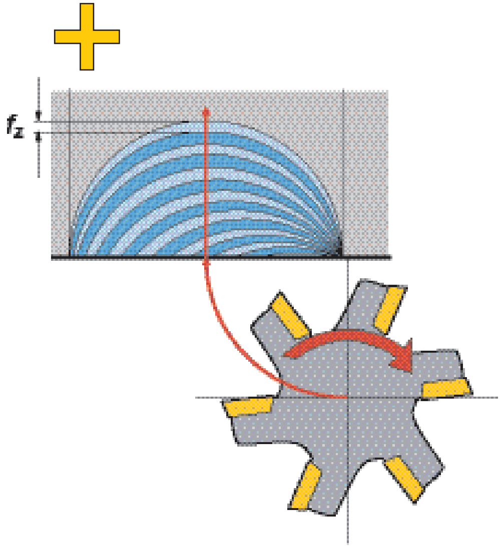
Direct feed into cut.

There are two solutions to remedy this common problem allowing optimized feed rate for when the cutter is fully engaged:

1) Program straight into cut but with the feed reduced to 50% until the cutter is fully engaged.

2) Roll into cut in a clockwise motion (counter clockwise will not solve the thick chip problem).

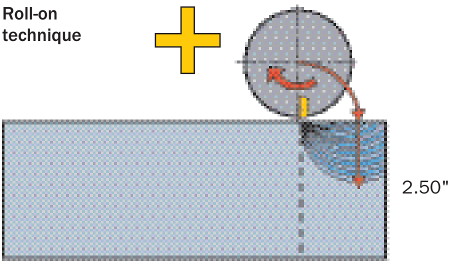
It can be seen that by rolling into cut, the chip thickness on exit is always zero, allowing higher feed and longer tool life.



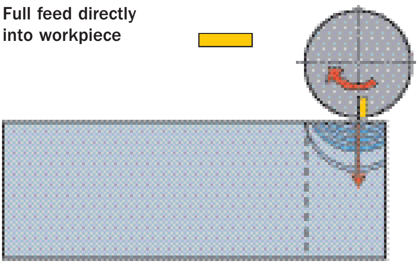
Roll into cut.

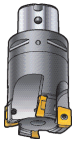
Effect of entry into cut on tool life

Roll-on technique

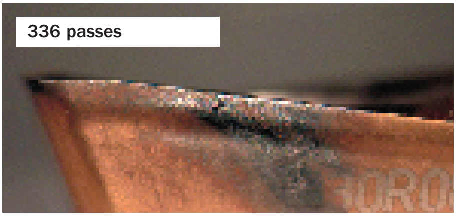


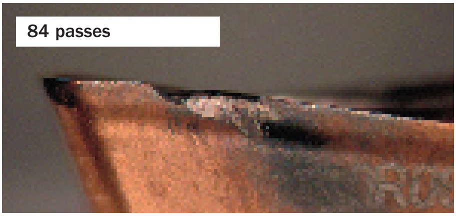
Full feed directly into workpiece

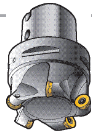




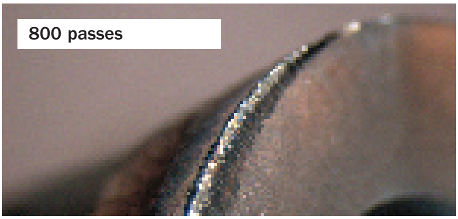
Material: Uddeholm Dievar (CMC 03.11)
Cutter: R390-032C5-11M095
Insert: R390-11T308M-PM 1030
Cutting data: v_c 980 SFM, f_z .008 inch/tooth, a_e .750 inch, a_p .200 inch

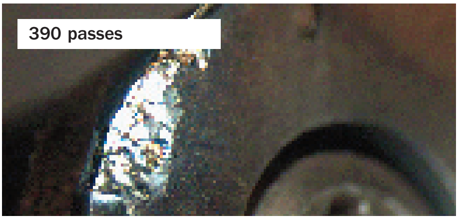


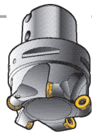




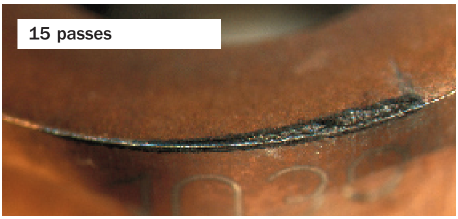
Material: Martensitic stainless steel – X22CrMoV12-1 (CMC 05.11)
Cutter: R300-052C5-12H
Insert: R300-1240E-PL 1030
Cutting data: v_c 980 SFM, f_z .014 inch/tooth, a_e 1.375 inch, a_p .118 inch

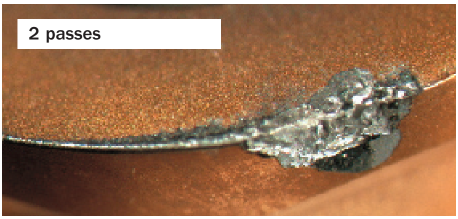






Material: Heat resistant super alloy – Inconel 718 (46 HRC)
Cutter: R300-052C5-12H
Insert: R300-1240E-PL 1030
Cutting data: v_c 100 SFM, f_z .012 inch/tooth, a_e 1.375 inch, a_p .080 inch



