Recommended Cutting Speed

ISO	Material		Condition	Strength Kpsi	Hardness HB	Material No.
P	Non-alloy steel and cast steel, free cutting steel	< 0.25 %C	Annealed	61	125	1
		>= 0.25 %C	Annealed	94	190	2
		< 0.55 %C	Quenched and tempered	123	250	3
		>= 0.55 %C	Annealed	109	220	4
			Quenched and tempered	145	300	5
	Low alloy steel and cast steel (less than 5% of alloying elements)		Annealed	87	200	6
			Quenched and tempered	135	275	7
				145	300	8
				174	350	9
	High alloy steel, cast steel, and tool steel		Annealed	99	200	10
			Quenched and tempered	160	325	11

ISO	Material	Condition	Strength Kpsi	Hardness HB	Material No.
M	Stainless steel and cast steel	Ferritic/martensitic	99	200	12
		Martensitic	119	240	13
		Austenitic	87	180	14

ISO	Material	Condition	Strength Kpsi	Hardness HB	Material No.
K	Cast iron nodular (GGG)	Ferritic/pearlitic		180	15
		Pearlitic		260	16
	Grey cast iron (GG)	Ferritic		160	17
		Pearlitic		250	18
	Malleable cast iron	Ferritic		130	19
		Pearlitic		230	20

Groove-Turn, Profiling, Undercutting (SFM)									Internal Grooving, Face Grooving (SFM)				
IB50	IC20N	IC570	IC9025	IC9015	IC508 IC908	IC354	IC9054	IC328	IC9025	IC508 IC908	IC354	IC9054	IC328
	FALSE	520-820	590-820	520-820	490-750	430-590	460-660	330-490	430-620	330-490	330-430	390-460	260-360
	490-690	430-620	460-660	430-620	390-520	330-460	360-520	230-390	300-430	200-330	260-330	330-390	200-300
	390-660	430-660											
	490-720	390-660	430-690	390-660	330-590	330-460	360-520	200-330	300-490	200-360	230-330	260-390	160-300
590-820	330-660	260-560											
	390-590	330-520	330-560	300-520	300-490	300-390	330-430	230-360	230-360	200-360	230-300	260-360	130-230
	360-660	300-590	360-620	300-590	260-520	260-390	260-430	200-330	230-430	230-360	200-260	230-330	130-200
590-820	330-660	260-560											
590-820	330-620	260-560	330-590	260-560	260-520	200-330	200-360	130-260	230-390	200-300	200-260	200-260	100-160
	460-590	390-520	430-490	390-490	230-390	200-260	200-260	130-200	300-360	200-300	200-300	200-260	100-160
590-750	460-660	390-590	430-560	390-520	200-390	160-200	160-200	100-130	300-390	160-260	130-160	130-160	100-130

Groove-Turn, Profiling, Undercutting (SFM)									Internal Grooving, Face Grooving (SFM)				
IC20N	IC570	IC9025	IC320	IC508 IC908	IC08	IC635	IC328	IC9025	IC508 IC908	IC08	IC354	IC9054	IC328
390-820	360-720	330-690	300-690	260-660	200-390	260-520	200-390	230-490	160-430	130-260	160-430	200-460	130-260
330-720	300-660	260-620	260-620	200-560	130-390	200-490	130-390	200-490	130-430	70-230	130-430	160-460	100-260

Groove-Turn, Undercutting (SFM)				Internal Grooving, Face Grooving (SFM)			
IC428	IC20	IC9025	IC418	IC428	IC9025	IC418	IC20
390-660	200-390	300-590	330-620	260-430	200-360	160-300	70-130
330-590	160-260	260-490	300-560	200-330	160-300	130-260	70-130
490-890	230-330	360-820	430-850	300-460	230-390	200-330	130-200
390-560	160-300	300-460	330-490	260-390	200-330	160-300	130-200
490-820	230-330	390-750	390-750	300-430	230-360	200-330	130-200
390-660	200-300	300-590	300-590	260-360	200-300	160-260	100-160

Recommended Cutting Speed

ISO	Material		Condition	Tensile Strength Kpsi	Hardness HB	Material No.
N	Aluminum-wrought alloy		Not cureable		60	21
			Cured		100	22
	Aluminum-cast, alloyed	<=12% Si	Not cureable		75	23
			Cured		90	24
		>12% Si	High temperature		130	25
	Copper alloys	>1% Pb	Free cutting		110	26
			Brass		90	27
			Electrolitic copper		100	28
	Non-metallic		Duroplastics, fiber plastics			29
			Hard rubber			30

ISO	Material		Condition	Tensile Strength Kpsi	Hardness HB	Material No.
S	High temp. alloys	Fe based	Annealed		200	31
			Cured		280	32
	Super alloys	Ni or Co based	Annealed		250	33
			Cured		350	34
			Cast		320	35
	Titanium and Ti alloys			RM 58		36
		Alpha+beta alloys cured		RM 152		37

ISO	Material	Condition	Tensile Strength Kpsi	Hardness HB	Material No.
H	Hardened steel	Hardened		55 HRc	38
		Hardened		60 HRc	39
	Chilled cast iron	Cast		400	40
	Cast iron	Hardened		55 HRc	41

