WALTER

Product handbook

Threading

_ THREADING WITH WALTER PROTOTYP

Precise, reliable, efficient

WALTER PROTOTYP

CONTENTS

Threading

Alphabetical keyword index

Technology, trends and innovations in thread production

There are different processes for producing a thread. In this handbook, we focus on **thread tapping**, **thread forming** and **thread milling** with tools from Walter Prototyp. In addition, this handbook also presents general technical information on these processes.

Thread tapping is still the most frequently used process for producing internal threads. Process reliability, quality and production costs per thread are the main considerations when developing tools. Great efforts have been made in the field of macro/micro geometry as well as into coatings, in order to guarantee a high level of process reliability even under unfavourable conditions. The costs per thread can be reduced sharply through the use of our high-performance tools from the Eco and Synchrospeed series. Even lower costs per thread can be achieved with solid carbide tools. Our HSC line is setting new standards in this regard – even in steel materials.
These tools are the first choice for mass production, for example in the fastener or automotive industries.

Thread forming as a process for producing internal threads has developed rapidly in the last 20 years. While in the past, oil was predominantly required as a lubricant with these tools, today, thanks to targeted further development of the shaped edge geometry and the coating, it is possible to form nearly all formable materials (even stainless steels) with a 5% emulsion on any machining centre. In addition, the static and particularly the dynamic tensile strength of the formed thread has been improved even further through the use of emulsion.

Carbide as a cutting tool material found its way into thread forming a long time ago. Absolute peak values are achieved today using our Protodyn® HSC line.

Thread forming is often the most cost-efficient method of producing an internal thread, provided that this process is permitted for the respective component.

In terms of process reliability and thread quality, **thread milling** is unchallenged at the top. Alongside the classic milling process, what is known as **"Orbital-thread milling"** has made a name for itself in recent times. With this method, users are able to produce very deep (e.g. 3 x D_N) and moreover very small (e.g. M1.6) internal threads even in demanding materials with absolute reliability.

And one final tip: Use our new **Walter GPS** software, the successor to the proven CCS, to select the ideal process. Here, you can compare all production processes with each other and decide on the most cost-efficient alternative.

WALTER PROTOTYP

Productive processes with Walter Prototyp

Nowadays, it is practically impossible to directly pass on increasing production costs through increasing per-part costs straight to the customer. This applies equally to your consumable goods as well as to produced goods. Successful companies close this yield gap through a systematic productivity increase in production.

As a manufacturer of precision tools used in machining, we can contribute a lot, as the chart shows. The tool costs account for only 3% of the overall machining costs. The machining time which accounts for 30% of the machining costs is nevertheless a significant cost factor.

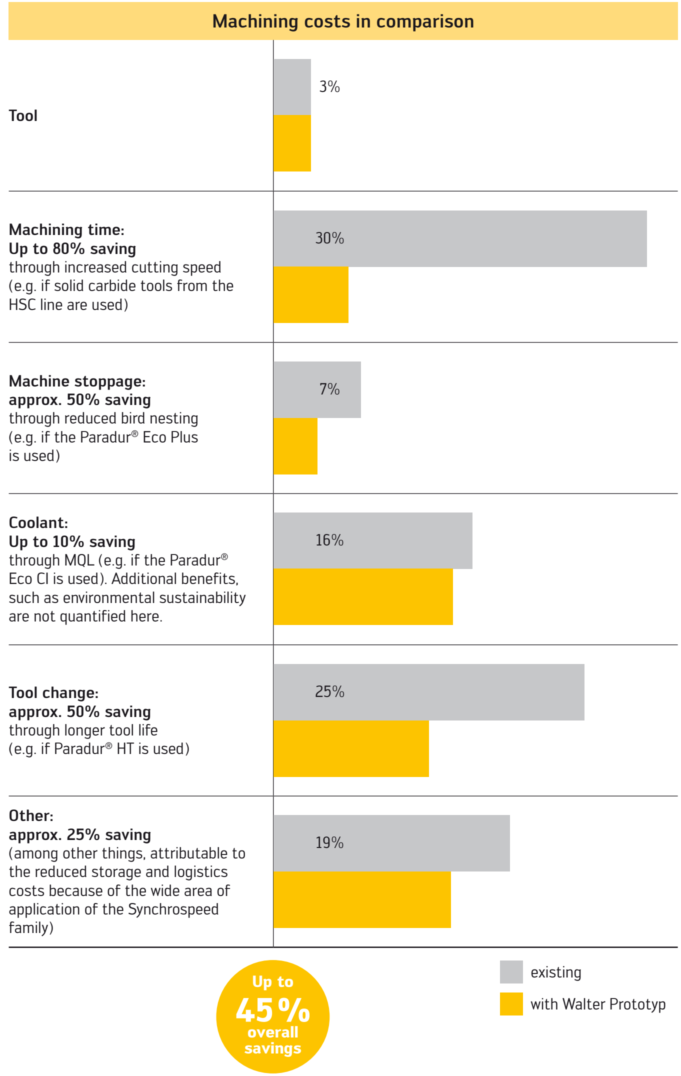
This means: with efficient metal cutting tools from Walter Prototyp, the machining costs can be reduced significantly. An increase in the cutting parameters leads to enormous cost savings. Because the tool price has an almost insignificant effect on the overall machining costs, tools from the competence brand Walter Prototyp are not measured solely on the tool price alone, but on the overproportional increase in productivity and therefore on the savings potential for our customers.

For this reason, at Walter Prototyp, we are strongly promoting the use of HSC machining (High Speed Cutting) with solid carbide tools from our product range. Therefore, when machining low alloy steels, for example, cutting speeds of up to 50 m/min are possible. For threading, this is a remarkable result! Particularly demanding customers for whom maximum productivity is of the utmost importance, Walter Prototyp has, in addition to the HSC line, specially developed tools for synchronous machining.

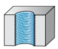
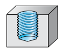
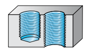
Minimum quantity lubrication (MQL) is an additional factor to consider when reducing the machining costs, as shown in the chart opposite. Walter Prototyp also offers it customers specially adapted coatings for MQL.

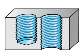
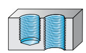
In short, the proportion of costs spent purely on tools may only be 3% of the actual production costs, but the tool has a decisive effect on the remaining 97% of the costs.

Allow our experts to demonstrate the savings potential in production to be gained through the use of tools from Walter Prototyp.



Walter Prototyp threading tool – Nomenclature/tool categories

Thread tapping*		
		
Prototex®… Tap with spiral point	**Paradur®…** Tap with right-hand helical flute	**Paradur®…** straight-fluted tools

Thread forming		Thread milling**
		
Protodyn®… Thread former without lubrication grooves	**Protodyn® S …** Thread former with lubrication grooves	**TM …** TM = Thread Mill…

*** Thread tapping exceptions:**

- Paradur® N with chamfer form D and Paradur® Combi: helical tools for producing through-hole threads
- Paradur® HT, Paradur® GG and Paradur® Engine: straight-fluted tools for blind hole threads (in materials with good chip breaking characteristics)
- NPT/NPTF taps: right-hand helical tools for machining blind and through holes

**** Thread milling exceptions:**

- TME (Thread Mill External): tool for producing external threads

Taps for universal applications

				Workpiece material group						
				P	M	K	N	S	H	O
Type description	Page in handbook	Application	Thread depth	Steel	Stainless steel	Cast iron	NF metals	Difficult-to-cut materials	Hard materials	Other
 **Prototex® Eco HT** – universal application – for wet and MQL machining	14 + 15	DL	3,5 x D_N	●●	●●	●●	●●	●		●
 **Paradur® Eco Plus** – universal application – for wet and MQL machining – successor to the proven Paradur® Eco HT	14 + 15	GL	3 x D_N	●●	●●	●●	●	●		●
 **Prototex® Synchrospeed** – synchronous machining – universal application – h6 shank tolerance	16 + 17	DL	3,0 x D_N	●●	●●	●●	●●	●●		●
 **Paradur® Synchrospeed** – synchronous machining – universal application – h6 shank tolerance	16 + 17	GL	2,5 x D_N	●●	●●	●●	●	●		●

GL = blind hole machining
DL = through hole machining

●● Primary application
● Additional application

Taps for special applications

Type description	Page in handbook	Application	Thread depth	P Steel	M Stainless steel	K Cast iron	N NF metals	S Difficult-to-cut materials	H Hard materials	O Other
				Workpiece material group						
 **Paradur® Eco CI** – for short-chipping materials – for wet and MQL machining	18	GL + DL	3 x D_N			●●	●●			●●
 **Paradur® HT** – for steels with medium to high tensile strength, and for short-chipping materials – Internal cooling required	19	GL	3,5 x D_N	●●		●●	●			●
 **Prototex® X·pert P** – for materials with low to medium tensile strength	20 + 21	DL	3 x D_N	●●			●			●
 **Paradur® X·pert P** – for materials with low to medium tensile strength	20 + 21	GL	3,5 x D_N	●●			●			●
 **Prototex® X·pert M** – for stainless and high-strength steels	22 + 23	DL	3 x D_N	●	●●					
 **Paradur® X·pert M** – for stainless and high-strength steels	22 + 23	GL	2,5 x D_N	●	●●					

Type description	Page in handbook	Application	Thread depth	P Steel	M Stainless steel	K Cast iron	N NF metals	S Difficult-to-cut materials	H Hard materials	O Other
				Workpiece material group						
 **Prototex® TiNi Plus** – for machining high-tensile Ti and Ni alloys with **emulsion** that tend to cause jamming	24 + 25	DL	2 x D_N					●●		
 **Paradur® Ti Plus** – for machining high-tensile Ti alloys with **emulsion** that tend to cause jamming	24 + 25	GL	2 x D_N					●●		
 **Prototex® HSC** – for high-strength and high tensile steel materials – h6 shank tolerance – Internal cooling required – Solid carbide	26	DL	2 x D_N	●●		●●				
 **Paradur® HSC** – for high-strength and high-tensile steel materials up to 55 HRC – h6 shank tolerance – Internal cooling required – Solid carbide	27	GL	2 x D_N	●●		●●			●●	