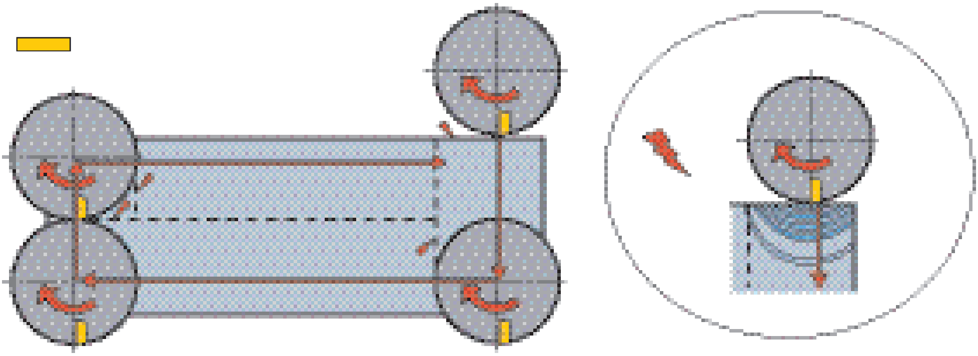


Path in cut

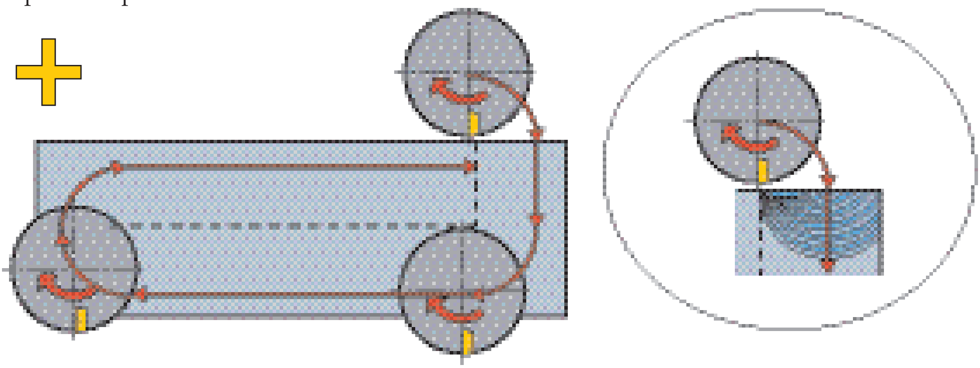
Rolling around external corners

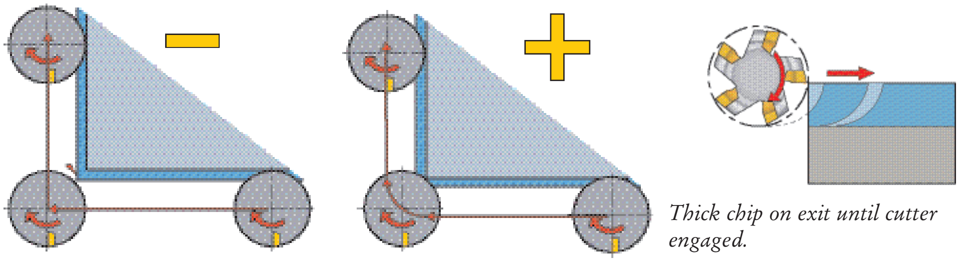
As described previously, entering the component directly will not provide an optimized process.

It can also be seen in the illustrations below that sharp changes of direction in cut will provide the same result as the initial entry – therefore magnifying the problem.



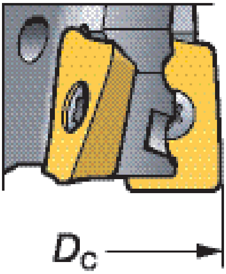
Rolling around all corners should always be applied as a key step to providing a robust, optimized process.



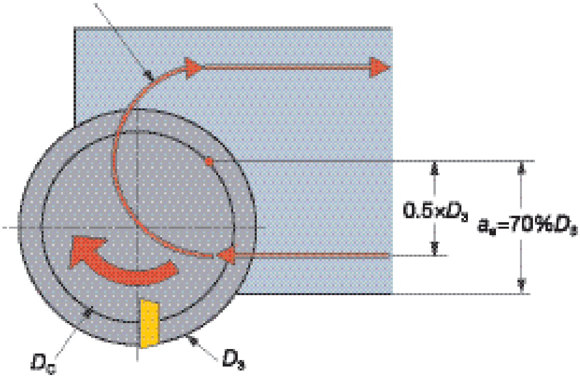


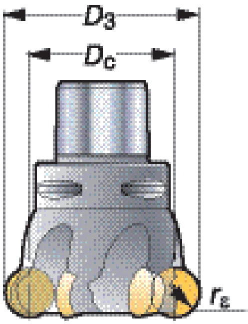
Thick chip on exit until cutter engaged.

To ensure that the corner is covered the maximum (and optimum) width of cut should be a_e 70% of D_c.



Programmed radius = 50% D_c





- Keep cutter constantly engaged.
- Program around interruptions and holes.