Technical Details

Application Grade

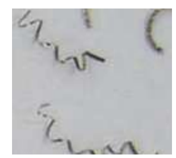
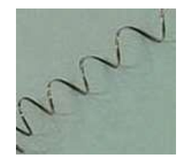
Grade	Features		Available insert type
HU30T	Universal grade	• For chip breaker type only • Stable machining on a wide application due to fine-grained carbide substrate with balanced heat resistance and toughness • Excellent wear resistance and oxidation resistance due to AiTN coating film Outstanding performance on high speed machining	CFA/CFB (Insert with Chip breaker)
HU35T	Specialized grade for threading inserts	• A tough sub-micron substrate with TiAIN coating provides good fracture toughness and excellent wear resistance • Outstanding performance on STS and hard to cut materials	ER/IR (Ground insert)

Application Range

Workpiece		● ←- - - - - - - - - - - - - - - - - - → ✱
P	Carbon steel, Alloy steel, Cast Steel	HU30T HU35T
M	Stainless steel, Heat resistant steel, Titanium alloy steel	HU30T HU35T
K	Carbon Iron, Cast Steel	HU30T HU35T
N	Aluminum, Copper	HU30T HU35T

Technical Details

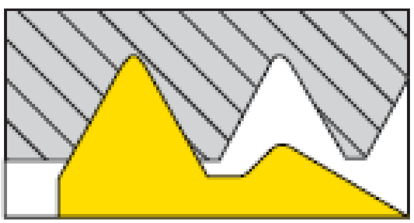
Threading Insert with Chip Breaker

Type	Ground insert without chip breaker		Insert with a chip breaker			
C/B Code	None		CFB		CFA	
Designation Example	ER16-A60		ER-16-AG60-CFB		ER16-AG60-CFA	
Machining	External	Internal	External	Internal	External	Internal
Insert shape						
Chip shape						
Class	P, M, K, N, S		P, M, K		P, M, K	
Features	• Groove-shaped chip breaker with superior chip evacuation lowers cutting load • Enables high precision machining • Applicable for machining of various shapes of threads • Applicable for machining of various workpieces		• Unique 3 dimensional chip breaker improves machinability with good chip control • Excellent cutting edge treatment technology ensures high precision sharp cutting edge		• Groove-shaped chip breaker with superior chip evacuation lowers cutting load • Reduces machining pass by 10~30% • Excellent cutting edge treatment achieves high precision sharp cutting edge	

Technical Details

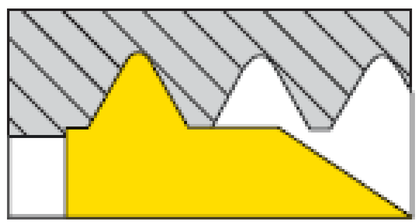
Insert profile style

Partial profile



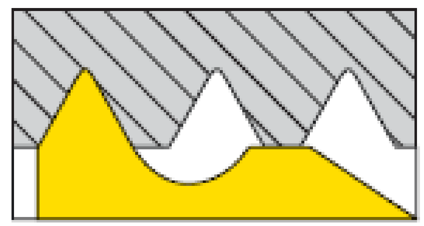
The V partial profile insert cuts without topping the outer diameter of the thread. The same insert can be used for a range of different thread pitches which have a common thread angle

Full profile



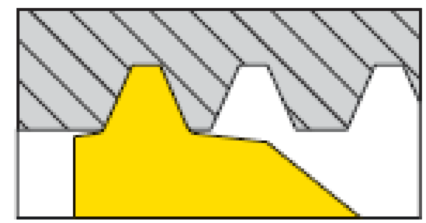
The full profile insert will form a complete thread profile including the crest. For every thread pitch and standard, a separate insert is required

Full Profile for fine pitches



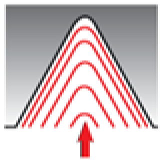
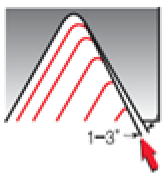
The full profile for Fine Pitches will form a complete thread
The topping of the outer diameter is generated by second tooth

Semi full



The Semi profile insert will form a complete thread including crest radius but without topping the outer diameter Mainly used for trapezoidal profiles

Thread infeed method

Infeed		Application	
	Radial infeed	• When the pitch is smaller than 16 tpi • For material with short chips • For work with hardened material	Radial infeed is the simplest and quickest method The feed is perpendicular to the turning axis, and both flanks of the insert perform the cutting operation Radial infeed is recommended in 3 cases
	Flank infeed (modified)	• When the thread pitch is greater than 16 tpi. Using the radial method,the effective cutting edge length is too large, resulting in chatter. for TRAPEZ and ACME. The radial method results in three cutting edges, making chip flow very difficult	Flank infeed is recommended in the following cases
	Alternate flank infeed	• This method divides the load equally on both flanks, resulting in equal wear along the cutting edges Alternate flank infeed requires more complicated programming, and is not available on all lathes	Use of the alternate flank method is recommended especially in large pitches and for materials with long chips

*Four options for `G76` Multiple Thread Cutting are available:

1. `P1`:Single edge cutting, cutting amount constant
2. `P2`:Double edge cutting, cutting amount constant
3. `P3`: Single edge cutting, cutting depth constant
4. `P4`: Double edge cutting, cutting depth constant

`P1` and `P3` both allow for single edge threading, but the difference is that with `P3` a constant depth cut is done with every pass. Similarly, `P2` and `P4` options allow for double edge cutting with `P4` giving constant depth cut with every pass. Based on industry experience, double edge cutting option `P2` may give superior threading results.