Groove-Turn, Profiling, Undercutting (SFM)		Internal Grooving, Face Grooving (SFM)
ID5	IC20	IC20
490-8200	980-2620	330-980
490-8200	750-1020	330-980
490-8200	920-2720	330-980
490-8200	660-1670	330-980
1080-2620	430-980	260-660
1080-2620	390-660	260-490
260-1310	300-490	200-330

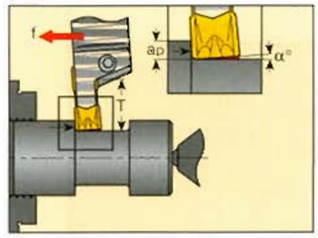
Groove-Turn, Profiling, Undercutting (SFM)						Internal Grooving, Face Grooving (SFM)				
IC320	IC508 IC908	IC08	IC20	IC635	IC328	IC9025	IC508 IC908	IC08	IC635	IC328
130-200	100-160	100-160	100-130	100-160	70-100	70-130	70-130	70-100	70-130	70-100
100-130	70-130	70-130	70-130	70-160	50-70	70-100	50-100	50-70	50-100	50-70
70-100	70-100	70-100	70-100	50-70	50-70	50-70	50-70	50-70	50-70	50-70
50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70
50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70	50-70
490-620	430-560	330-430	330-430		260-330	330-460	300-390	260-330		200-260
160-260	130-230	70-160	70-160		50-100	130-200	70-160	70-130	50-100	50-100

Groove-Turn, Profiling, Undercutting (SFM)					Internal Grooving, Face Grooving (SFM)			
IC428	IC9025	IC418	IB50	IC20	IC428	IC9025	IC418	IC20
100-160	70-100	70-100	300-360	70-130	50-80	50-80	50-80	50-70
100-160	70-100	70-100	260-330	70-100	50-80	50-80	50-80	50-70
100-160	70-130	70-130	590-660	70-160	50-80	50-80	50-80	50-80
100-160	70-100	70-100	300-360	70-130	50-80	50-80	50-80	50-80

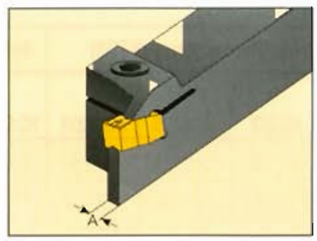
B

Principles of Turning with Groove-Turn Tools

The basic principle in turning with groove-turn tools is the deflection of the cutting tool, which results in a frontal clearance angle α° between the insert and the workpiece. The clearance angle α° is a function of the side cutting forces and is not constant, as is the case with ISO inserts.
The deflection is influenced by the following factors:

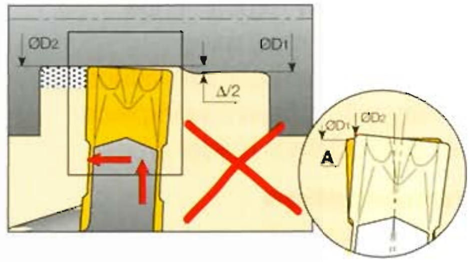


Feed **f**
Depth of Cut **ap**
Overhang of Insert Support **T**
Width of Insert Support **A**
Cutting Speed **Vc**
Workpiece Material

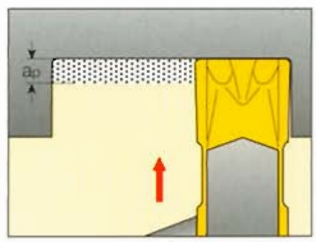


When the above factors remain constant during turning, a high degree of accuracy with a tolerance up to ±.0004 can be achieved.

Finishing Operation: Diameter Compensation

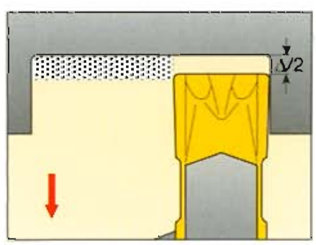


A compensation factor for the final diameter must be used in the final machining operation. After the initial grooving to the required final diameter, the machining direction is normally changed for longitudinal turning. At this point the deflection occurs, and if machining continues without correction, corner **A** will penetrate the material. This will result in two different diameters: D_1 from the grooving and D_2 from the turning. The difference between D_1 and D_2 is a value we define as Δ.
The compensation factor is Δ/2, as shown below.



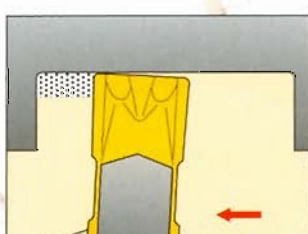
$$\frac{\Delta}{2} = \frac{øD_1 - øD_2}{2}$$

Using the compensation factor will eliminate the small surface step. Follow this simple procedure during machining:



- Groove to the final diameter.
- Pull back the tool, a distance equal to the value of Δ/2.
- Continue the finish turning operation.

Characteristic values of Δ are shown in the diagrams on the next page.