GL = blind hole machining
DL = through hole machining

●● Primary application
● Additional application

Thread formers

Type description	Page in handbook	Application	Thread depth	Workpiece material group P Steel	M Stainless steel	K Cast iron	N NF metals	S Difficult-to-cut materials	H Hard materials	O Other
 **Protodyn® S Eco Plus*** – For universal application – higher performance compared to Protodyn® S Plus – for wet and MQL machining	28	GL + DL	3,5 x D_N	●●	●●		●●	●		
 **Protodyn® S Plus*** – For universal application	29	GL + DL	3,5 x D_N	●●	●●		●●	●		
 **Protodyn® Eco LM** – For soft materials with tendency to cause jamming	30	GL + DL	2 x D_N	●			●●	●●		
 **Protodyn® S Eco Inox*** – especially for machining stainless steels with **emulsion**	31	GL + DL	3,5 x D_N	●	●●		●	●		
 **Protodyn® S Synchrospeed*** – For universal application – Synchronous machining – h6 shank tolerance	32	GL + DL	3,5 x D_N	●●	●●		●●	●		
 **Protodyn® S HSC*** – for high forming speeds – h6 shank tolerance – Solid carbide	33	GL	3,5 x D_N	●●	●		●●	●		

* Version with lubrication grooves, marked with an S

Thread mills

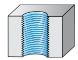
Type description	Page in handbook	Application	Thread depth	Workpiece material group P Steel	M Stainless steel	K Cast iron	N NF metals	S Difficult-to-cut materials	H Hard materials	O Other
 **TMC thread mill** – with countersink for universal application	34 + 35	GL + DL	2 x D_N	●●	●●	●●	●●	●●		●
 **TMG thread mill** – without countersink – For universal application	35	GL + DL	1,5 x D_N 2 x D_N	●●	●●	●●	●●	●●		●
 **TMO orbital thread mill** – For universal application in machining of small and deep threads	36 + 37	GL + DL	2 x D_N 3 x D_N	●●	●●	●●	●●	●●		●
 **TMO HRC orbital thread mill** – For small and deep threads in hard materials up to 65 HRC	37	GL + DL	2 x D_N	●●				●	●●	●
 **TMD thread milling cutter** – For aluminium and grey cast iron machining	38 + 39	GL + DL	2 x D_N			●●	●●			
 **TME thread mill 20** – for external threads	–	External thread	2 x D_N	●●	●●	●●	●●	●●		●

GL = blind hole machining
DL = through hole machining

●● Primary application
● Additional application

The high-tech all-rounder

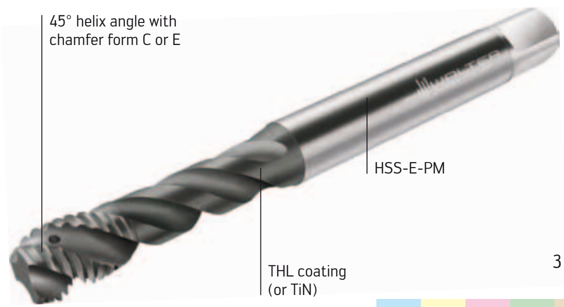
3,5 x D_N



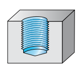
P	M	K	N	S	H	O
●●	●●	●●	●●	●		●

Variants: without IK, with KR*

Prototex® Eco HT **Type:** E2021342



3 x D_N



P	M	K	N	S	H	O
●●	●●	●●	●	●		●

Variants: without IK, with KA, with KR*

Paradur® Eco Plus **Type:** EP2051312

The tool

– universal high performance tap
– THL hard material coating minimises built up edges and guarantees long tool life

Prototex® Eco HT:

– special spiral point form B guarantees high process reliability

Paradur® Eco Plus:

– tapered guide reduces the tendency toward fractures
– thread nearly to the bottom of the hole with chamfer form E

The application

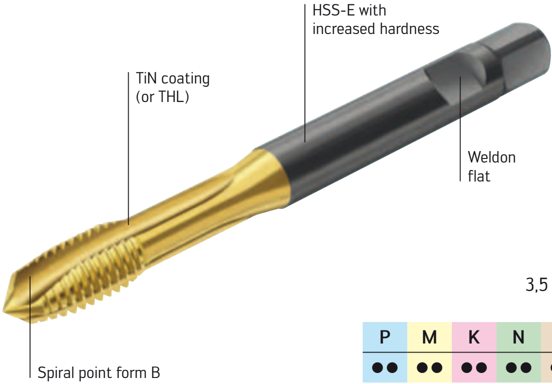
– for use in long and short-chipping materials with a tensile strength from approx. 200 N/mm² to approx. 1300 N/mm²
– suitable for synchronous machining and suitable for use in floating chucks

Your advantages

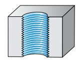
– reduction in tool inventory thanks to a wide area of application
– increased productivity through high cutting speeds and long tool life
– special geometry for safe processes, even in soft materials
– MQL machining possible

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

Wear-resistant, universal use

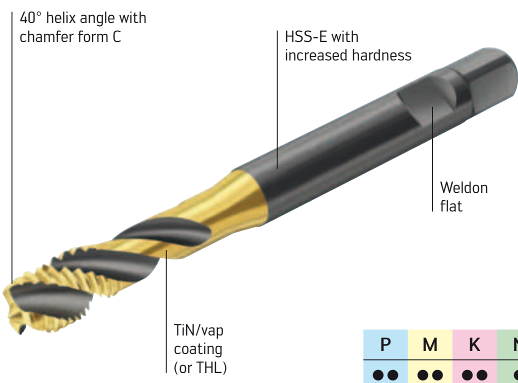


3,5 x D_N

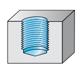


P	M	K	N	S	H	O
●●	●●	●●	●●	●●		●

Prototex® Synchrospeed **Type:** S2021305



2,5 x D_N



P	M	K	N	S	H	O
●●	●●	●●	●	●		●

Variants: without IK, with KA*

Paradur® Synchrospeed **Type:** S2051305

The tool

– high face clearance and short threading section for extremely high cutting speeds
– h6 shank tolerance (e.g. for use in shrink-fit chucks)
– shank diameter adapted to standard shrink-fit chuck

Special features of the Paradur® Synchrospeed:

– variant with TiN/vap coating: vaporised flutes for perfect chip formation and optimum chip removal; TiN coating for increased wear resistance
– internal cooling with axial output in the standard product range

Practical tip:

It is generally recommended to use adaptors with minimum compensation (e.g. Protoflex C) for synchronous machining (advantage: longer tool life and increased process reliability).

The application

– for use on machine tools with a synchronous spindle (not suitable for floating chucks or cutting units)
– for universal use in all long and short-chipping materials

Prototex® Synchrospeed:

– can be used up to approx. 1400 N/mm²

Paradur® Synchrospeed:

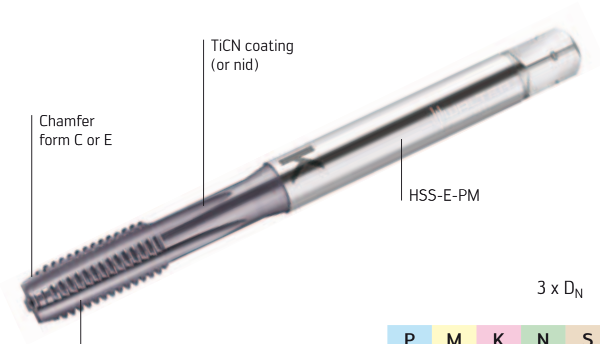
– can be used up to approx. 1300 N/mm²

Your advantages

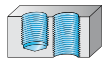
– increased productivity through high cutting speeds and long tool life
– reduced tool inventory costs through universal use in short- and long-chipping materials
– excellent thread surface thanks to very sharp cutting edges
– miscutting excluded through synchronous machining

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

Extremely high speed in short-chipping materials



3 x D_N



P	M	K	N	S	H	O
		●●	●●			●●

Variants: without IK, with KA, with KR*

Paradur® Eco CI **Type:** E2031416

The tool

– innovative surface treatment "Xtra·treat" for best wear behaviour when machining abrasive, short-chipping materials
– increased number of flutes reduces cutting edge load and produces short chips
– tolerance grade 6HX for maximum tool life
– versions with axial or radial coolant outlets for optimum chip evacuation with deep blind and through hole threads

Your advantages

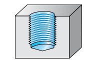
– lower production costs per thread as result of high cutting speeds and long tool life
– even wear behaviour and therefore absolute process reliability
– reduced tool costs, because it can be used for blind- and through hole threads
– MQL machining possible

The application

– blind- and through hole thread in short-chipping materials
– ISO K: primarily for GJL (GG) materials; in GJS (GGG) materials up to maximum 2 x D_N thread depth; vermicular cast iron (e.g. GJV450)
– ISO N: Mg alloys, and abrasive AlSi alloys with Si content > 12%

Short cycle time, optimum chip breaking

3,5 x D_N



P	M	K	N	S	H	O
●●		●●	●			●

KA absolutely essential*

Paradur® HT **Type:** 2031115

The tool

– cutting edge geometry produces short chips even in long-chipping materials
– axial internal cooling and straight flutes enable optimum transport of short broken chips
– increased face clearance for higher cutting speeds
– long versions with elongated flutes in the standard product range

Your advantages

– higher cutting speed and longer tool life compared to conventional blind hole taps
– no swarf packing, i.e. less machine stoppage
– extremely high process reliability even with deep threads
– Standard product range with large sizes

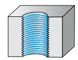
The application

– blind hole thread in long and short-chipping materials
– ISO P: steel material with tensile strength of 600 - 1,400 N/mm²,
– ISO K: grey cast iron (GGG)
– ISO N: AlSi alloys > 12% Si content, Cu alloys and Mg alloys
– typical areas of application:
  • automotive industry (camshafts, crankshafts,connecting rods)
  • large thread dimensions (general mechanical engineering, transmission shafts, housings, etc.)

