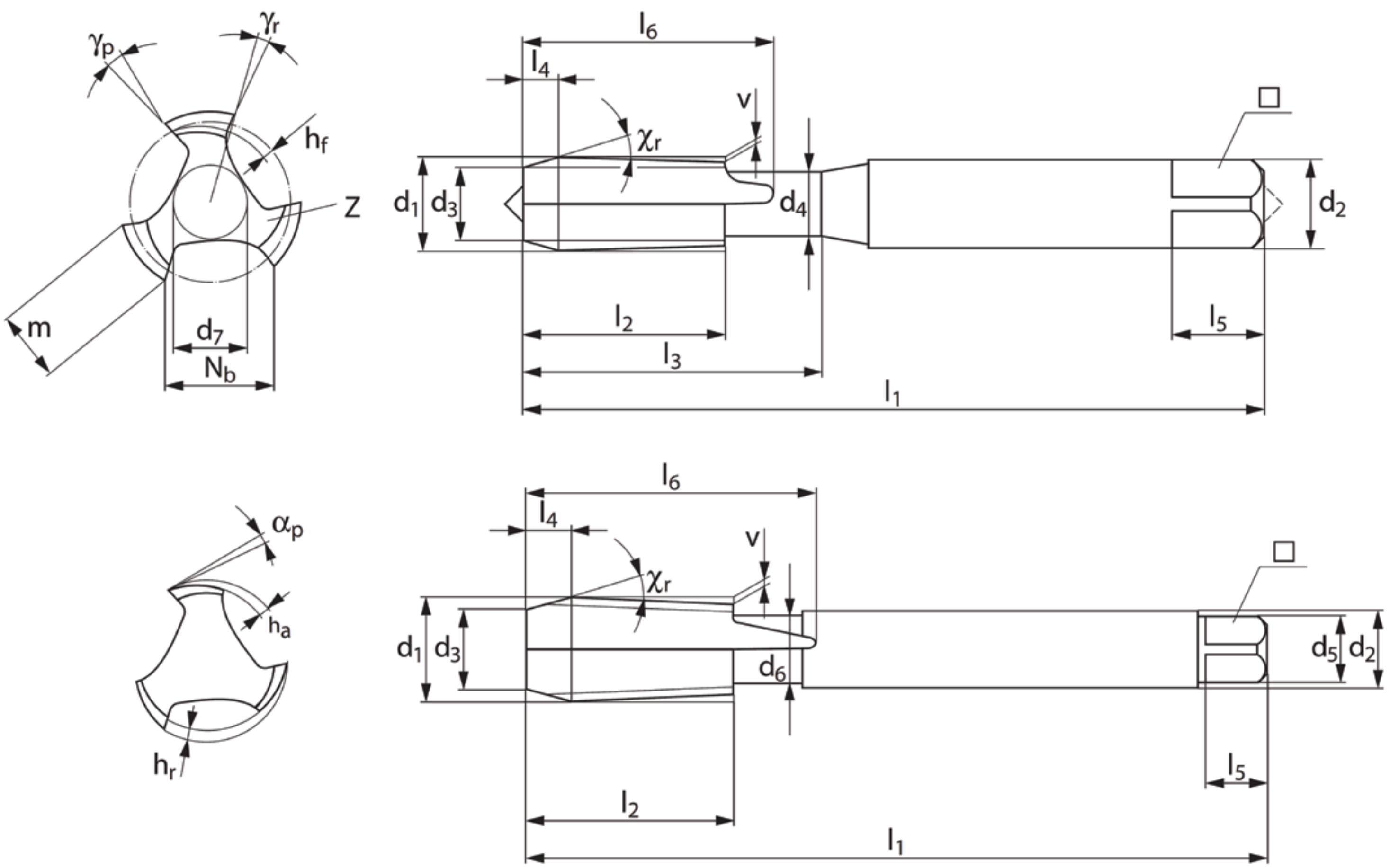
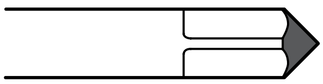
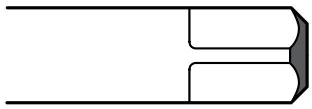
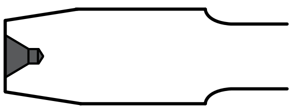
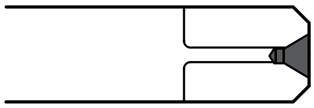
Technical Information

