Speeds and Feeds

How To Use This Chart:

1) Select your material in the ISO color chart.
2) Select your Insert Cutting Width in the columns.
3) Start with the middle range of the recommended sfm (vc) and feed (inch/rev)
-Adjust the sfm and/or feedrate based on your cutting conditions.
-For Non-ferrous metals, a pecking cycle is recommended for help with breaking the chips

Parting off

ISO	VDI 3323	Material Description		Insert Grade	Cutting Width (inch)		
				HU30	0.079	0.118	0.157
P	P1-5	Non-alloy steel	●	Cutting Speed (vc = sfm)	260 - 590	260 - 590	260 - 590
				Feed (fn = Ipr)	0.001 - 0.006	0.001 - 0.008	0.003 - 0.012
P	P6-9	Low alloy steel	●	Cutting Speed (vc = sfm)	230 - 490	230 - 490	230 - 490
				Feed (fn = Ipr)	0.001 - 0.006	0.001 - 0.008	0.003 - 0.012
M	M12-14	Stainless steel	●	Cutting Speed (vc = sfm)	200 - 460	200 - 460	200 - 460
				Feed (fn = Ipr)	0.001 - 0.004	0.001 - 0.006	0.003 - 0.010
K	K15-20	Grey cast iron	●	Cutting Speed (vc = sfm)	160 - 330	160 - 330	160 - 330
				Feed (fn = Ipr)	0.002 - 0.005	0.004 - 0.010	0.004 - 0.012
N	N21-28	Non-ferrous metals	○	Cutting Speed (vc = sfm)	660-1480	660-1480	660-1480
				Feed (fn = Ipr)	0.002 - 0.005	0.004 - 0.010	0.004 - 0.012

Grooving, Turning

ISO	VDI 3323	Material Description		Insert Grade	Cutting Width (inch)		
				HU30	0.079	0.118	0.157
P	P1-5	Non-alloy steel	●	Cutting Speed (vc = sfm)	260 - 590	260 - 590	260 - 590
				Feed (fn = Ipr)	.002-.004	.002-.005	.002-.006
P	P6-9	Low alloy steel	●	Cutting Speed (vc = sfm)	260-530	260-530	260-530
				Feed (fn = Ipr)	.002-.003	.002-.004	.002-.005
M	M12-14	Stainless steel	●	Cutting Speed (vc = sfm)	200-330	200-330	200-330
				Feed (fn = Ipr)	.002-.004	.002-.005	.002-.005
K	K15-20	Grey cast iron	●	Cutting Speed (vc = sfm)	200-430	200-430	200-430
				Feed (fn = Ipr)	.002-.003	.002-.004	.002-.004
N	N21-28	Non-ferrous metals	○	Cutting Speed (vc = sfm)	490-1310	490-1310	490-1310
				Feed (fn = Ipr)	.002-.006	.003-.006	.003-.006

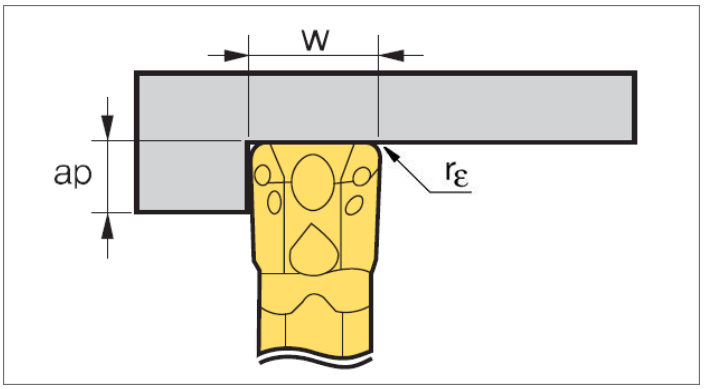
● Optimal ○ Secondary

Technical Details

Selection of Insert:

Feedrate: Fmax = W x .075

Depth of cut: D(min) should be larger than the corner radius of the insert (rε)

