Technical Details

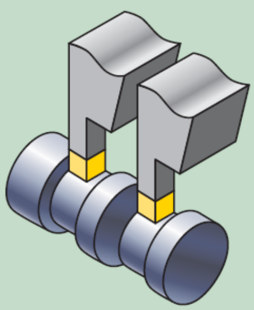
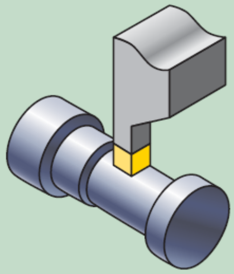
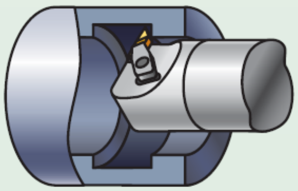
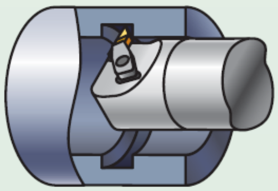
Perfect for shallow grooving operations, the Haas Notch clamping system provides a complete line of grooving geometries and a general purpose grade selection to meet even the most demanding application requirements.

For increased rigidity, versatility, chip control, and carbide grade options, the Haas Notch clamping system is the proven solution.

With maximum clamping rigidity and superior versatility, Haas Notch inserts employ a top rake chip control geometry that efficiently evacuates chips and produces better quality parts, faster than ever before.

Utilize this comprehensive, easy-to-use guide for the information necessary to identify, choose, and select the appropriate cutting tools for your specific needs.

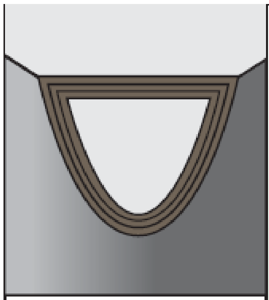
What you need to know:
- Material being machined.
- Groove depth, width, and profile or Thread Size
- Application to be performed (O.D. or I.D. Grooving or Threading).
- Tool holder requirements (square shank/Boring bar, right/left).

		application
		O.D. Grooving and Plunge and Turn or O.D. Threading
		I.D. Grooving or I.D. Threading

Insert Grade:
HU25

	Recommended conditions for HU25 grade					
cutting condition	**steel**	**stainless steel**	**cast iron**	**non-ferrous metals**	**high-temp alloys**	**hardened materials**
smooth cut, pre-turned surface	●	●	○	○	●	○
varying depth of cut, casting, or forging skin	●	●	○	○	●	○
lightly interrupted cut	●	●	○	○	●	
heavily interrupted cut	●	○		○		

● **Optimal** ○ **Secondary**

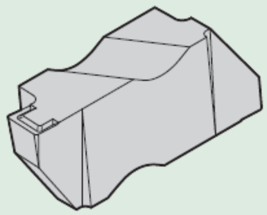
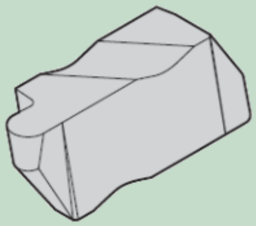
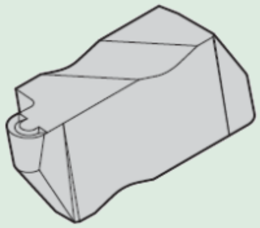


HU25

An advanced PVD TiAlN-coated grade with a tough, ultra-fine-grain unalloyed substrate. For general-purpose machining of most steels, stainless steels, high-temp alloys, titanium, irons, and non-ferrous materials. Speeds may vary from low to medium and will handle interruptions and high feed rates.

P
M
K
N
S
H

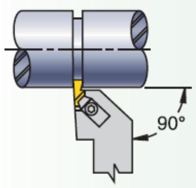
Insert ChipBreaker

insert style	application	rake angle
HNG-K 	• Chip control geometry. • General-purpose grooving. • O-ring grooving. • Circlip grooving. • Light turning.	10° positive
HNR 	• Full radius grooving. • Turning and profiling.	neutral
HNR-K 	• Chip control geometry. • Full radius grooving, turning, and profiling.	10° positive

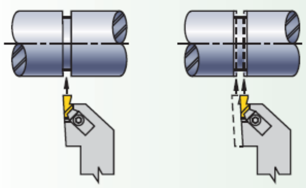
Grooving Tool Failure and Solution Guide

Practical Solutions to Common Grooving Problems

Holder Position for Grooving Operation

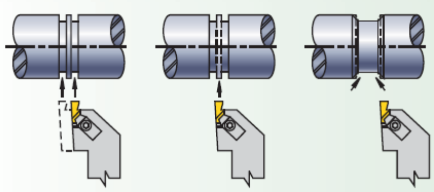


How to Cut a Groove Slightly Wider than the Groove Tool



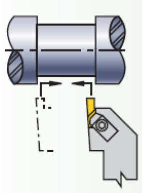
1. Plunge the center of the groove.
2. Plunge each side of the groove to get the specified width. Use a slower feed rate when cutting groove sides.