Tap Geometries



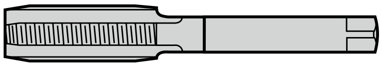
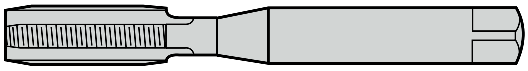
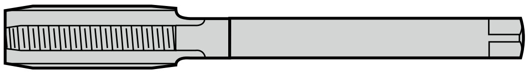
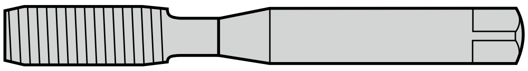
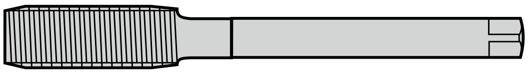
d1	Thread diameter		Z	Number of flutes
d2	Shank diameter		v	Back taper (axial relief)
d3	Chamfer diameter		xr	Chamfer angle
d4	Neck diameter		Nb	Width of flute
d5	Recessed square diameter		m	Width of land
d6	Neck diameter		ha	Chamfer relief
d7	Web diameter		hf	Relief on flanks
l1	Total length		hr	Chamfer relief
l2	Thread length		αp	Relief angle
l3	Useful length		γp	Rake angle
l4	Chamfer lead length			
l5	Square length			
l6	Flute length			
☐	Sqaure dimension			

Tap Centering

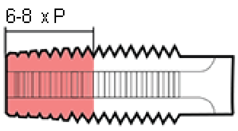
 Centre point	Ø 0,8 mm - 6,0 mm	 Centre point
 Centring point set off	Ø 6,0 mm - 12,8 mm	 Centring bevel
 Centring hole	> Ø 12,8 mm	 Centring hole

Technical Information

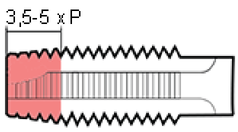
Tap Construction dimensions

 Gewindebohrer / Tap	DIN 352	DIN 2181	DIN 5157	DIN 40432	
 Gewindebohrer / Tap	DIN 371	DIN 40435			
 Gewindebohrer / Tap	DIN 376	DIN 374	DIN 5156	DIN 40433	DIN 40435
 Gewindeformer / Forming Tap	DIN 2174	DIN 371			
 Gewindeformer / Forming Tap	DIN2174	DIN 376	DIN 2189		

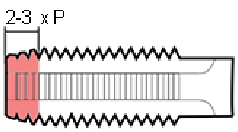
Tap Chamfer Forms



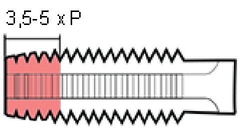
Form A
long, 6 - 8 thread
straight flutes
for short through holes



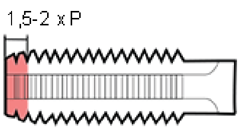
Form B
medium, 3,5 - 5,5 threads
straight flutes with spiral point
for through holes in medium- and long chipping materials



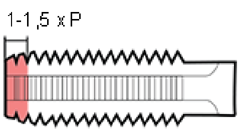
Form C
short, 2 - 3 threads
straight and spiral flutes
for blind holes in medium- and long chipping materials
for through holes in short chipping materials



Form D
medium, 3,5 - 5 threads
straight and spiral flutes 15°
for blind holes with long thread runout
for short through holes



Form E
very short, 1,5 - 2 threads
straight and spiral flutes 15°
for blind holes with short thread runout



Form F
very short, 1 - 1,5 threads
straight and spiral flutes
for blind holes with short thread runout
Avoid if possible.