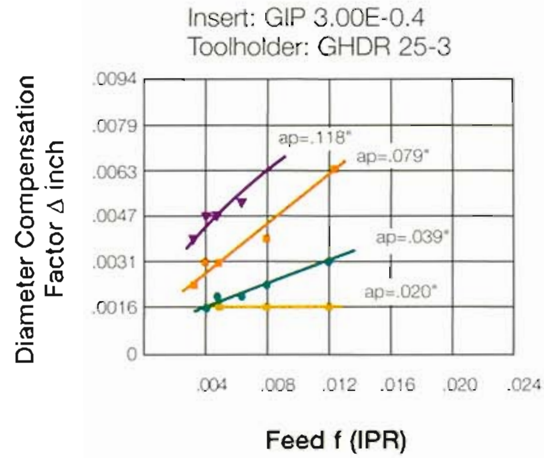
B

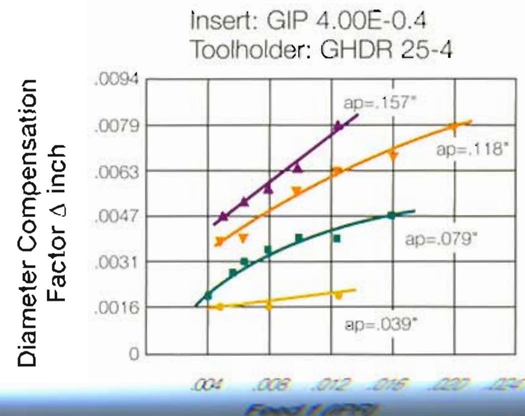
Characteristic Values of Δ

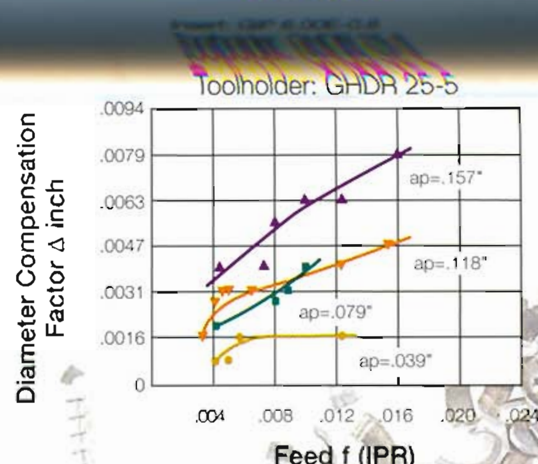
The diagrams show experimental results for specific machining conditions. These are sample values that will vary with different workpiece materials and different holder types.

Recommendation

Measure the Δ value for your finishing operation in a short test using your selected finishing conditions. Do not run your test using the final diameter.

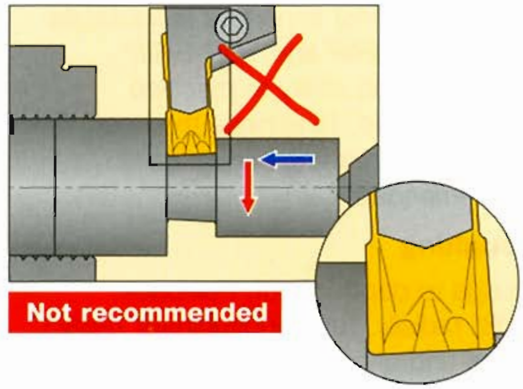




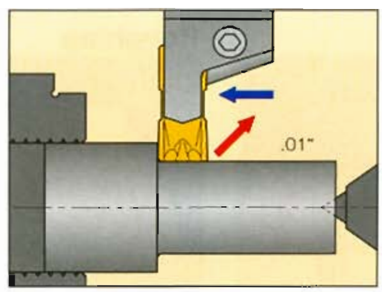


Multifunction Operations

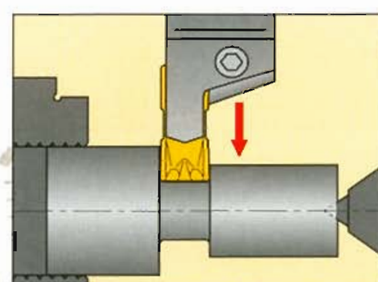
The groove-turn tools are multifunction tools, able to operate in a sequence of grooving and turning modes. Moving from turning to grooving requires consideration of the basic GRIP principle, thereby eliminating the possibility of insert breakage. In this situation one must release the side deflection which is necessary in turning, but not recommended in grooving.



The following rough machining sequence is suggested:
After completing the longitudinal turning, but before starting the grooving, the side deflection must be released. Retract the tool by .01"x45° and return to the original position without side load.



Then, after the deflection has been released and the holder is perpendicular to the workpiece, the grooving operation may start.



Machining Between Walls

One of the most important advantages of the CUT-GRIP and TOP-GRIP systems is the ability to machine between walls. To achieve the best results, the following sequence is recommended:

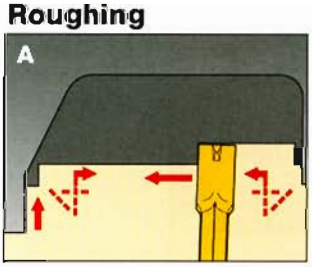
Roughing

Plunge to depth of cut. Pull back .008" radially. Turn longitudinally, retract at the end of the cut by .008" simultaneously in radial and axial directions. Plunge again and repeat same cycle leaving .008" for the finishing cut.