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

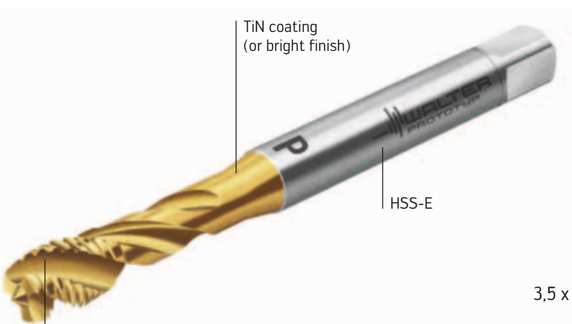
Large product range, high cost efficiency

3 x D_N

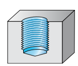


P	M	K	N	S	H	O
●●			●			●

Prototex® X·pert P **Type:** P2031005



3,5 x D_N



P	M	K	N	S	H	O
●●			●			●

Paradur® X·pert P **Type:** P2051905

The tool

– low flank clearance angle and therefore no miscutting in soft materials

Prototex® X·pert P

– variants with reduced number of flutes in the standard product range

Paradur® X·pert P

– long flutes for deep threads
– Tapered guide prevents fractures

The application

Prototex® X·pert P

– ISO P:
  - variant with 3 flutes: tensile strength < 1000 N/mm²
  - variant with 2 flutes: tensile strength < 700 N/mm² (available up to size M6)
– ISO N: AlSi alloys with Si content between 0.5 to 12%
– version with reduced number of flutes is ideally suited to soft, long-chipping materials (optimum for machining soft structural steels, e.g. St37) due to improved chip formation

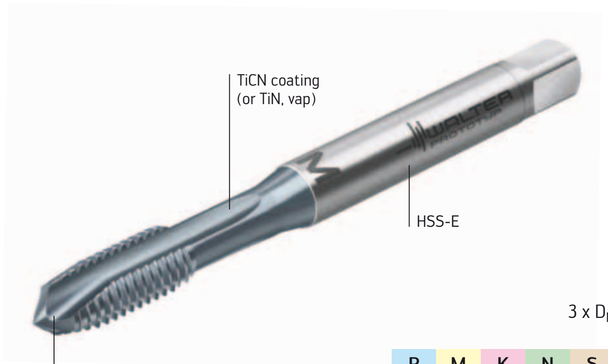
Paradur® X·pert P

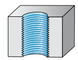
– ISO P: steel < 1000 N/mm², particularly in long-chipping materials
– ISO N: AlSi alloys with Si content between 0.5 to 12%

Your advantages

– cost-efficient for small and medium batch sizes
– high flexibility and short delivery times, because of the comprehensive standard product range (diverse thread profiles, sizes and tolerances in stock)
– thread with very good surface finish quality thanks to wide rake angle

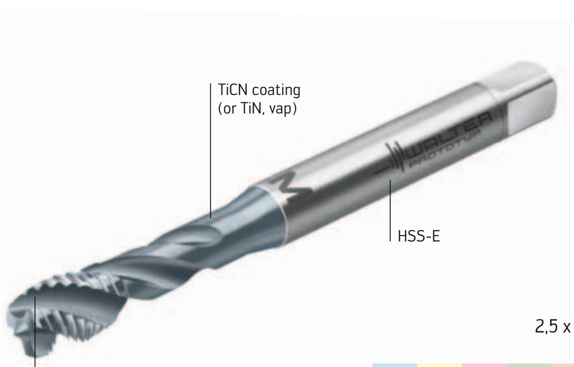
Reliable in stainless steels

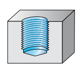



3 x D_N

P	M	K	N	S	H	O
●	●●					

Prototex® X·pert M **Type:** M2021306




2,5 x D_N

P	M	K	N	S	H	O
●	●●					

Paradur® X·pert M **Type:** M2051306

The tool

- raised core guarantees true to gauge threads and ensures reliable deburring in the thread – important above all for machining stainless materials
- increased flank clearance angle for machining materials that tend to cause jamming

Special features of the Paradur® X·pert M:

- tapered guide to prevent fractures

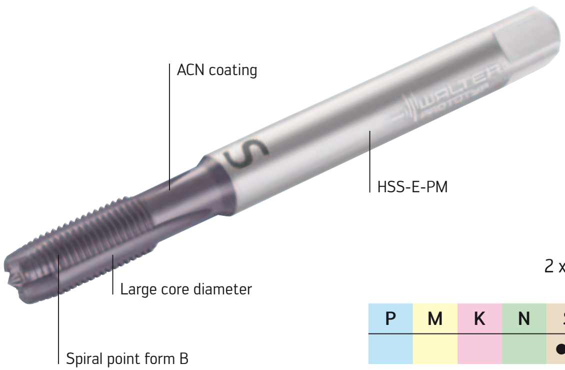
The application

- ISO M: stainless steels from 350 to 1200 N/mm²
- ISO P: very well suited to steels from 700 to 1200 N/mm²

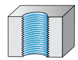
Your advantages

- high process reliability in long-chipping materials that tend to cause jamming
- cost-efficient for small and medium-sized batches
- high flexibility and short delivery times, because of comprehensive standard product range (diverse thread profiles, sizes and tolerances in stock)
- lower tool inventory because of use in ISO M and ISO P materials

Strong in high-tensile titanium



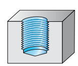
2 x D_N



P	M	K	N	S	H	O
				●●		

Prototex® TiNi Plus **Type:** 2021763

2 x D_N



P	M	K	N	S	H	O
				●●		

Paradur® Ti Plus **Type:** 2041663

The tool

– especially for machining ISO S materials with a geometry designed for **emulsion**
– very high flank clearance angle for reducing the friction in materials that tend to cause jamming
– designed for machining hard materials thanks to small rake angle
– wear-resistant, titanium-free ACN coating reduces weld formations

The application

– for applications in aerospace technology, as well as medical industry
– especially for high tensile and titanium alloys with a tensile strength from 700 to 1400 N/mm² that tend to cause jamming

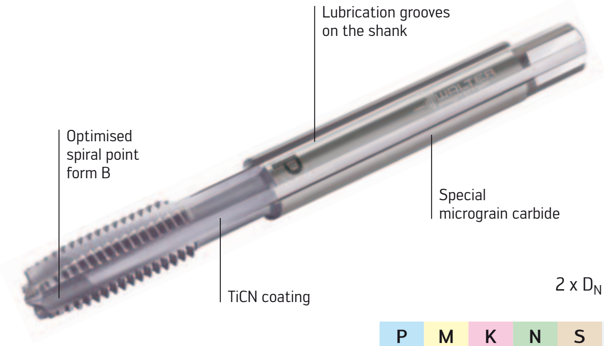
Prototex® TiNi Plus

– can also be used on nickel alloys

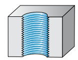
Your advantages

– often possible to work with emulsion instead of oil
– high process reliability through high tool stability
– long tool life through an innovative hard material coating and stable cutting edges
– excellent thread quality

Long tool life, extremely high speeds



2 x D_N



P	M	K	N	S	H	O
●●		●●				

IK via flutes on the shank*

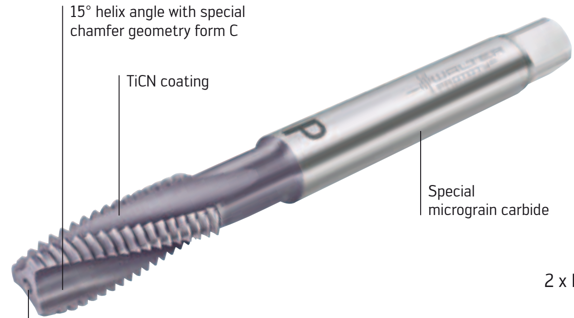
Prototex® HSC **Type:** 8021006

The tool
- special solid carbide with high resistance to wear and extreme toughness at the same time
- longer tool life through an increased number of flutes
- Shank tolerance h6 (e.g. for use in shrink-fit chucks)

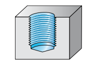
– The application
- ISO P: steels with a tensile strength from approx. 700 to 1400 N/mm²
- ISO K: primarily GJS (GGG) materials
- mass production with the goal of minimum costs per thread
- large-scale manufacturers focused on increasing productivity

Your advantages
- minimum production costs and extremely high productivity thanks to a cutting speed that is up to 3 times higher when compared to HSS-E taps
- optimum machine output due to longer tool life

Requirements:
- internal cooling
- stable application conditions
- modern machining centres or modern transfer lines
- for carbide tools, synchronous machining and the use of adaptors with minimum compensation (e.g. Protoflex C) is recommended (increases the tool life and increases process reliability)



2 x D_N



P	M	K	N	S	H	O
●●		●●			●●	

KA absolutely essential*

Paradur® HSC **Type:** 8041056

The tool
- special chamfer geometry and reduced helix for short broken chips also in long-chipping materials
- Shank tolerance h6 (e.g. for use in shrink-fit chucks)

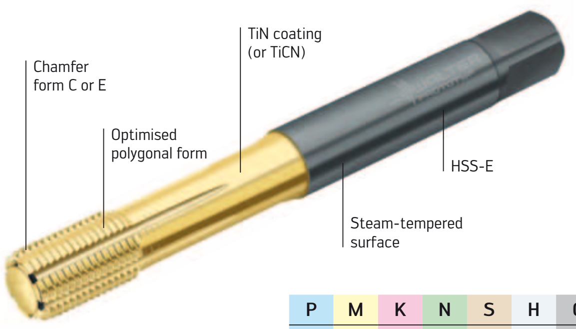
The application
- ISO P/H: steel materials from approx. 700 N/mm² to 55 HRC
- ISO K: cast iron workpieces such as: GGG40, GJV450, ADI800
- mass production with a focus on minimum costs per thread
- large-scale manufacturers focused on increasing productivity

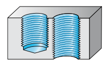
Your advantages
- minimum production costs and extremely high productivity thanks to a cutting speed that is up to 3 times higher when compared to HSS-E taps
- fewer tool changes resulting in optimum machine output due to long tool life
- high process reliability through perfect chip breaking

Requirements:
See Prototex® HSC on page 26

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

The high-tech thread former





	P	M	K	N	S	H	O	
Protodyn® S Eco Plus	●●	●●		●●	●			3,5 x D_N
Variants: without IK, with KR*								
Protodyn® Eco Plus	●●	●		●●	●			3 x D_N
Variants: without IK, with KA*								

Protodyn® S Eco Plus **Type:** EP2061745

The tool

- new type of TiN coating and additional steam treatment for extremely long tool life without cold welding
- innovative chamfer geometry ensures better running-in and wear behaviour
- special surface treatment and optimised polygonal form lead to longer tool life through reduced friction (important for MQL)
- versions with radial internal cooling for long thread depths in the standard product range

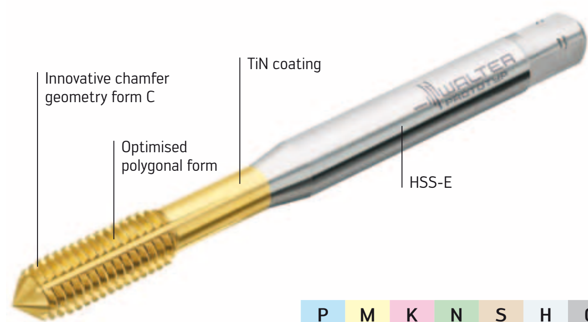
The application

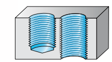
- universal high-performance thread former for use in all formable materials up to approx. 1200 N/mm²
- special variant with TiCN coating for machining carbon steels, as well as abrasive aluminium alloys

Your advantages

- fewer tool changes, optimum machine output and increased productivity through high forming speeds and long tool life
- reduced cooling lubricant costs due to the possibility for MQL machining
- higher performance compared to Protodyn® S Plus

Low tool costs, good performance





	P	M	K	N	S	H	O	
Protodyn® S Plus	●●	●●		●●	●			3,5 x D_N
Protodyn® Plus	●●	●		●●	●			3 x D_N

Protodyn® S Plus **Type:** DP2061705

The tool

- innovative chamfer geometry for better running-in and even wear behaviour
- optimised polygonal form for reduced friction and longer tool life

The application

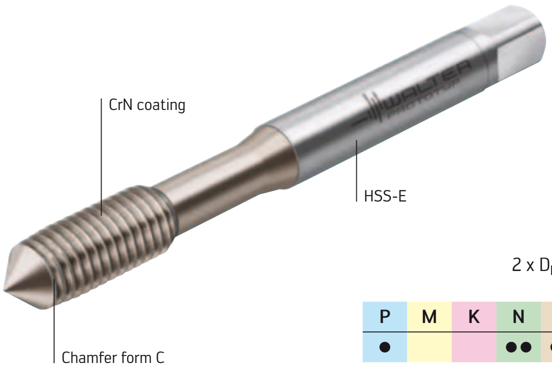
- for universal use in all formable materials up to approx. 1200 N/mm²

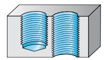
Your advantages

- lower purchase price (and lower performance) compared to Protodyn® S Eco Plus
- reduction in tool inventory, since it can be used universally in a broad material spectrum

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

Ideal solution for soft materials



2 x D_N 

P	M	K	N	S	H	O
●			●●	●●		

Protodyn® Eco LM **Type:** E2061604

The tool

– titanium-free CrN coating

Comment:
For threads > 2 x D_N, we recommend grinding lubrication grooves into the thread section, made possible by semi-standard modification services.