How to Cut Wider Grooves



1. Plunge out both sides of groove width.
2. Plunge center area to remove web of material remaining.
3. Plunge both sides of groove at the required angle, using approximately one-half the width of the grooving tool for maximum width of cut.

Finish Turning the Groove



1. Follow recommendations explained above.
2. To avoid insert chipping and to achieve groove wall perpendicularity, follow the tool path outlined here.
3. Use the lightest depth of cut that still enables good chip surface finishing.

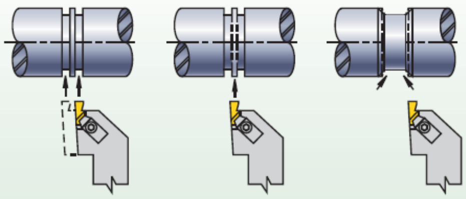
problem	solution
bur	1. Ensure tool center height. 2. Use sharp tool (index more often). 3. Use positive rake PVD-coated insert. 4. Use correct grade for workpiece material. 5. Use correct geometry (e.g., positive rake for work-hardening material). 6. Chamfer before grooving. 7. Change tool path.
poor surface finish	1. Increase speed. 2. Use sharp tool (index more often). 3. Dwell tool in bottom 1–3 revolutions (max). 4. Use proper chip control geometry. 5. Increase coolant flow/concentration. 6. Ensure proper setup (overhang, shank size). 7. Use correct geometry (e.g., positive rake for work-hardening material).
groove bottom that is not flat	1. Use sharp tool (index more often). 2. Dwell tool in bottom 1–3 revolutions (max). 3. Reduce tool overhang (increase rigidity). 4. Ensure correct tool alignment. 5. Reduce feed rate at groove bottom. 6. Use a wider insert. 7. Ensure tool center height.
poor chip control	1. Use “K” chip control geometry insert. 2. Use sharp tool (index more often). 3. Increase coolant concentration. 4. Adjust feed rate (usually increase first).
chatter	1. Reduce tool and workpiece overhang. 2. Adjust speed and feed (usually increase first). 3. Ensure center height.
insert chipping	1. Use correct grade for workpiece material. 2. Increase speed. 3. Reduce feed. 4. Use a stronger grade. 5. Increase tool and setup rigidity.
side walls not straight	1. Check tool alignment for square. 2. Use correct insert hand. 3. Reduce workpiece and tool overhang. 4. Use sharp insert (index more often).

Technical Details

Machining Guidelines for Chip Control • Grooving

When the proper cutter diameter is not available, proper cutter positioning will provide positive results.

- Center height of insert should be positioned at the center of the workpiece or up to .005" (0,13mm) above.
- Dwell time in the bottom of the groove (more than three revolutions) is not recommended.
- Chip control is feed-rate related and should be adjusted to fit the particular situation. Recommended feed range is .003–.012 IPR (0,08–0,3 mm/rev).

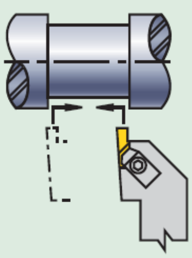


Machining Guidelines for Chip Control • Turning/Profiling

Maximum depth of cut for side cutting (turning/profiling) depends on the material being cut and the width of the tool.

- .031–.062" (0,79–1,6mm) wide insert can cut up to .025" (0,6mm) deep.
- .067–.128" (1,7–3,3mm) wide insert can cut up to .040" (1mm) deep.
- .138–.189" (3,5–4,8mm) wide insert can cut up to .080" (2mm) deep.
- .197–.250" (5–6,35mm) wide insert can cut up to .120" (3mm) deep.

Finish Turning the Groove



1. Plunge both sides of groove width.
2. Plunge center area to remove web of material remaining.
3. To avoid insert chipping and to achieve groove wall perpendicularity, follow the tool path outlined.
4. Use the lightest depth of cut that still allows good chipbreaking, tool life, and surface finish.