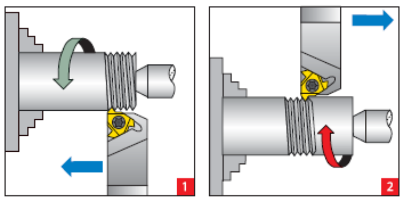
*See Haas lathe manual G-Code section for complete programming information on G76

Technical Details

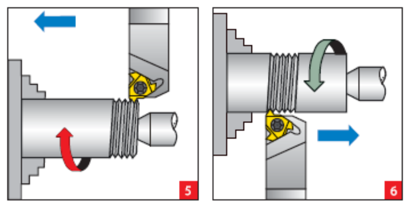
Thread turning method

Thread	Inserts & Tool holder	Rotation	Feed direction	Helix method	Drawing no.
Right Hand Externa	EX RH	Counter clockwise	Towards chuck	Regular	1
	EX LH	Clockwise	From chuck	Reversed	2
Right Hand Internal	IN RH	Counter clockwise	Towards chuck	Regular	3
	IN LH	Clockwise	From chuck	Reversed	4
Left Hand External	EX LH	Clockwise	Towards chuck	Regular	5
	EX RH	Counter clockwise	From chuck	Reversed	6
Left Hand Internal	IN LH	Clockwise	Towards chuck	Regular	7
	IN RH	Counter clockwise	From chuck	Reversed	8

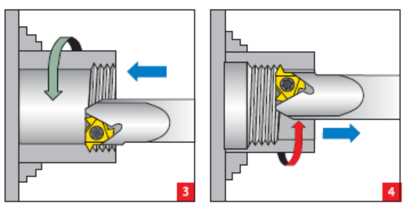
External RH thread



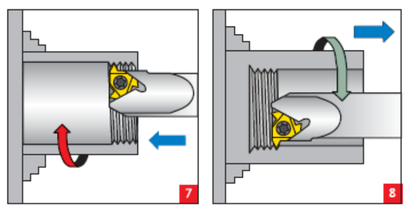
External LH thread



Internal RH thread



Internal LH thread



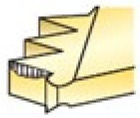
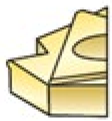
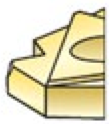
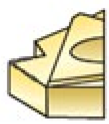
Number of passes

Pitch																	
Pitch	mm	0.50	0.75	1.00	1.25	1.50	1.75	2.00	2.50	3.00	3.50	4.00	4.50	5.00	5.50	6.00	8.00
	tpi	48	32	24	20	16	14	12	10	8	7	6	5.5	5	4.5	4	3
No. of passes		4~6	4~7	4~8	5~9	6~10	7~12	7~12	8~14	9~16	10~18	11~18	11~19	12~20	12~20	12~20	15~24

※ One cutting depth is calculated by total cutting depth divided into machining times
ex) ER16-1.5ISO, hmin 0.92: If 10 times machining, one cutting depth is 0.092 (0.92/10)

Technical Details

Trouble shooting

Problem		Possible cause	Solution
	Increased flank wear	Cutting speed too high	Reduce cutting speed/ use coated insert
		Depth of cut too low/too many passes	Increase the depth of cut per pass
		Unsuitable carbide grade	Use a coated carbide grade
		Insufficient cooling	Increase coolant flow rate
	Uneven cutting edge wear	Incorrect helix angle	Choose the correct shim
		Wrong infeed method	Use the Alternating Flank Infeed method
	Extreme plastic deformation	Depth of cut too large	Decrease depth of cut/ increase number of passes
		Insufficient cooling	Increase coolant flow rate
		Cutting speed too high	Reduce cutting speed
		Unsuitable carbide grade	Use a tougher carbide
		Nose radius too small	Use an insert with a larger radius, if possible
	Cutting edge breakage	Depth of cut too large	Decrease depth of cut/ increase number of passes.
		Extreme plastic deformation	Use a tougher carbide
		Insufficient cooling	Increase flow rate and/ or correct flow direction
		Unsuitable ca rbide grade	Use a tougher carbide
		Instability	Check stability of the system
	Built-up edge	Incorrect cutting speed	Change the cutting speed
		Unsuitable carbide grade	Use a coated carbide
	Thread profile is too shallow	The tool is not at the workpiece axis height	Change tool height
		Insert is not machining the thread crest	Measure the workpiece diameter
		Worn insert	Change the cutting edge sooner
	Poor surface quality	Too low cutting speed	Increase cutting speed
		Wrong shim	Choose correct shim
		Flank infeed method is not appropriate	Use the alternate flank or radial infeed method