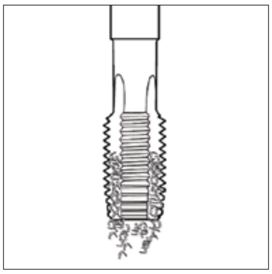
Please note:
Short chamfers cut threads close to the bottom of the borehole.
Long chamfers reduce the forced on the cutting edges (recommended for materials with higher material strength)
Long chamfers increase the required torque.

Technical Information

Tap flutes

Form A, C, E
straight-fluted
for through and blind holes

The flutes can hold a part of the chips. Chips get only partically removed in cutting direction. For this reason it is not recommended to use the tap for deep holes.

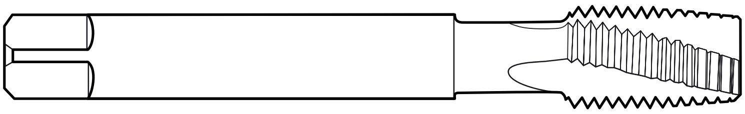


Form B
straight-fluted with spiral point
for through holes

Due to the spiral point the chips are getting removed tightly rolled in cutting direction and prevents chip-packing.
Coolant-lubricant can flow freely.

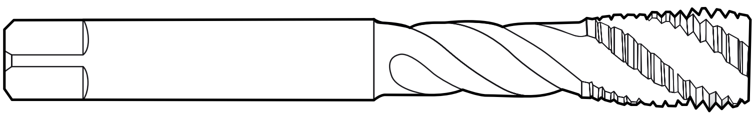
Form C, D
8 - 15° left-hand spiral flutes
for through holes

Due to the left-hand spiral flutes the rake angle remains constant and ensure stable chamfer teeth to produce threads in high-strength materials.
The left-hand spiral flutes forces the chips to remove ahead of the tap.



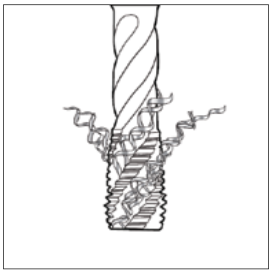
Form C, E
10 - 15° right-hand spiral flutes
for blind holes

Especially suitable for automatic lathes and multi-spindle machines. Due to the chip removal against the cutting direction a secured tapping process is assured in hard conditions, even for threads with cross holes.