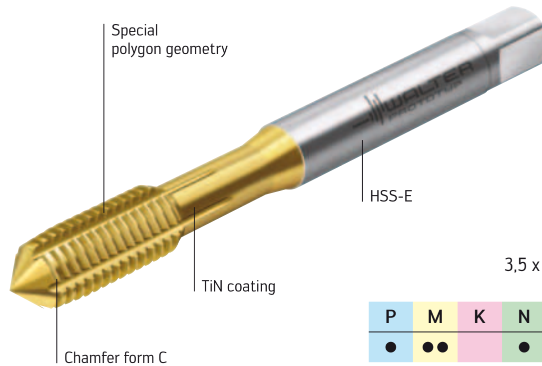
The application

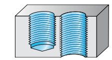
– for long-chipping, soft materials and for materials with a tendency to cause lubrication
– with a tensile strength from approx. 200 to 700 N/mm²
– ISO N: AlSi alloys with an Si content up to 12% and for long-chipping copper alloys
– ISO S: Ti alloys up to approx. 1100 N/mm² (if heavy duty oil is used)
– ideal under moderately good lubrication conditions in which TiN or TiCN has a tendency toward weld formations
– suitable for MQL

Your advantages

– increased process reliability and higher tool life due to a minimised tendency toward weld formations
– possible to machine wrought aluminium and cast alloys with **emulsion** instead of oil

The specialist for machining stainless steel



3,5 x D_N 

P	M	K	N	S	H	O
●	●●		●	●		

Protodyn® S Eco Inox **Type:** E2061305

The tool

– special polygon geometry makes it possible to machine stainless steels with **emulsion**

The application

– machining stainless steels with emulsion

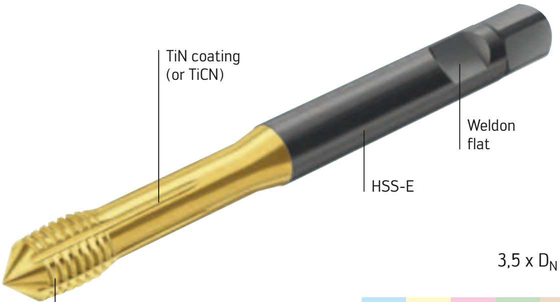
Comment:
With conventional thread formers, stainless steels can only be machined with oil. Machining centres, however, are generally operated with emulsion. To form threads, the machines would have to be stopped in order to manually lubricate the thread with oil . In addition to the increased machining time, there is the risk of the emulsion separating because of the foreign oil being added.

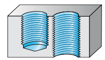
– can be used in all formable materials, however performance is lower compared to universal thread formers

Your advantages

– reduction in the machining time of stainless materials, because no manual intervention in the machining process is required
– the emulsion does not separate, because no foreign oil is used

Ideal for synchronous machining, universal use



3,5 x D_N 

P	M	K	N	S	H	O
●●	●●		●●	●		

Variants: without IK, with KR*

Protodyn® S Synchrospeed **Type:** S2061305

The tool

– the short thread section ensures reduced friction and high forming speeds
– variants with radial internal cooling for deep threads in the standard product range
– Shank tolerance h6 (e.g. for use in shrink-fit chucks)

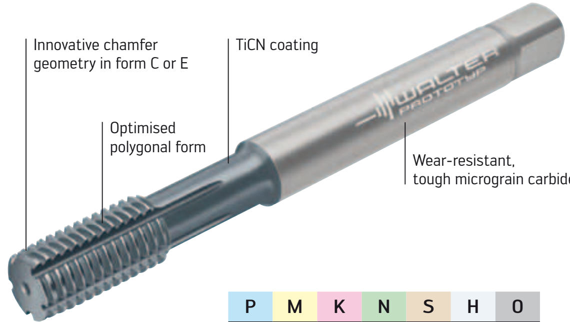
The application

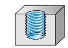
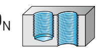
– for use on machine tools with a synchronous spindle; not suitable for floating chucks or cutting attachments
– for universal use in nearly all formable materials up to approx. 1200 N/mm²
– suitable for MQL
– it is generally recommended to use adaptors with minimum compensation (e.g. Protoflex C) (advantage: longer tool life and increased process reliability)

Your advantages

– high productivity due to high forming speeds
– reduction in inventory costs due to universal use
– possible to use simple adaptors without compensation mechanism

Long tool life, extremely high speeds



	P	M	K	N	S	H	O	
Protodyn® S HSC	●●	●		●●	●			4 x D_N 
	Variants: with KA*							
Protodyn® HSC	●●	●		●●	●			3 x D_N 
	Variants: without IK*							

Protodyn® S HSC **Type:** HP8061716

The tool

– optimised polygonal form reduces friction and increases tool life
– new type of chamfer geometry for uniform wear pattern
– h6 shank tolerance (e.g. for use in shrink-fit chucks)

Protodyn® S HSC:

– lubrication grooves and axial coolant supply for deep blind hole threads up to 4 x D_N

The application

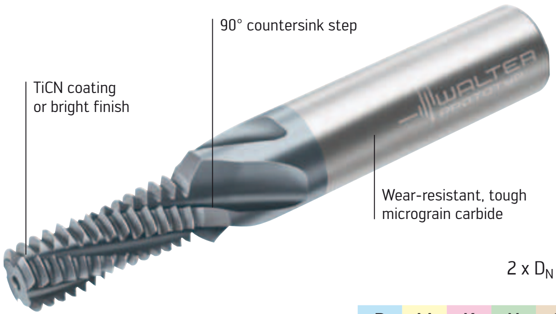
– ISO P: steel with a tensile strength up to 1200 N/mm²
– ISO M: stainless materials with a tensile strength up to 1000 N/mm² (preferably with oil)
– ISO N: AlSi alloys with an Si content up to 12% as well as Ni alloys with a tensile strength less than 900 N/mm²

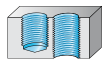
Your advantages

– extremely high productivity due to increased forming speeds
– fewer tool changes because of very long tool life
– attractive price/performance ratio on a mass-production scale
– best possible use of the drilling depth because the tool has no point

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

Universal with countersink step



2 x D_N 

P	M	K	N	S	H	O
●●	●●	●●	●●	●●		●

Variants: without IK, with KA (from size M4)*

Solid carbide thread mill TMC – Thread **M**ill **C**ountersink **Type:** H5055016

The tool

- solid carbide thread mill with countersink step
- concentricity < 10 µm for outstanding thread quality and long tool life

The application

- for universal use in a wide range of materials with a tensile strength up to approx. 1500 N/mm² and 48 HRC

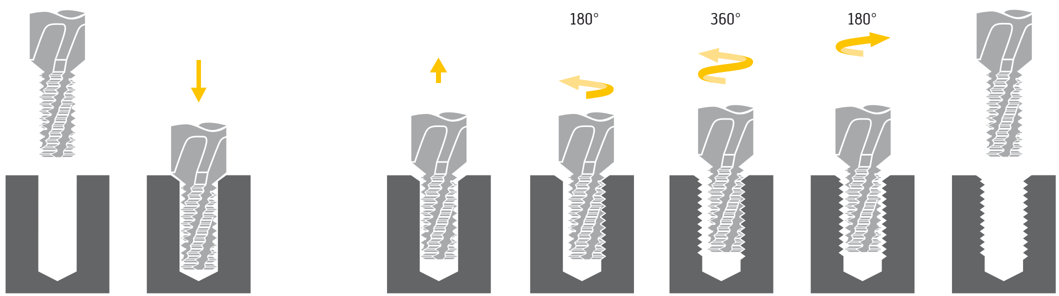
Your advantages

- long tool life and high cutting data because of improved substrate
- very good operational smoothness and soft cutting action because of optimised geometry

Comment:

If a countersink step is not required, use of thread mills from the **TMG** family is recommended. Their field of application is aligned with that of the TMC family. The TMC thread mills in the standard product range begin with size M3, the smallest size in the TMG family is M6.