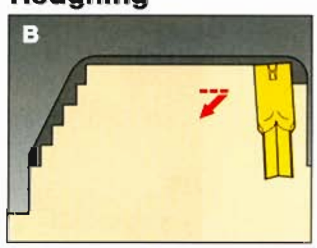
Finishing

Plunge on the right side reaching the tangent of bottom radius. Retract and relieve the tangent point of the other side radius. Retract and machine all the contour, pulling back by compensation value along the bottom (see pages B157-158).

Roughing

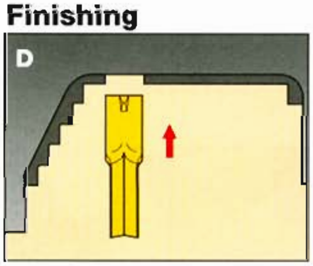


Roughing

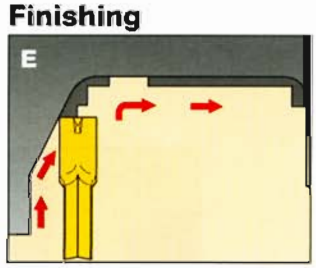


Finishing

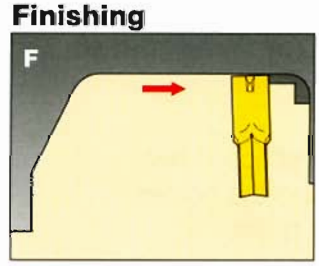
Finishing



Finishing



Finishing

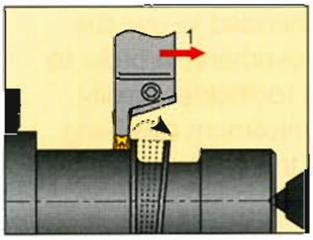


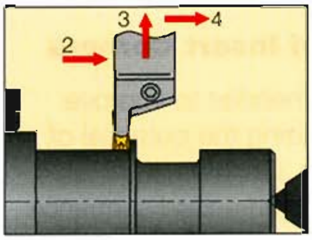
Eliminating a “Hanging Ring”

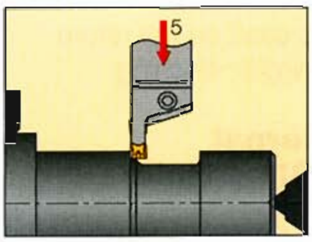
When turning at the end of a bar or toward a recess between two walls, a "hanging ring" may be formed.

How to eliminate the unwanted "hanging ring":

- Turn toward the recess. Stop a short distance before reaching the recess.
- Pull back the groove-turn tool and re-position it.
- Machine as shown. This final operation achieves the size and flatness of the side wall.



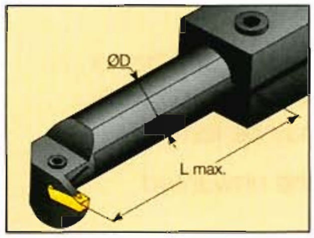




Optimizing Internal Machining

Toolholder Overhang

It is always recommended to use the minimum possible overhang in order to maintain maximum toolholder rigidity. As a general rule, maximum overhang should not exceed three times the holder-bar diameter.

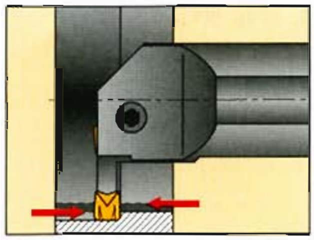


Lmax. ≤ 3D

Efficient Use of Insert Corners

It is always recommended to improve efficiency by optimizing the potential of groove-turn inserts. On internal machining, the recommended sequence utilizes both corners of the insert.

- The first pass uses one corner for roughing.
- The other corner is used on the return path for semi-finishing or finishing.



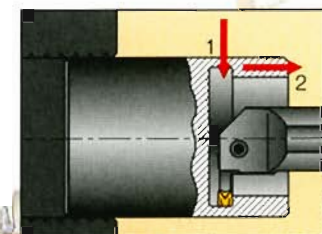
Improving Internal Turning in a Blind Hole

Internal turning in a blind hole brings about the problem of chip exit. When the tool reaches the rear side wall, chips may be caught between the wall and the insert. This may cause insert breakage.

Two solutions follow that can eliminate this problem:

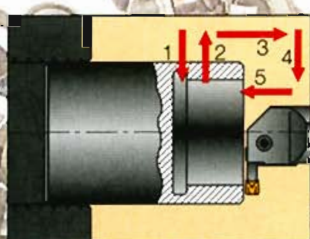
First Solution

- Start by grooving at the rear wall.
- Continue by turning from the inside toward the outside.



Second Solution

- Start by grooving at the rear wall.
- Pull the tool back to the outside.
- Turn the final diameter from outside toward the groove.