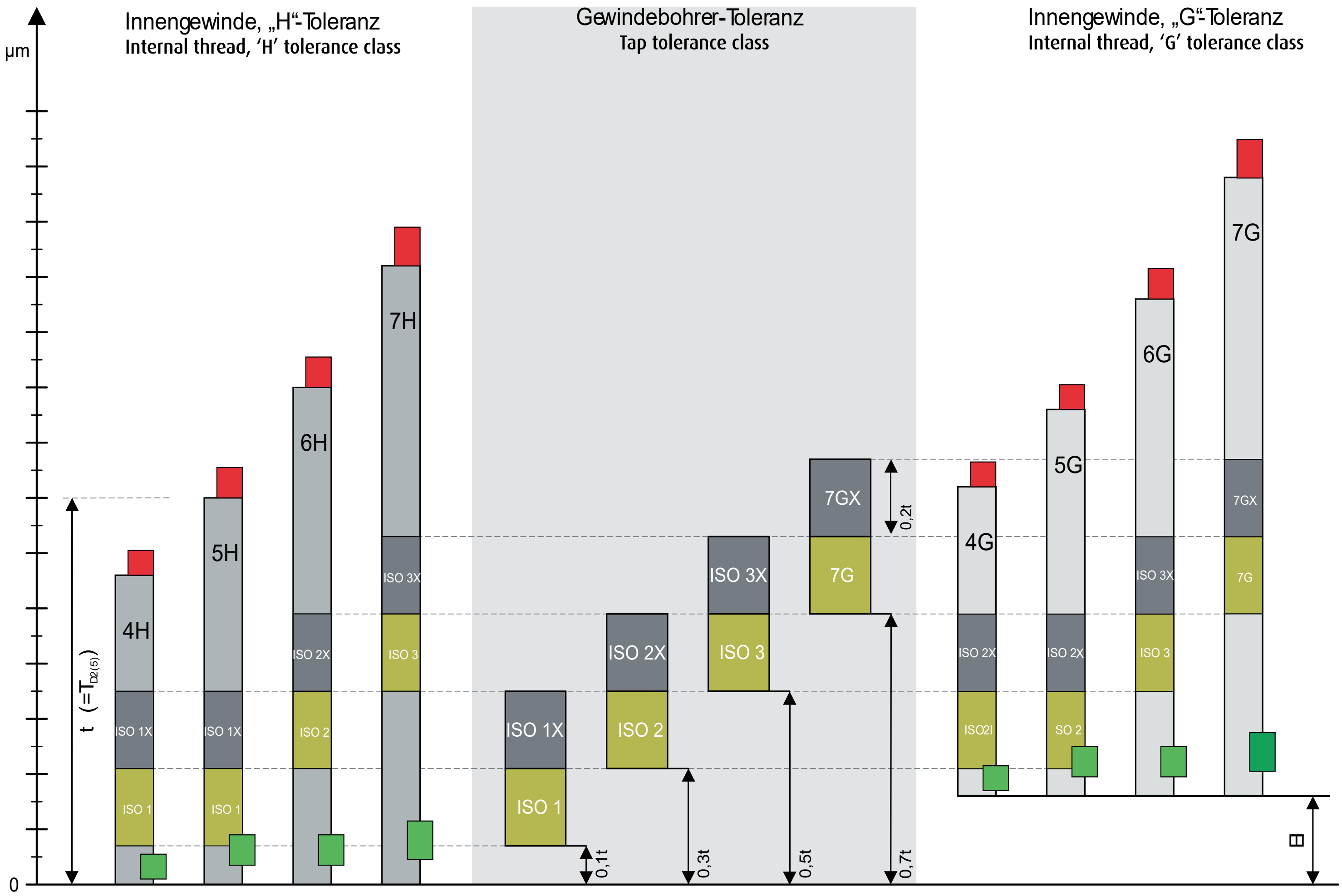
Form C, E
35 - 50° right-hand spiral flutes
for blind holes

Due to the high spiral flutes, the chips can be removed securely also in long-chipping and deep blind holes.



Technical Information

Tolerances



- Flankendurchmesser-Toleranz des Innengewindes nach DIN ISO 965-1
  Pitch diameter tolerance of internal thread acc. DIN ISO 965-1
- Gewindebohrer mit spezieller Flankendurchmesser-Toleranz
  Taps with specific pitch diameter tolerance
- Flankendurchmesser-Toleranz des Gewindebohrers nach DIN EN 22857
  Pitch diameter tolerance of the tap acc. DIN EN 22857
- Flankendurchmesser-Toleranz des Ausschusslehrdorns nach DIN ISO 1502
  Pitch diameter tolerance of the no-go thread plug gauge acc. DIN ISO 1502
- Flankendurchmesser-Toleranz des Gutlehrdorns nach DIN ISO 1502
  Pitch diameter tolerance of the go thread plug gauge acc. DIN ISO 1502

EI = Grundmaß
Basis

t = Toleranzklasse 5 des Innengewindes (Toleranzeinheit)
Tolerance class of the internal thread (tolerance unit)

TD2 = Toleranz des Flankendurchmessers
Pitch diameter tolerance

tolerance class of tap		tolerance field internal thread					remarks	Applications
4H (DIN 802/1)	ISO 1	4H	5H				undersize	Threads with small clearance
6H (DIN 802/1)	ISO 2	4G	5G	6H			normal	Thread with normal clearance
6G (DIN 802/1)	ISO 3			6G	7H	8H	oversize	Threads with large clearance
7G (DIN 802/4)					7G	8G	oversize	before heat treatment, causing distortion
6H +0,1							oversize	Electroplating allowance ≈ 25µm thickness
6H +0,2							oversize	Electroplating allowance ≈ 50µm thickness

Technical Information

Tap Surface Treatments

TIN (titanium nitride) coating
The TIN surface treatment (titanium-nitride gold-yellow) increases the surface hardness (approx. 2300 HV) and the sliding properties. It provides a better cutting performance and increased tool life time.