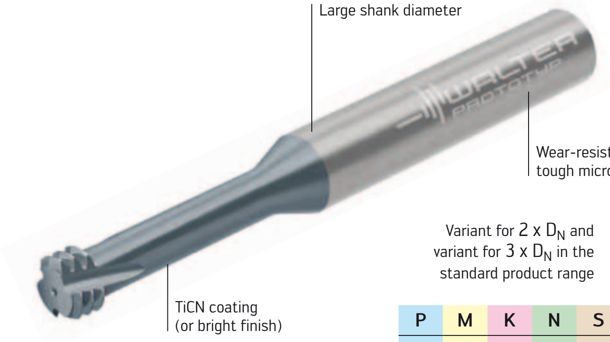
The strategy: TMC thread milling



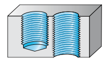
1. Positioning over core hole
2. Plunging and axial chamfering
3. Feeding back to required thread depth
4. Radial plunging into the thread 180°/¼ pitch
5. Creating thread using 360° helix
6. Exit loop 180° back to centre
7. Retracting the tool to the start position

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

Extremely high process reliability in the smallest of threads



Variant for 2 x D_N and variant for 3 x D_N in the standard product range



P	M	K	N	S	H	O
●●	●●	●●	●●	●●		●

Variants: without IK, with KA (from size M5)*

Thread mill TMO - Thread **M**ill **O**rbital **Type:** H5087016

The tool

- short cutting edge, smaller helix angle and positive rake angle for reduced forces and a soft cutting action
- larger shank diameter for vibration-free use, even with longer clamping lengths
- stable basic construction with large core diameter

The application

- for universal use in a broad material spectrum with a tensile strength up to 1500 N/mm² and 48 HRC
- excellent machining properties even for high-strength materials that tend to cause jamming (e.g. high-tensile stainless steels and Ti alloys)

Your advantages

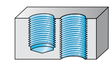
- long tool life because of innovative milling strategy
- small and deep threads (e.g. M1,6, 3 x D_N depth) can be produced reliably
- can be used profitably where conventional tools have reached their limits:
  - machining difficult-to-cut materials such as Inconel
  - producing deep threads
  - solution where (multiple) radial cutting passes would be necessary with conventional thread mills due to their conical threads

Comment:

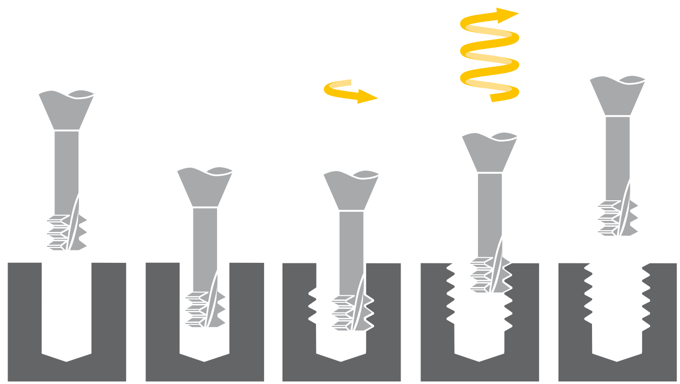
Orbital thread mills are available in the TMO HRC version. These tools are specially designed for machining hardened and high-strength materials.

Primary area of application: hardened steels up to 65 HRC, steels and alloyed steels from 1400 to 1600 N/mm²

P	M	K	N	S	H	O
●●				●	●●	●



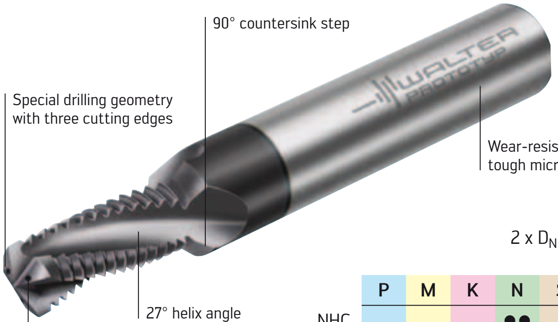
The strategy: TMO orbital-thread milling



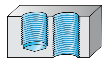
1. Positioning over core hole
2. Feeding in to thread depth
3. Radial plunging into the thread 180°/¼ pitch
4. Producing threads using helical interpolation
5. Retracting the tool to the start position

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

Drilling, countersinking and threading in one operation



2 x D_N



	P	M	K	N	S	H	O
NHC				●●			
TAX			●●				

IK absolutely essential*

Solid carbide thread milling cutter TMD - Thread **M**ill **D**rill **Type:** H5075018

The tool

- solid carbide thread milling cutter
- cutting length and countersink step matched to 2 x D_N thread depth
- TAX coating for ISO K materials
- NHC coating for ISO N materials

The application

- ISO K: cast iron workpieces such as GG25 (GGG materials can only be machined in exceptional cases. Machining these materials is made possible in part by a double-edged special tool.)
- ISO N: cast aluminium with an Si content of 7% and above; short-chipping Mg and Cu alloys
- Direct machining of precast core holes

Your advantages

- greater cost efficiency for less than 8 identical threads per component compared to conventional tools**
- increased productivity by shortening processing times by up to 50%
- space savings in the tool magazine
- exact positioning of core hole and thread

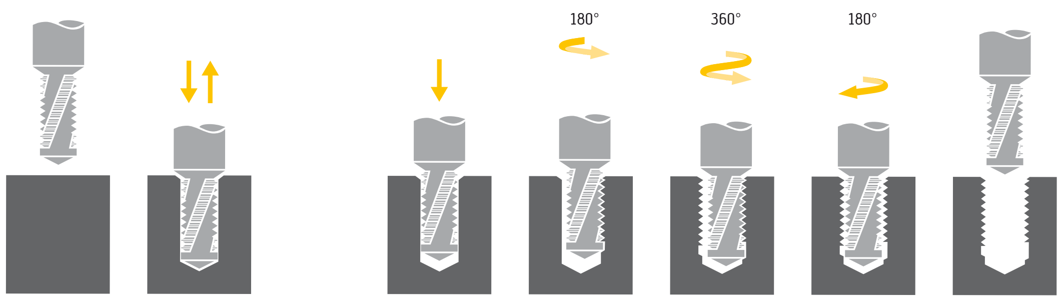
** the advantages can vary depending on the chip-to-chip time

Practical tip:

Use of TMD is practical if one single thread has a different specification to all of the other threads in the component.

Example: 13 threads per component. 12 of them are M8, 1 thread is M6. Instead of using a core-hole drill and a threading tool, this thread can be made more economically with the TMD.

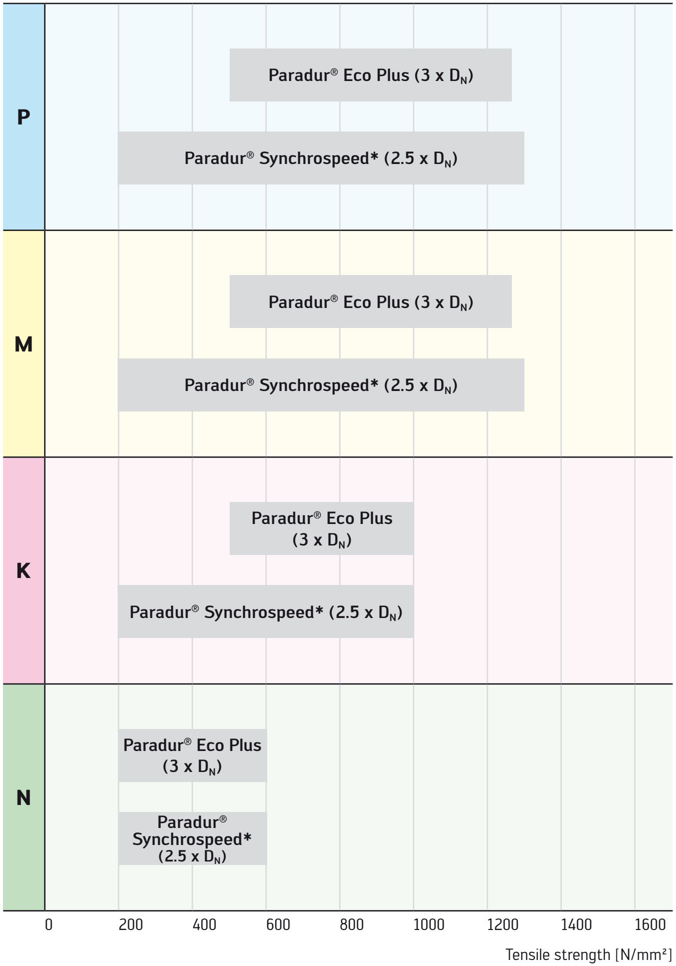
The strategy: TMD thread milling with countersink step



1. Positioning over core hole
2. Spot drilling, drilling, countersinking the core hole and chip removal
3. Moving to the start position of the thread milling cycle
4. Radial plunging into the thread 180°/¼ pitch
5. Creating thread in counter-rotation using 360° helix
6. Exit loop 180° back to centre
7. Retracting the tool to the start position

* IK = internal coolant supply
KA = internal coolant supply with axial coolant outlet
KR = internal coolant supply with radial coolant outlet