Machining Conditions

Calculation of Required Machine Power

Turning

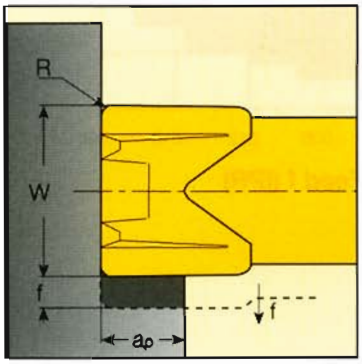
$$P = \frac{K_C \cdot ap \cdot f \cdot v_C}{\eta \cdot 33670} \text{ [HP]}$$

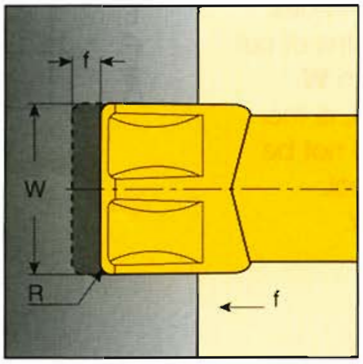
Grooving/Parting

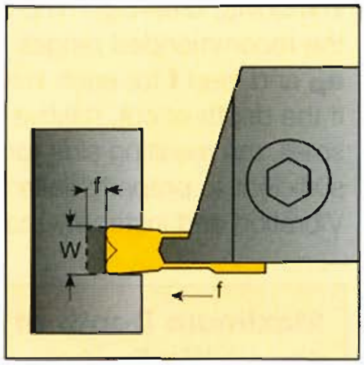
$$P = \frac{K_C \cdot W \cdot f \cdot v_C}{\eta \cdot 33670} \text{ [HP]}$$

Face Grooving

$$P = \frac{K_C \cdot W \cdot f \cdot v_C}{\eta \cdot 33670} \text{ [HP]}$$







Where:
Kc- Specific Cutting Forces (PSI),
turning values could be used.
η - Efficiency ($\eta \approx 0.8$)
ap - D.O.C. (inch)
f - Feed (IPR)
Vc - Cutting speed (SFM)

Kc Values

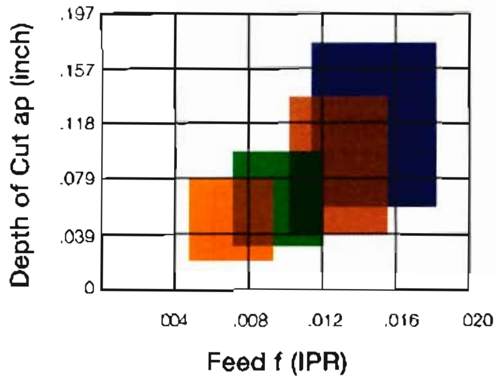
Mtl. Gr. No.	Kc [Kpsi]	Mtl. Gr. No.	Kc [Kpsi]
1	290	21	73
2	305	22	116
3	312	23	116
4	319	26	102
5	319	27	102
6	305	28	247
7	305	31	435
8	305	32	450
9	305	33	479
10	363	34	479
11	471	35	464
12	334	36	247
13	406	37	247
14	377	38	667
15	160	39	682
16	189	40	667
17	160	41	653
18	261		
19	131		
20	145		

For material groups, see page G2.

B

How to Choose Depth of Cut and Feed

The depth of cut and the feed directly influence the performance and tool life of an insert. If the depth of cut is too large for the width of an insert, or the feed too high, the insert may be overloaded, causing immediate breakage. The chart provides the recommended ranges for depths of cut **ap** and feed **f** for each insert width **W**.
If the depth of cut, relative to feed, is too small, the resulting side forces will not be sufficient to properly deflect the tool. Vibration and instability may occur.



Maximum Depth of Cut
apmax=**W x.8**

Maximum Feed
fmax=**W x.075**

Minimum Depth of Cut

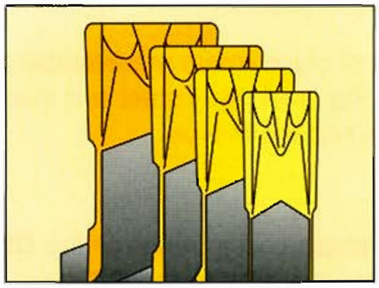
The depth of cut has a significant influence on side deflection. Using a small depth of cut with a wide insert may result in a deflection that is too small to be effective. This may result in vibration. In the finishing operation, when the depth of cut is normally minimal, it is important to select the proper insert with a small width and a small corner radius.

The chart shows the recommended minimum depth of cut and minimum feed, as functions of insert width and corner radius.

W (inch)	**R** (inch)	a_p min (inch)	**f** min (IPR)
.118	.0157	.0197	.0078
.157	.0157	.0197	.0078
.157	.0315	.0315	.0078
.197	.0157	.0197	.0078
.197	.0315	.0315	.0078
.236	.0315	.0315	.0078
.315	.0315	.0315	.0086
.393	.0315	.0315	.0098

Insert Width

- The width of an insert contributes to its strength and, therefore, should be as large as possible relative to the dimensions of the workpiece.
- The width of an insert determines the permitted overhang of the tool.
- The larger the insert, the wider the upper and lower jaws can be; therefore, higher forces are required to effect the necessary side deflection.
- If the depth of cut is small, the width of the insert should be proportionately smaller in order to guarantee the required deflection.