VAP (vaporized - steam tempered)
The oxide surface (vaporized) improves the adhesion of the cutting oil and provides a stabil lubricant film. Cold welding in the tap flanks is reduced.

TiAIN (titanium aluminium nitride) coating
The TiAIN surface treatment increases the surface hardness (approx. 3300 HV), the sliding properties (friction coefficient: 0,25) and with high temperature resistance up to 800°C. It provides a better cutting performance and increased tool life time.

TiCN (titanium carbonitride) coating
The TiCN surface treatment (titanium carbon nitride - grey violet) increases the surface hardness (approx. 3000 HV) and the sliding properties (coefficient of friction: 0,3). It provides a better cutting performance and increased tool life time.

Coating services
Every threading tool of our product range - whether or not cataloged - can be delivered with any coating or surface treatment in short time.

Troubleshooting guide for tapping

Tapping undersized threads	Pitch error Too small tolerance Geometry of the tap is not suitable for the material
Tapping oversized threads	Cutting speed is too high Bad running accuracy Chip jammings in flutes Incorrect positioning of workpiece or tap Inconsistent feed of tap Too high tolerance
Poor thread surface	Cutting speed not suitable Cooling is not suitalbe or not existing Geometry of the tap is not suitable for the material Core hole too small
Thread break of the thread to be cut	Geometry of the tap is not suitable for the material Core hole too small Core hole not deep enough Chip jammings in flutes Bad running accuracy Incorrect positioning of workpiece or tap Inconsistent feed of tap
Low tap lifetime	Cutting speed not suitable Cooling is not suitalbe or not existing Geometry of the tap is not suitable for the material Surface treatment/ coating for tap needed Core hole too small
Tool outbreaks	Geometry of the tap is not suitable for the material Core hole too small Core hole not deep enough Chip jammings in flutes Bad running accuracy Incorrect positioning of workpiece or tap Inconsistent feed of tap Worn tap
Cold welding on the tap	Cutting speed not suitable Increase coolant supply

Technical Information

Tap Cutting speeds

The cutting speeds for taps depend on following parameters:
- material of the workpiece
- tap geometries
- lubrication
- and more.

You can fin the cutting speed in the product describtions of the taps in our online-shop. Just enter the item number into the Search-field and find the cutting speeds in the bottom part of the product describtion.

Lubrication and cooling

It is recommended to use one of the following coolings or lubrications to improve the thread results andtool life time.

Dry machining and pressurized air
- cast iron
- (Cooled) pressurized air is used for chip removal

Emulsion
- Most common coolant-lubricant for thread cutting

Thread cutting oil
- Achieving excellent thread surfaces and tool life time

Thread cutting paste
- suitable for forming taps
- good results with horizontal cutting direction
- for bigger diameters and through holes