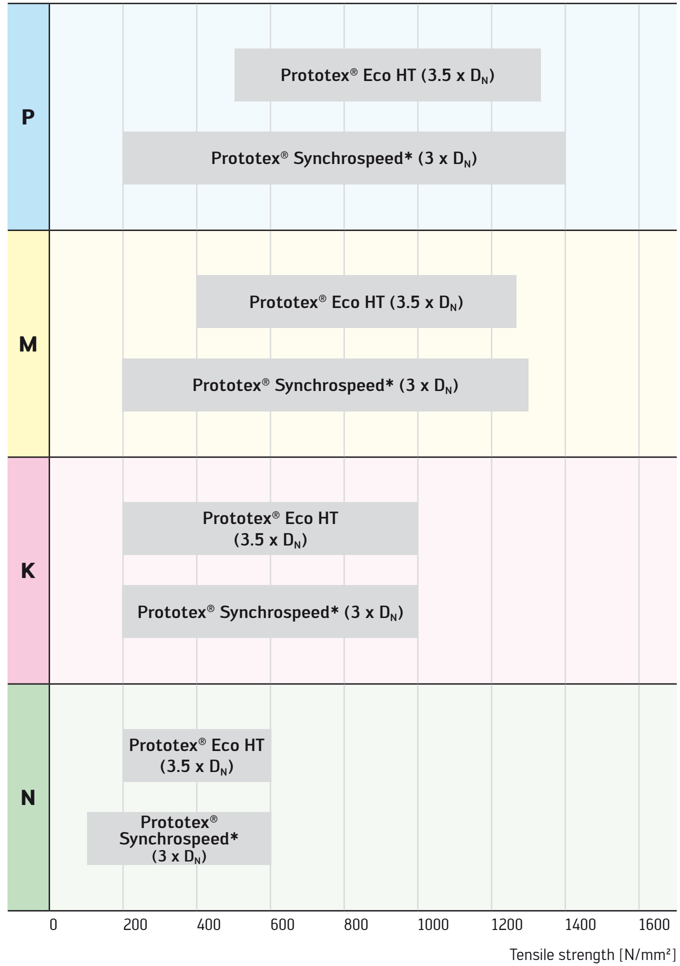
Universal blind hole taps



HSS-E or HSS-E-PM cutting tool material

* only for synchronous machining

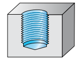
Universal through-hole taps

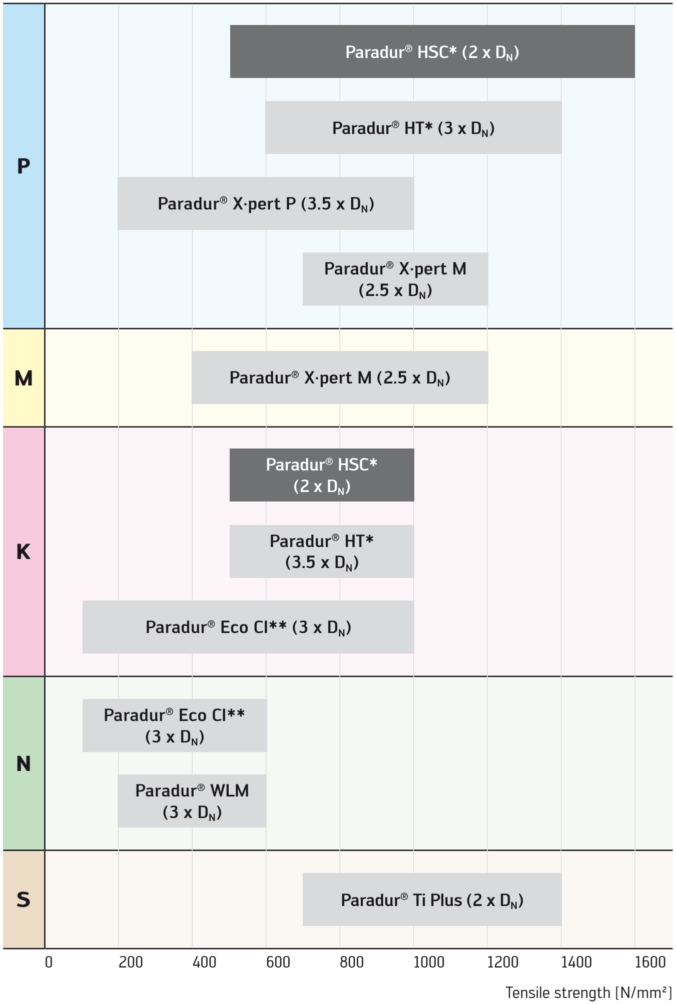


HSS-E or HSS-E-PM cutting tool material

* only for synchronous machining

Blind hole taps for special applications





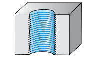
Solid carbide cutting tool material

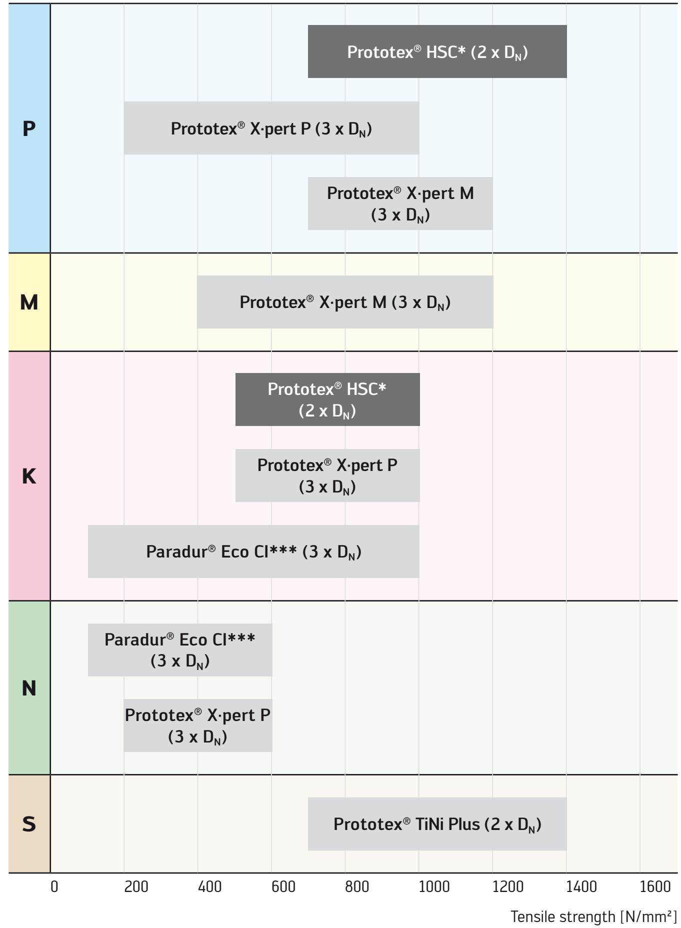
HSS-E or HSS-E-PM cutting tool material

* internal cooling required

** only for short-chipping materials; internal cooling recommended

Through-hole taps for special applications





Solid carbide cutting tool material

HSS-E or HSS-E-PM cutting tool material

* internal cooling required

*** only for short-chipping materials

Thread formers

●● Primary application
● Additional application

Material group	Structure of main material groups	Workpiece material	Brinell hardness HB		Tensile strength R_m N/mm^2						
	Thread depth					2,0 x D_N	3,5 x D_N				
	Type					Protodyn® Eco LM	Protodyn® S Plus	Protodyn® S Eco Plus	Protodyn® S Eco Inox	Protodyn® S Synchrospeed	Protodyn® S HSC
	Product information: Page					30	29	28	31	32	33
											
P	Unalloyed and low alloy steel	annealed (tempered)	210		700	●●	●●	●●	●	●●	●
		free cutting steel	220		750	●●	●●	●●	●	●●	●
		tempered	300		1010		●●	●●	●	●●	●●
		tempered	380		1280		●	●	●	●	●●
		tempered	430		1480						
	High-alloy steel and high-alloy tool steel	annealed	200		670		●●	●●	●	●●	●
		hardened and tempered	300		1010		●●	●●	●	●●	●●
		hardened and tempered	400		1360						
	Stainless steel	ferritic/martensitic, annealed	200		670		●●	●●	●●	●●	●●
		martensitic, tempered	330		1110		●●	●●	●●	●●	●●
M	Stainless steel	austenitic, duplex	230		780		●●	●●	●●	●●	●●
		austenitic, precipitation hardened (PH)	300		1010		●	●	●	●	●
K	Grey cast iron		245		–						
	Cast iron with spheroidal graphite	ferritic, pearlitic	365		–						
	GGV (CGI)		200		–						
N	Aluminium wrought alloys	not precipitation hardenable	30		–	●●	●●	●●	●	●●	●●
		precipitation hardenable, precipitation hardened	100		340	●●	●●	●●	●	●●	●●
	Cast aluminium alloys	≤ 12% Si	90		310	●●	●●	●●	●	●●	●●
		> 12% Si	130		450						
	Magnesium alloys		70		250						
	Copper and copper alloys (bronze/brass)	unalloyed, electrolytic copper	100		340	●●	●	●	●	●	●
		brass, bronze, red brass	90		310						
		Cu-alloys, short-chipping	110		380						
		high-strength, Ampco	300		1010						
S	Heat-resistant alloys	Fe-based	280		940						
		Ni or Co base	250		840		●●	●●	●	●●	●●
		Ni or Co base	350		1080						
	Titanium alloys	pure titanium	200		670	●●					
		α and β alloys, precipitation hardened	375		1260	●●					
		β alloys	410		1400	●●					
	Tungsten alloys		300		1010						
	Molybdenum alloys		300		1010						

Thread mills

●● Primary application
● Additional application

		Thread depth				**1,5 x D_N 2,0 x D_N**	**2,0 x D_N**			**2,0 x D_N 3,0 x D_N**
		Type				**TMG**	**TMC**	**TMO HRC**	**TMD**	**TMO**
		Product information: Page				35	34	37	38	36
Material group	**Structure of main material groups**	**Workpiece material**	Brinell hardness HB		Tensile strength R_m N/mm²					
P	Unalloyed and low alloy steel	annealed (tempered)	210		700	●●	●●			●●
		free cutting steel	220		750	●●	●●			●●
		tempered	300		1010	●●	●●			●●
		tempered	380		1280	●●	●●			●●
		tempered	430		1480	●●	●●	●●		●●
	High-alloy steel and high-alloy tool steel	annealed	200		670	●●	●●			●●
		hardened and tempered	300		1010	●●	●●			●●
		hardened and tempered	400		1360	●●	●●	●●		●●
	Stainless steel	ferritic/martensitic, annealed	200		670	●●	●●			●●
		martensitic, tempered	330		1110	●●	●●	●		●●
M	Stainless steel	austenitic, duplex	230		780	●●	●●			●●
		austenitic, precipitation hardened (PH)	300		1010	●●	●●			●●
K	Grey cast iron		245		–	●●	●●		●●	●●
	Cast iron with spheroidal graphite	ferritic, pearlitic	365		–	●●	●●		●●	●●
	GGV (CGI)		200		–	●●	●●		●●	●●
N	Aluminium wrought alloys	not precipitation hardenable	30		–	●●	●●		●●	●●
		precipitation hardenable, precipitation hardened	100		340	●●	●●		●●	●●
	Cast aluminium alloys	≤ 12% Si	90		310	●●	●●		●●	●●
		> 12% Si	130		450	●●	●●		●●	●●
	Magnesium alloys		70		250	●●	●●		●●	●●
	Copper and copper alloys (bronze/brass)	unalloyed, electrolytic copper	100		340	●●	●●		●●	●●
		brass, bronze, red brass	90		310	●●	●●		●●	●●
		Cu-alloys, short-chipping	110		380	●●	●●		●●	●●
		high-strength, Ampco	300		1010	●●	●●		●●	●●
S	Heat-resistant alloys	Fe-based	280		940	●●	●●			●●
		Ni or Co base	250		840	●●	●●			●●
		Ni or Co base	350		1080	●●	●●			●●
	Titanium alloys	pure titanium	200		670	●●	●●			●●
		α and β alloys, precipitation hardened	375		1260	●●	●●			●●
		β alloys	410		1400	●●	●●			●●
	Tungsten alloys		300		1010	●●	●●	●		●●
	Molybdenum alloys		300		1010	●●	●●	●		●●
H	Hardened steel		50 HRC		-			●●		
			55 HRC		-			●●		
			60 HRC		-			●●		

Comparison of the processes for producing threads

Thread tapping

Advantages

- no special requirements for the machine
- almost all machinable materials can be processed

Disadvantages

- chip removal is often challenging and requires tool diversity as well as special modifications (particularly with deep blind hole threads in long-chipping materials)
- reduced tool stability due to flutes; risk of fracture increases
- risk of workpiece having to be rejected if the tool breaks
- process may react sensitively to batch-related changes in the properties of the workpiece materials
- increased risk of machine stoppage due to bird nesting

Thread forming

Advantages

- high process reliability
  - no chips and therefore no problems with chip removal: even deep threads can therefore be produced reliably
  - low risk of fracture because of stable tools
- high thread quality
  - high static and dynamic strength of the thread because of cold work hardening
  - very good thread surface with minor roughness
- longer tool life compared to thread tapping
- tools can be used universally
- GL and DL threads with one tool

Disadvantages

- risk of workpiece having to be rejected if the tool breaks
- area of application limited due to elongation at fracture, tensile strength and thread pitch
- tighter tolerance of the core hole increases the production costs; profitability comparison with thread tapping absolutely essential
- not approved for use in the food industry, the medical industry and the aerospace industry