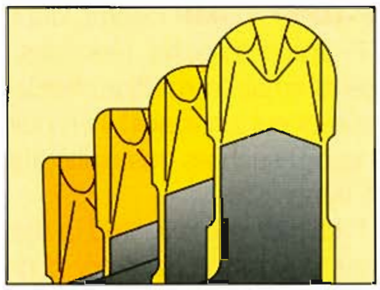


Choosing Insert Geometry

Insert Radius

Choosing the insert radius for a particular application is a combination of many factors. The corner radius of the groove-turn insert influences the product shape and tool life.

- A larger radius - in turning operations - normally improves surface quality.
- An insert with a larger radius gives better distribution of the cutting load and of the generated heat. It is stronger and ensures longer tool life.
- Small radii on CUT-GRIP inserts result in increased side forces and side deflection, preventing instability, especially with small depths of cut and feeds.
- The best radius to use is basically determined by the geometry and dimensions of the workpiece. The more securely the workpiece is fastened in the machine tool, the larger the radius may be.
- When the ratio of workpiece length compared to diameter is large, inserts with smaller radii will prevent chatter.
- The radius should be larger than the maximum feed.
- In profiling operations, inserts with large corner radius or full radius are required.



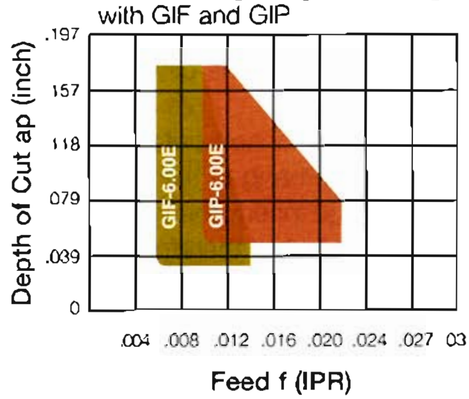
Selection of Suitable Chipformer

Two types of chipformers are available for turning and grooving steel, alloy steel and stainless steel: the F-type and the P-type.

Turning

The basic chipbreaking range of the **GIP** chipformer compared to the **GIF** type is shown in the diagram. An insert with the P-type chipformer can handle .020", whereas an F-type reaches a maximum of .014 IPR. under the same machining conditions.

Chipbreaking Range in Turning with GIF and GIP



F Chipformer

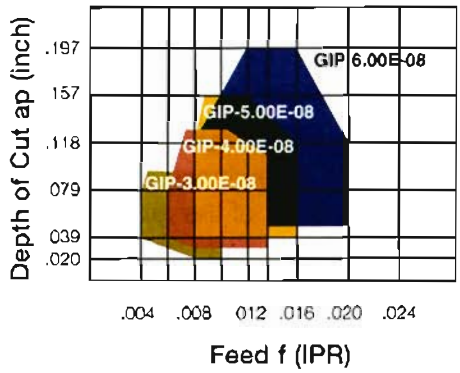
The F-type chipformer is available on:

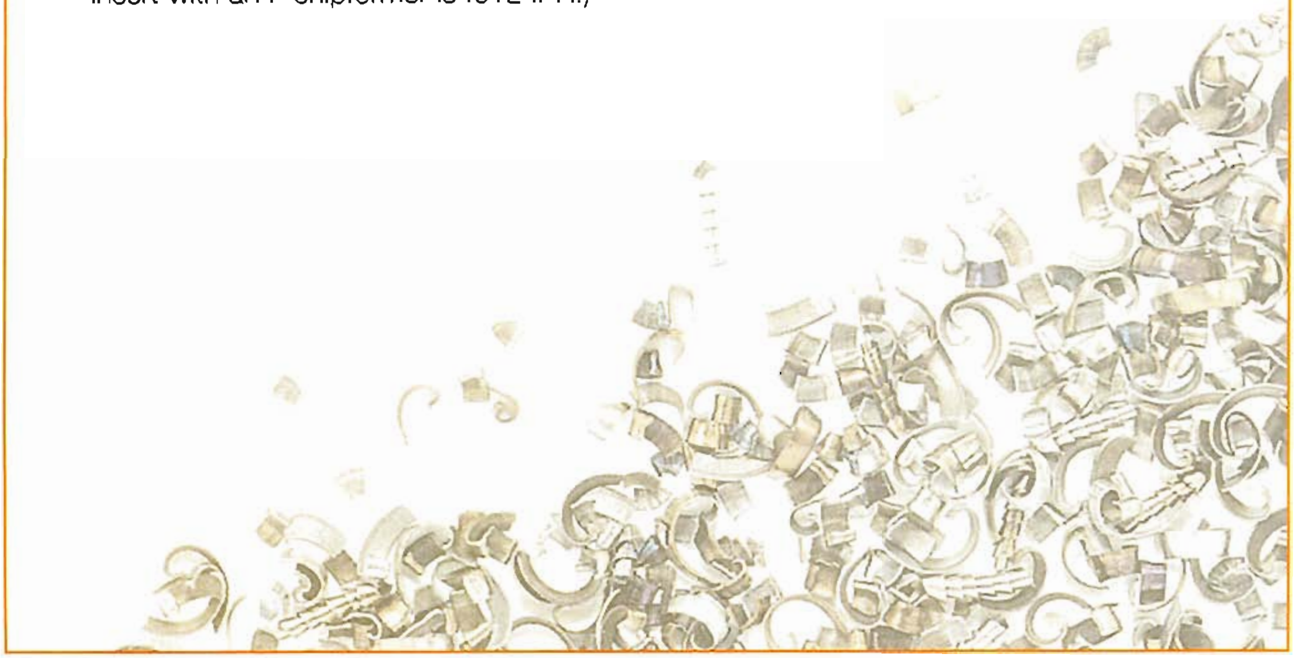
- **CUT-GRIP GIF** inserts, double-ended, ground.
- **CUT-GRIP GIMF** inserts, single-ended, utility.
- **TOP-GRIP TGMF** inserts, double-ended, utility.