MQL - Minimum-quantity lubrication
- cooling by aerosol

Technical Information

Formula

Describtion	Symbol	Unit	Formula	Example
Rotation speed	n	min^{-1}		
Angular speed	ω	s^{-1}	$\omega = \frac{2 \cdot \pi \cdot n}{60}$	$n = 400\ min^{-1}$ $\omega = \frac{2 \cdot \pi \cdot 400}{60}$ $\omega = 41{,}89\ s^{-1}$
Cutting speed	v_c	m/min	$v_c = \frac{d \cdot \omega}{33{,}3}$ $v_c = \frac{d \cdot \pi \cdot n}{1000}$	$n = 400\ min^{-1}$, $d = 20$ mm $v_c = \frac{20 \cdot \pi \cdot 400}{1000}$ $v_c = 25{,}133$ m/min
Feed per revolution	f	mm		
Feed speed	v_f	mm/min	$v_f = f \cdot n$	$n = 400\ min^{-1}$ $f = 0{,}800$ mm $v_f = 0{,}800 \cdot 400$ $v_f = 320$ mm/min
Force	F	N	$F = k_c \cdot A$	
Torque	M	Nm	$M = \frac{F \cdot d}{2000}$	$F = 200$ N, $d = 20$ mm $M = \frac{200 \cdot 20}{2000}$ $M = 2$ Nm
Mechanical work	W	J	$W = F \cdot U$	$F = 200$ N, $d = 20$ mm $U = \frac{d \cdot \pi}{1000} = 0{,}063$ m $W = 200 \cdot 0{,}063 = 12{,}6$ J
Performance	P	W	$P = \frac{F \cdot v_c}{60}$ $P = M \cdot \omega$	$F = 200$ N, $v_c = 25{,}133$ m/min $P = 83{,}78$ W $M = 2$ Nm, $\omega = 41{,}89\ s^{-1}$ $P = 83{,}78$ W
Efficiency	η	–	$\eta = \frac{P_{ab}}{P_{an}}$ $\eta < 1$	$P_{ab} = 58{,}65$, $P_{an} = 83{,}78$ W $\eta = \frac{58{,}65}{83{,}78} = 0{,}7$

$\pi = 3{,}141592654$
d = Diameter in mm
U = circumference in m
1 kW = 1,36 PS
1 PS = 0,736 kW