Thread milling

Advantages

- high flexibility
  - universal use of the tools in the most varied materials
  - one tool for blind-hole and through-hole threads
  - different thread dimensions (with the same pitch) can be produced with one tool
  - any tolerance grades can be produced with one tool
  - single and multi-start threads as well as right-hand and left-hand threads can be produced with one tool
- high process reliability
  - no risk of bird nesting
  - workpiece does not have to be rejected if the tool breaks
  - low torque even with large dimensions
  - inclined run-in and run-out are unproblematic
  - machining of thin-walled components is possible thanks to low cutting pressure
- low spindle stress due to a smooth sequence of movements
- very good thread surface

Disadvantages

- high tool costs compared to HSS-E taps and thread formers
- 3D CNC machine absolutely essential
- more complex programming
- in mass production, thread milling is often inferior to thread tapping and thread forming in terms of cost-effectiveness

	Process reliability	Machining speed	Universality/ flexibility	Tool life	Tool costs	Thread depth	Typical batch size
Thread tapping	–	+	–	–	–	+	low to very high
Thread forming	+	+	+	++	+	++	low to very high
Thread milling	++	–	++	+	+	–	low to medium

– Reference
\+ higher than reference
++ significantly higher than reference

Tolerance grades of taps and thread formers

The tolerance grade of the internal thread produced depends not only on the tool dimensions, but also on the material and the machining conditions. In some cases, it is better to choose tolerances that deviate from the standard. This toleration is identified by the X placed after the tolerance class (e.g. 6HX instead of 6H). Please note that these X grades vary from manufacturer to manufacturer, because they are based solely on company standards.

Taps, which are designed for tough materials, are produced by Walter Prototyp in X grades in order to counteract the resilient properties of the materials. At Walter Prototyp, this means increasing the dimensions for taps by half a tolerance grade. The X·pert M product range used for stainless steels is therefore designed in X grade. Taps for high-tensile titanium and nickel alloys are measured in X grade for the same reason.

If abrasive materials such as grey cast iron are being machined and miscutting is not a problem, then it also advisable to produce the tools in X grade. The tool life is increased due to the tolerance in X grade, because it takes longer for the tool to become heavily worn. For example, the Paradur® Eco CI tap is produced in this tolerance grade for precisely this reason.

Thread formers are produced in X grades, because the material rebounds stronger when forming threads than when cutting threads. The X grades for thread formers differ from those for taps. Nevertheless, this does not affect the tolerance of the female thread being produced, as can be seen in the table below.

The tolerance class of the tool (e.g. 4H) complies with the tolerance field of the female thread for which the tool has been designed. The table below shows that these tools can also be used to produce other tolerance fields.

Coatings that are subsequently applied to the female thread must be compensated for on the tap with a material removal calculation. The material removal can be calculated using the following formula:

$$A = T \times f \text{ where } f = \frac{2}{\sin \frac{\alpha}{2}}$$

A is the material removal to be calculated, T is the coating thickness of the subsequently applied coating and α is the flank angle.

Example:
metric thread, electroplated coating with a thickness of 25 µm

With a flank angle of 60°, this results in:

$$f = \frac{2}{\sin \frac{60°}{2}} = \frac{2}{0,5} = 4$$

from this it follows that

A = 0.025 mm x 4 = 0.1 mm

If a normal screw connection is to be achieved, a tool from tolerance class 6H + 0.1 must therefore be chosen.

Comment:
When thread milling, one tool can be used to produce any tolerance grades, because the tolerance grades are specified when programming.

Tolerance class of tool		Producible tolerance range of the female thread		Producible tolerance range of the female thread			Technical application
DIN designation for taps	Company standards for taps and thread formers						
ISO1/4H	4HX	4H	5H	–	–	–	screw connection with little clearance
ISO2/6H	6HX	4G	5G	6H	–	–	normal screw connection
ISO3/6G	6GX	–	–	6G	7H	8H	screw connection with a lot of clearance
7G	7GX	–	–	–	7G	8G	to prevent distortion during heat treatment

Coatings and surface treatments

	bright finish	vap	nid (nit + vap)	TiN	TiCN	THL
Primary areas of application	– very deep blind holes in soft steels – used if there are problems with chip removal	– primarily for stainless materials – in materials that are soft, tough and have a tendency toward weld formations – for very deep blind hole threads	– DL: Steel up to 1200 N/mm², cast iron and aluminium machining; – GL: only short-chipping materials (GG, AlSi alloys with Si content > 7%, C70); steels with high pearlite content; – not for stainless materials that tend to cause jamming	– low-alloy steels – stainless materials – suitable for Ni alloys	– alloyed and unalloyed steels – abrasive materials such as grey cast iron, AlSi (Si > 5%), Cu bronze alloys – universal layer for GFR up to 48 HRC – suitable for Ni alloys	– steels in general and stainless steels in particular – deep blind holes – MQL machining – GJS (GGG)
Features	– lower v_c/ shorter tool life compared to coated tools – more tightly rolled chips	– better coolant adhesion which reduces weld formations – lower v_c/shorter tool life than coated tools – better chip removal	– longer tool life because of increased surface hardness – increasing brittleness – nidamised means nitrided and vaporised	– universal layer – suitable for many materials – not for Ti alloys	– wear resistant to abrasive materials – highly suited to solid carbide tools – not for Ti alloys	– better chip formation than TiN and TiCN – tendency toward weld formations in materials containing manganese
Appearance						

	CrN	NHC	DLC	ACN	TAX	Diamond
Primary areas of application	– thread tapping in Al and Cu alloys – thread forming in Ti alloys – machining of ductile steels	– non-ferrous metals (Cu-, brass-, bronze- and Ti-alloys) – AlSi alloys with an Si content up to 12%	– Al alloys with a tendency to cause jamming	– Ti alloys – Ni alloys	– used universally for thread milling – also for hardened steels and HSC machining	– abrasive materials such as AlSi alloys with an Si content > 12%
Features	– reduces weld formations	– reduces built up edges – resistant to abrasive wear – sharp cutting edges are possible, because of the thin layer	– significant tool life increases are sometimes possible	– no affinity to titanium alloys, because it is a titanium-free layer	– high temperature resistance – universal layer	– resistant to abrasive wear
Appearance						

GL = blind hole machining
DL = through hole machining

Coatings and surface treatments

		Low to medium tensile strength								Medium to high tensile strength		Low to high tensile strength		Low to very high tensile strength
Material	P	X	X	X						X		X	X	X
	M		X	X						X		X	X	X
	K		X	X						X		X	X	X
	N	X	X	X	X	X		X	X			X		
	S				X						X			
	H											X		X
Surface treatment		bright finish	vap	TiN	CrN	NHC		DLC	Diamond	nid	ACN	TiCN	THL	TAX
Thread tapping		X	X	X	X			X		X	X	X	X	
Thread forming				X	X			X				X		
Thread milling						X		X	X		X	X		X
Thread mill drill						X								X

Selection of coatings for thread forming

Material	TiN	TiCN
Magnetic soft iron	●●	●
Structural steel	●●	●
Carbon steel	●	●●
Alloyed steel	●●	●
Tempered steel	●●	●
Stainless steel	●	●●
Austenitic	●	●●
Ferritic, martensitic, duplex	●	●●
Highly heat-resistant	●	●●
Unalloyed Al/Mg	●●	●
Al, alloyed Si < 0.5%	●	●●
Al, alloyed Si < 0.5% to 10%	●	●●
Al, alloyed Si > 10%	●	●●

●● Recommended ● Possible application

Cooling and lubrication

We usually talk about "coolant" when referring to this, although with thread cutting and thread forming in particular, lubrication is more important than cooling. There are the following different methods of coolant supply:

– external coolant supply
– external coolant supply via outlets parallel to the axis on the chuck
– "internal" coolant supply via flutes on the shank
– internal coolant supply (= **IK**) with axial coolant outlet (= **KA**)
– internal coolant supply with radial coolant outlet (= **KR**)

External coolant supply is the most common method and works in most cases. When machining blind hole threads vertically, the core hole fills with coolant (with the exception of very small tool diameters) and this facilitates the thread machining process.

When producing through-hole threads, the core hole is unable to be filled because during thread tapping the chips are transported in the feed direction and during thread forming no chips are created; nevertheless the coolant may still be able to penetrate right to the chamfer in deep threads. The coolant flow should be set as close and parallel as possible to the tool axis.

Supplying the coolant externally becomes difficult when deeper threads are being machined with the spindle in a horizontal position. The coolant cannot penetrate right to the cutting edge in this case. The removal of chips also hinders the supply of coolant during blind hole tapping.

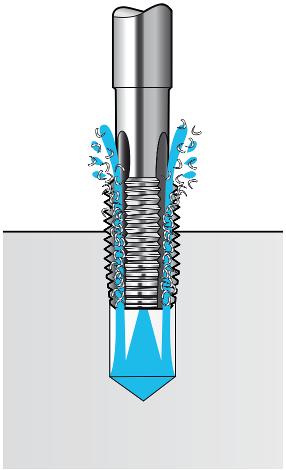
The supply of coolant parallel to the axis via cooling grooves in the shank has significant advantages, because the coolant is always reliably supplied to the cutting edge regardless of the tool length. It must only be noted that as the rotation speeds increases, the coolant is flung away radially if the coolant pressure is too low.

The internal coolant supply ensures that the coolant reaches the cutting edge at all times. Optimum cooling and lubrication of the cutting edge is always guaranteed and in many cases aids chip removal.