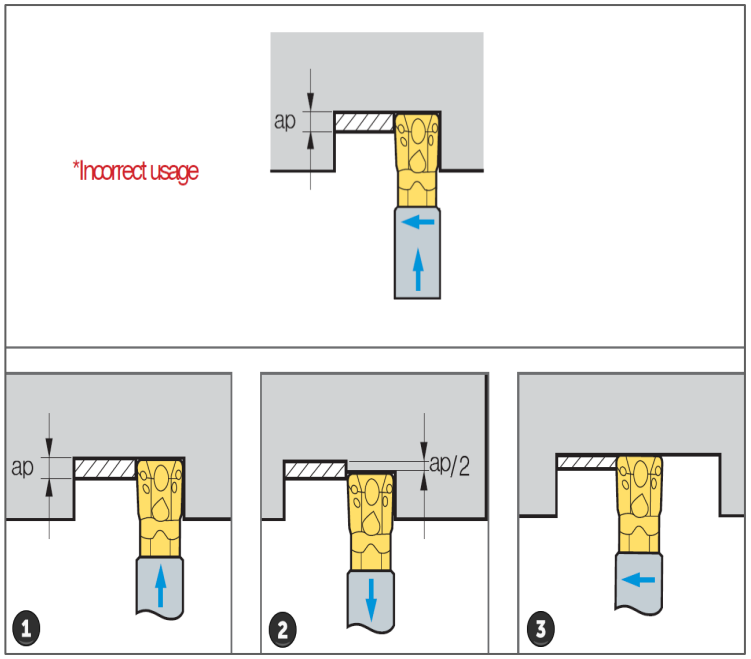


Finishing Guide:

Cutting a groove to a desired diameter may cause the workpiece to deflect, which can affect subsequent turning operations.

Before performing the turning operation, it may be necessary to relieve the deflection, to achieve the desired diameter and surface finish.

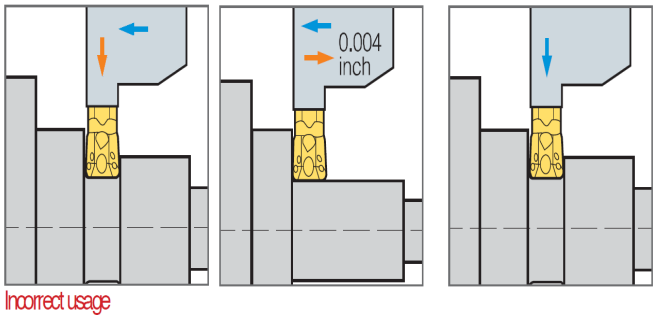
1. Groove to the desired diameter.
2. Pull the tool back a total distance of ap/2 to relieve any deflection.
3. Return to the desired cutting diameter, and continue the external or internal turning operation.



Technical Details

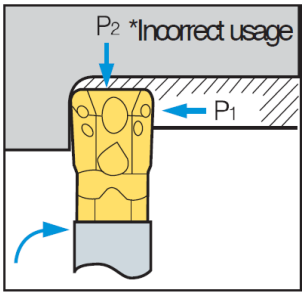
Roughing Guide:

- When using the HG tool in a standard ISO turning application, the cutting forces affect both the side and the tip of the tool. This can cause deflection when the cutting direction changes.
- After a turning operation, it may be necessary to relieve this deflection before cutting a groove, in order to achieve the desired diameter and finish on the workpiece.
- To relieve the deflection, offset the tool .004" from the endpoint, and then return to the original position to perform the grooving application.



Machining a Large Corner Radius or Chamfer:

- HG tools create an unequal cutting load (P1 and P2) when machining a radius that is larger than the corner radius of the insert (rε).
- These unequal cutting forces may break the insert or holder.
- Follow the steps below to eliminate this possibility.



1. Groove to the required depth at the endpoint of the radius or chamfer
2. Return to the start and face down to the start of the radius or chamfer
3. Form the radius or chamfer
4. Continue machining from the start of the groove