$1\ \frac{m}{s} = 60\ \frac{m}{min}$

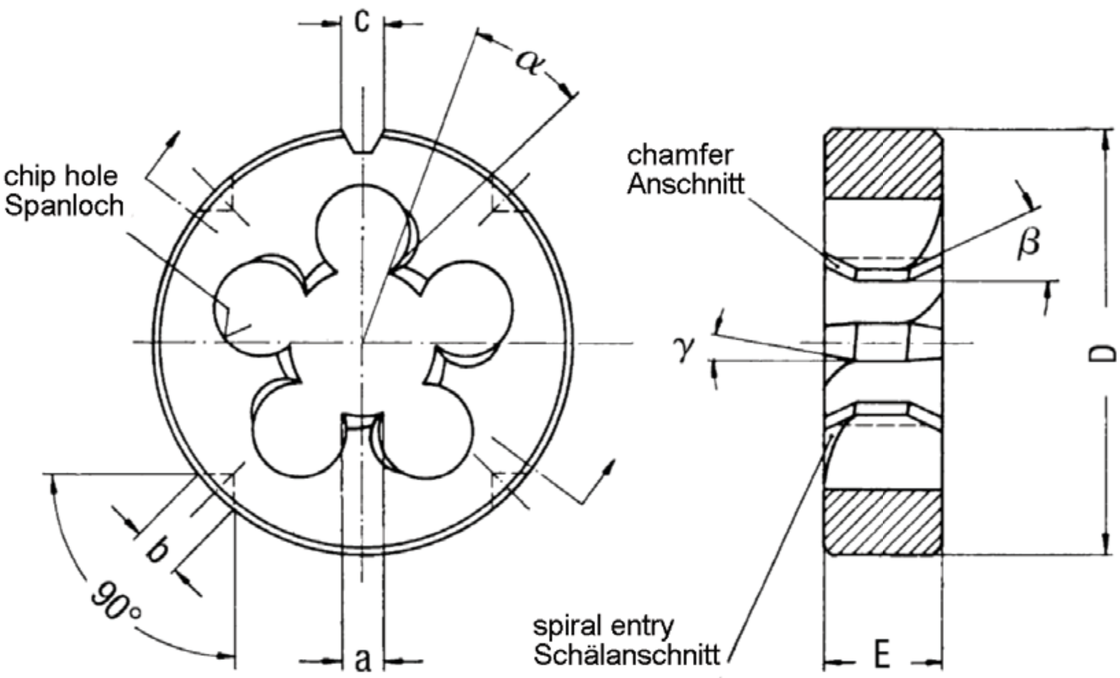
$1/s = 60/min = 1\ Hz$

$1J = 1\ Nm = 1\ \frac{kg \cdot m^2}{s^2}$

$1W = 1\ \frac{J}{s} = 1\ \frac{Nm}{s} = 1\ \frac{kg \cdot m^2}{s^3}$

Technical Information

Round Dies Geometries



D	outside diameter	b	diameter of hole for fixing screw
E	thickness	α	rake angle
a	width of tooth	β	chamfer angle
c	width of notch	γ	spiral angle

Preparation of workpiece:
- concentrical chamfer - ease entry of cutting
- workpiece diameter must be less than the nominal diameter

Chamfer
- Standard chamfer (length, see at the item)
- Spiral entry: free flow of chips ahead of the die and a reducing of cutting torque.
  This result in an improved surface finish on the cut threads and longer die life
- 70° (short chamfer) - chamfer length: 1,25 pitch = 70°

Tolerances for Round Dies

Metric ISO thread acc. DIN 13 - coarse and fine thread

4h = tolerance class „fine“
6h = tolerance class „middle“, for small diameters (up to M 1,4)
6g = tolerance class „middle“ - Standard
6e = undersize tolerance; for bolts that receive a surface treatment or a galvanizing (layer thickness up to 8 µm).
Cutting dies with 6e tolerance are cutting about 0,03 mm smaller than normal cutting dies with 6g tolerance.
8e = undersize tolerance; for bolts that receive a strong surface coating (layer thickness about 16 - 18 µm)

Unified thread UNC, UNF, UNEF, UNS, UN, UNJC, UNJF etc.

3A = tolerance „fine“
2A = standard tolerance „middle“
1A = tolerance „coarse“

Whitworth pipe thread G (BSP) acc. DIN-ISO 228

A = standard tolerance „middle“
B = tolerance „coarse“

Technical Information

Cutting speeds for Machine Forming Taps

materials	tensile strength	forming speed in m/min	recommended lubrication
construction steels, free-macining steels, cold-extrusion steels etc.	< 600 N/mm²	20 - 80	Cutting oil/ Emulsion
construction steels, heat-treatable steels, cast steels etc.	< 800 N/mm²	20 - 60	Cutting oil/ Emulsion
heat-treatable steels, cold-extrusion steels, nitriding steels etc.	< 1000 N/mm²	10 - 40	Cutting oil
corrosion and acid proof steels ferritic, martensitic	< 950 N/mm²	10 - 25 (with emulsion just limitedly applicable)	Cutting oil
corrosion and acid proof steels austenitic	< 950 N/mm²	10 - 25 (with emulsion just limitedly applicable)	Cutting oil
aluminium wrought alloys	< 550 N/mm²	15 - 40	Cutting oil/ Emulsion
aluminium cast alloys	Si < 12%	15 - 40	Cutting oil/ Emulsion
pure copper	< 400 N/mm²	20 - 40	Cutting oil/ Emulsion
copper-zinc alloys (brass long-chipping)	< 550 N/mm²	40 - 80	Emulsion

forming (cutting) speed [m/min] = (diameter * π * number of rotation) / 1000
number of rotation n [1/min] = (cutting speed in m/min * 1000) / (diameter * π)
feed programming [mm/min] = number of rotation * pitch
Please notice that the mentioned cutting speeds are only for orientation.
The right cutting speed is depend on lubrication and application.

BAER Vertriebs GmbH
Robert-Bosch-Str. 5
68542 Heddesheim
Germany
Tel: +49 (0) 6203 4048 790
Fax: +49 (0) 6203 4048 791
E-Mail: info@gewindebohrer.de
www.threading-tools.com