The F-type chipformer functions properly in a range of small-to-medium feeds with small depths of cut on materials requiring high chip formation, such as austenitic stainless steel and high-temp alloys.

The F chipformer is limited in the upper feed ranges due to the deflector geometry. (For example, the maximum feed for a .236" wide insert with an F chipformer is .012 IPR.)

Chipbreaking Range in Turning as a Function of Insert Width

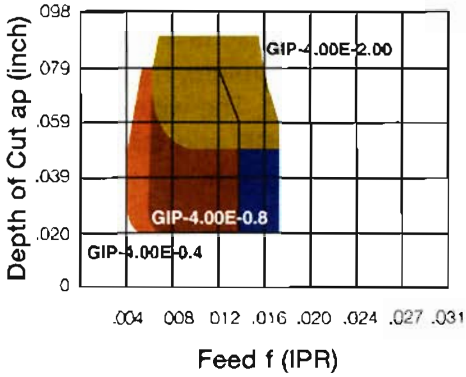




P Chipformer

The P-type chipformer is available on **CUT-GRIP GIP** double-ended, ground inserts. **GIP** inserts are very efficient in turning at medium-to-high feeds, especially over .010 IPR with relatively low power consumption. The P-type chipformer is comparable to the standard ISO M-type chipformer with an open chipformer design. In general the chipbreaking range depends upon insert width and corner radius as shown in the diagrams. When selecting the width of an insert, consider the chipforming range. Small widths of inserts are designed to cover small feeds and small depths of cut.

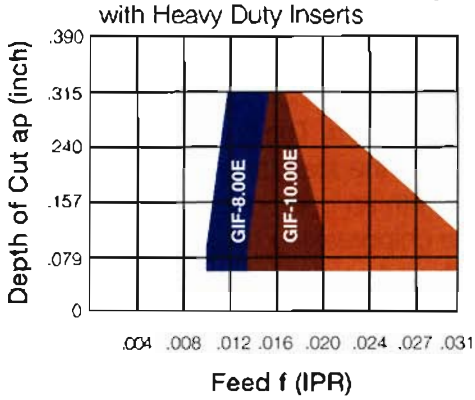
Chipbreaking Range in Turning as a Function of Corner Radius



Chipbreaking with Heavy Duty GIF-8 and GIF-10 Inserts

The design of the inserts allows for high side forces and therefore, high feeds and large depths of cut. The wide application range of **GIF-8.00E** and **GIF-10.00E** is shown in the diagram.

Chipbreaking Range in Turning with Heavy Duty Inserts

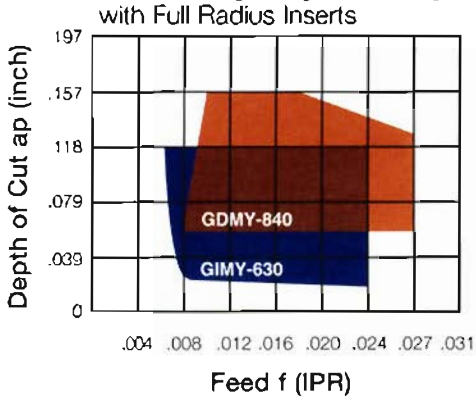


Chipformers on Full Radius Inserts for Turning and Profiling

GIMY and GDMY - the DAISY Family

The new **Daisies**, single-ended **GIMY** and the double-ended **GDMY**, are full radius utility inserts. Each has a clearance capable of cutting an arc up to 250°. For example, the diagram shows the chipbreaking ranges for types **GIMY-630** and **GDMY 840** inserts.

Chipbreaking Range in Turning with Full Radius Inserts



GIPA and GIDA Inserts for Aluminum

GIPA and **GIDA** inserts are recommended for aluminum and high temp. alloys. Both are ground inserts with sharp cutting edges and have a 15° positive rake angle around the cutting edge. Both are suitable for external and internal operations.