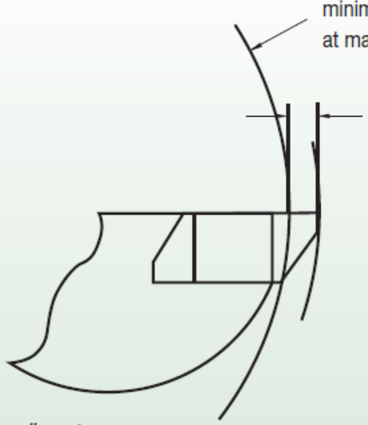
Groove Limits				
insert number	maximum internal groove depth		minimum bore diameter	
	inch	mm	inch	mm
NG-2031R/L	.050	1,27	.730	18,54
NG-2062R/L	.102	2,59	1.750	44,45
NG-2094R/L	.098	2,49	1.500	38,10
HNG-3047R/L	.075	1,91	.800	20,32
HNG-3062R/L	.094	2,39	1.750	44,45
HNG-3094R/L	.138	3,51	1.875	47,63
	.150	3,81	2.375	60,33
HNG-3125R/L	.138	3,51	1.875	47,63
HNG-3189R/L	.138	3,51	1.875	47,63
HNG-4125R/L	.150	3,81	2.750	69,85
HNG-4189R/L	.245	6,22	5.000	127,00
HNG-4250R/L	.200	5,08	2.500	63,50

NOTE: The same maximum groove depth and minimum bore diameter values also apply to metric, HNG-K (chip control), and HNR and HNR-K (full radius) inserts of similar size.

The same internal grooving depth limits are a function of bar clearance versus bore diameters.

Technical Details

Internal Groove Depth versus Bar Interference



NOTE: Internal grooving depth limits are a function of bar clearance versus bore diameters.

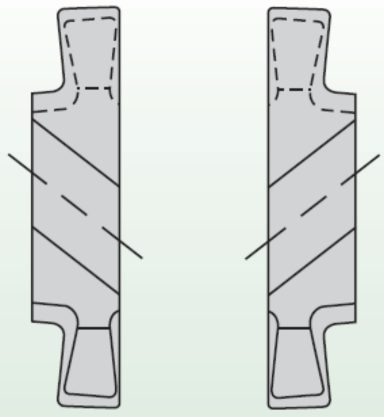
Haas Notch Insert Selection Guide

All Haas Notch inserts are precision ground to provide accurate edge location and secure locking of the insert in the toolholder pocket.

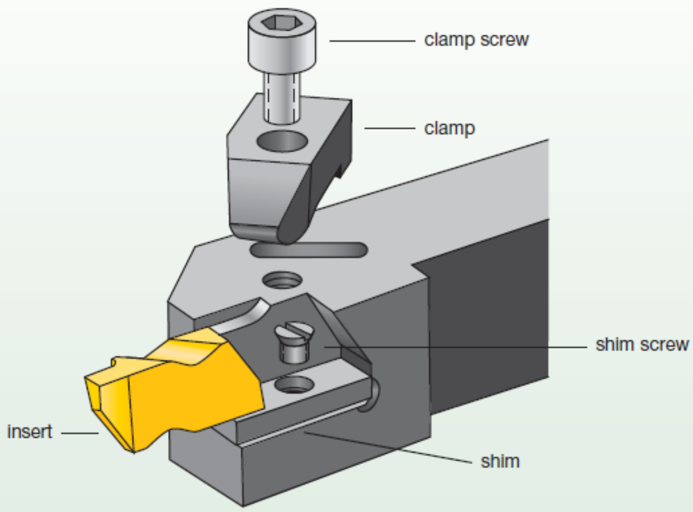
Haas Notch inserts can be used in either toolholders or boring bars.

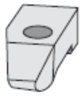
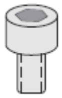
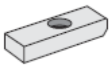
Right-hand Haas Notch toolholders use right-hand inserts. Left-hand Haas Notch toolholders use left-hand inserts.

Right-hand Haas Notch boring bars use left-hand inserts. Left-hand Haas Notch boring bars use right-hand inserts.



Tool Holder Accessories



insert size and style	 clamp	 clamp screw	 shim	 shim screw

Applies to Both OD and ID tool holders
Shim and Shim screw not required for all tool holders. Check related products