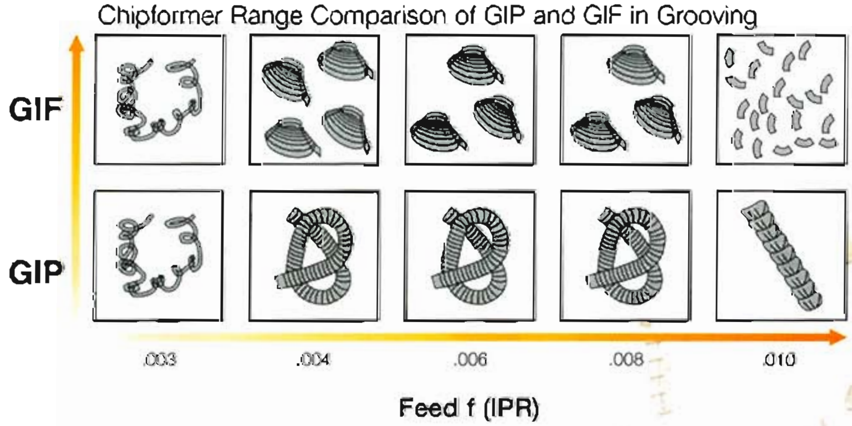
The **GIPA** insert has a clearance angle of 7° without a chip deflector and is capable of cutting an arc up to 250°.

The **GIDA** insert has a clearance angle of 10° and is capable of cutting an arc up to 270°. Its central bump deflector guarantees chipbreaking on aluminum in a wider range of feeds compared to the **GIPA.**

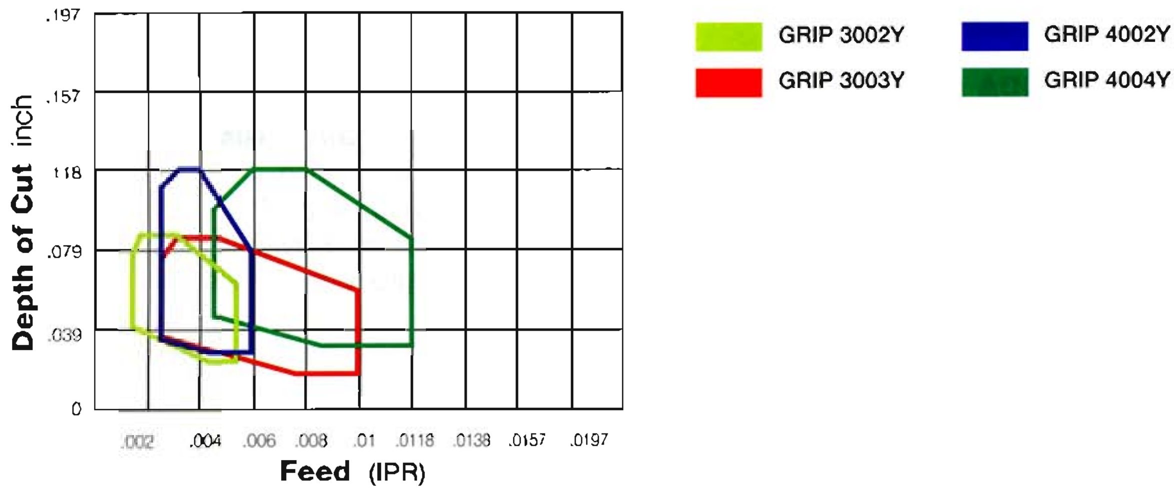
Grooving and Parting

The F-type chipformer (**GIF** or **GIMF**) is recommended for all grooving applications on a wide range of workpiece materials. The P-type chipformer (GIP) in grooving is limited in performance.

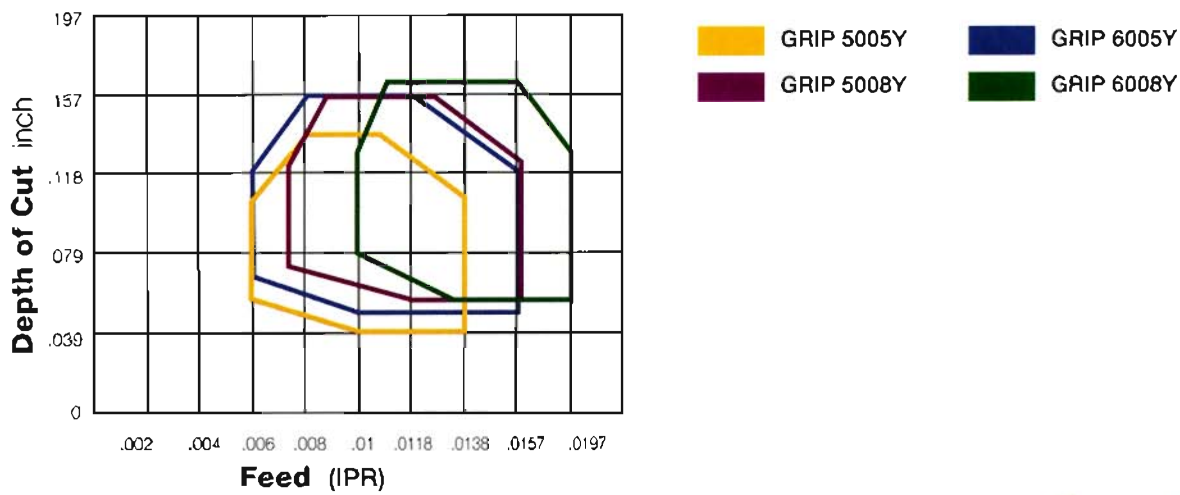
Chipformer Range Comparison of GIP and GIF in Grooving



Turning Application Range



Turning Application Range



Turning Application Range