Material group	Material	Thread cutting	Thread forming	Thread milling
P	Steel	Emulsion 5%	Emulsion 5 - 10%	Emulsion/MQL/ air blast
P	Steel 850 - 1,200 N/mm²	Emulsion 5 - 10%	Emulsion 10% or oil (Protofluid)	Emulsion/MQL/ air blast
P	Steel 1,200 - 1,400 N/mm²	Emulsion 10% or oil (Protofluid)	Emulsion 10% or oil (Protofluid or Hardcut 525)	Emulsion/MQL/ air blast
P	Steel 1,400 - 1,600 N/mm² equivalent to 44 - 49 HRC	Oil (Protofluid or Hardcut 525)	Forming generally not possible	Emulsion/MQL/ air blast
M	Stainless steel	Emulsion 5 - 10% or oil (Protofluid)	Oil (Protofluid) [emulsion 5-10% only possible with special tools (Protodyn® S Eco Inox)]	Emulsion
K	Grey cast iron GG	Emulsion 5%	Forming not possible	Emulsion/MQL/ air blast
K	Ductile cast iron GGG	Emulsion 5%	Emulsion 10%	Emulsion/MQL/ air blast
N	Aluminium up to max. 12% Si	Emulsion 5 - 10%	Emulsion 5 - 15%	Emulsion/MQL/ air blast
N	Aluminium over 12% Si	Emulsion 5 - 10%	Emulsion 5 - 10% Forming only practical in exceptional cases	Emulsion/MQL/ air blast
N	Magnesium	Oil (Protofluid)	Forming not possible at room temperature	Dry
N	Copper	Emulsion 5 - 10%	Emulsion 5 - 10%	Emulsion/MQL/ air blast
S	Titanium alloys	Emulsion 10% or oil (Protofluid or Hardcut 525)	Oil (Hardcut 525)	Emulsion
S	Nickel alloys	Emulsion 10% or oil (Protofluid or Hardcut 525)	Oil (Protofluid or Hardcut 525)	Emulsion
H	Steel >49 HRC	Oil (Hardcut 525) possible only with carbide tools	Forming not possible	Dry/MQL
O	Synthetics	Emulsion 5%	Forming does not produce dimensionally accurate threads	Emulsion/MQL

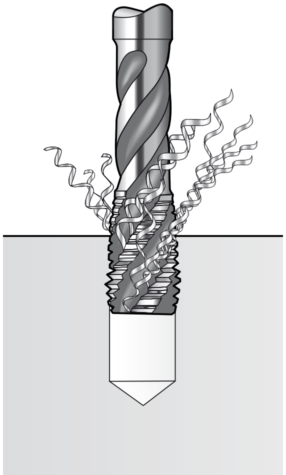
Cooling and lubrication – Thread tapping

There are two cases which apply to **blind hole tapping**:



Case 1: Short chips
The best results in terms of performance and process reliability are attained if the chips can be broken when short. These short chips can be easily flushed out of the threads using coolant. The best way to break the chips when short is with straight-fluted taps (e.g. Paradur® HT). The KA is recommended for blind hole threads.

> **Comment:**
> If blind hole threads are being produced in short-chipping materials without IK, the chips collect at the bottom of the hole. If the safety margin has been measured too tight, the tool runs up against the chips and may break.



Case 2: Long chips (chips cannot be broken)
With steels lower than 1000 N/mm² or with stainless steels and other very tough materials in general, it is normally not possible to break the chip when short. In these cases, the chip must be removed using helically fluted tools. If there is internal cooling, the coolant only helps with chip removal. In some cases, taps with a shallower helix can be used which increases the tool life.

Cooling and lubrication – Thread milling

Wet machining is generally recommended for **thread milling**, however it should only be applied if evenly distributed cooling can be guaranteed, otherwise the emerging thermal shocks lead to the formation of microcracks, which in turn result in fractures and this reduces the tool life. Wet machining with an externally supplied lubricant often means that evenly distributed cooling cannot be guaranteed. Dry machining with compressed air is generally possible when thread milling, however some tool life is lost.

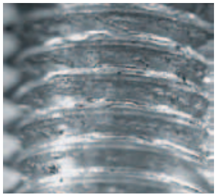
When blind hole machining, it is generally recommended to use a tool with an axial coolant outlet. The best option is to use emulsion. No thermal shocks occur because the tool is completely submerged. In addition, the flow of coolant aids chip removal and therefore ensures that the process is reliable. Alternatively, internally supplied compressed air or MQL can also be used here , however this results in a shorter tool life. The use of externally supplied emulsion when producing blind hole threads is not recommended, because chips may accumulate in the core hole and this has a negative effect on the tool life. Moreover, there is an increased risk of thermal shocks if externally supplied lubricant is used.

Externally supplied emulsion, MQL or compressed air is recommended for producing through hole threads. Wet machining may nevertheless lead to problems here, because externally supplied coolant cannot always guarantee an even cooling of the tool. With small thread dimensions in particular, there is a risk of the externally supplied coolant not being able to enter the narrow hole fully, with the result that even cooling of the tool cannot be guaranteed.

> **Comment:**
> When thread milling, having no cooling is less of a problem than intermittent cooling.

Cooling and lubrication – Thread forming

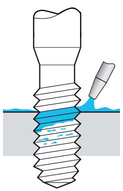
Cooling and lubrication in particular are of central importance when thread forming. Insufficient lubrication causes a sharp drop in the surface quality of the thread, as these photographs show:



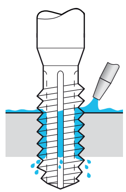
flaked surface from insufficient lubrication; Remedy: Lubrication grooves

smooth surface from excellent lubrication

There are basically two differing types of tools: **Thread formers with lubrication grooves and thread formers without lubrication grooves.** The different areas of application are explained below.



without lubrication grooves



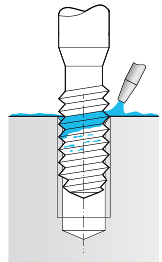
with lubrication grooves

The area of application for tools without lubrication grooves is limited to:

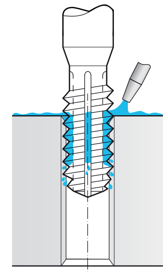
- steel sheet extrusions
- through hole threads up to 1.5 x D_N (because coolant cannot accumulate in the core hole)
- blind hole threads when machining vertically (KA is recommended for very deep blind hole threads)

Lubrication grooves ensure uniform lubrication even at the bottom of the thread which is why thread formers with lubrication grooves can be used universally. Vertical through hole threads up to approx. 3.5 x D_N can be produced with lubrication grooves even when internal cooling is not used.

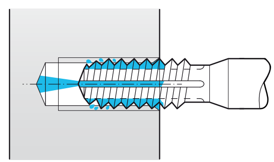
There are four different cases to consider for the tool design:



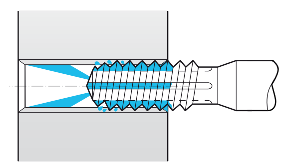
Vertical blind hole machining
Lubrication grooves and internal coolant supply are not required; external coolant supply is sufficient (KA is recommended for very deep threads).



Vertical through hole machining (> 1.5 x D_N)
Lubrication grooves are required; internal coolant supply is not necessary. Externally supplied coolant can penetrate into the thread profile through the lubrication grooves (KR is recommended for very deep threads).



Horizontal blind hole machining
Lubrication grooves and internal coolant supply are necessary. Axial coolant outlet is sufficient.



Horizontal through hole machining
Lubrication grooves are required. Internal coolant supply with radial outlet is recommended.

Minimum quantity lubrication

Coolant is used in machining operations to reduce tool wear, to dissipate heat from the workpiece and machine, and to aid chip breaking as well as chip removal. Moreover, the remnants of chips are removed from the workpiece, tool and the fixtures. All of these factors are important prerequisites for manufacturing in an efficient, trouble-free and cost-effective manner.

Nevertheless, the costs for procuring, maintaining and disposing of coolant continue to rise . The poor environmental compatibility of lubricants and the heath risks they represent for machine operators are under increasing scrutiny. As stated on page 7, the costs associated with lubricants amount to approx. 16% of the total production costs. Reducing the consumption of lubricants for economical and environmental reasons is therefore very important for successful companies who are working toward sustainability.

This plan can be achieved using Minimum Quantity Lubrication (MQL). With MQL machining, a small amount of highly effective lubricant is added to the compressed air. Even with very small doses of lubricant (approx. 5-50 ml/hr), weld formations on materials that tend to cause jamming can be prevented. In addition, MQL can be used to reduce friction which in turn reduces the process temperature.

In the most simple case, the lubricant is supplied externally. This method can be retrofitted inexpensively to existing machines, however the limit is reached with threads that have a depth greater than 1.5 x D_N. It is better to supply the lubricant through the spindle and this should be taken into consideration when purchasing machines.

The modified tool requirements for MQL must be taken into consideration when the tools are designed. For example, the tools must be designed so that as little heat as possible occurs during machining – small or even negative rake angles are therefore to be avoided. Similarly, the geometry is to be designed so that reliable chip removal can be achieved without the supportive effect of a lubricant. The coating in particular plays a central role in MQL machining, because the hard material layer takes on the lubrication function to a large extent. Furthermore, the coating reduces friction as well as insulating the tool against heat.

At thread depths > 1.5 x D_N, the prerequisite for MQL is an internal coolant supply with a radial outlet. Furthermore, the coolant channels in the tool must be designed so that the oil-air mixture does not become separated.

For MQL machining, Walter Prototyp recommends the specially developed THL coating for taps. This coating is available as standard for Paradur® Eco Plus (successor to the proven Paradur® Eco HT), Prototex® Eco HT as well as for Paradur® and Prototex® Synchrospeed tools. The THL coating has a lubricant layer which ensures very good friction behaviour even with MQL and also prevents build-up on the cutting edges. The layer is continuously polished during the course of the tool's life.

The Protodyn® Eco Plus, Eco LM and Synchrospeed families are suitable for minimum quantity lubrication when thread forming.

Your advantages
from MQL machining with Walter Prototyp tools:

- reduction in production costs and an increase in competitiveness
- reduction in lubricant, maintenance and disposal costs
- reduction in energy costs
- prevention of health risks for employees
- often no compromise in performance compared to wet machining
- trough-like components do not fill with lubricant
- less effort required for cleaning components

Comment:
In contrast to thread tapping and thread forming, dry machining is generally possible with thread milling, however some loss of tool life has to be accepted. If working dry, the use of an air blast is recommended for chip evacuation. When thread milling, it is often better to use MQL instead of wet machining, because the tool is not subject to thermal shocks.

Materials that are suitable for MQL machining	Materials that are not suitable for MQL machining
– non or low alloyed steels as well as cast steel < 1000 N/mm² – grey cast iron – brass – AlSi alloys – copper alloys	– high-tensile, high-alloy steels – Ti and Ni alloys – stainless steels

Notes:
- High-tensile and hardened materials can be machined with MQL during thread milling.
- In practice, there may be cases where the above-mentioned classification does not apply.

Clamping devices

Tapping chucks (also called tool adaptors) are the connecting piece between the spindle and the tool.

Tasks of the tool adaptor during thread tapping and thread forming:

- transmitting torque
- axial and/or radial compensation of differences between the spindle position and tool target position, where required

Tasks of the tool adaptor during thread milling:

- transmitting torque
- minimising the deflection of the tool (chuck must be rigid to oppose radial forces)
- damping vibration

General tasks:

- transferring the lubricant from the spindle to the tool
- protecting the spindle bearings if the tool breaks
- protecting the tool against breakage (can only be achieved to a limited extent)

In terms of the interaction between the spindle and the feed rate, it is crucial when thread tapping and thread forming to know if the spindle rotation speeds and the feed rate are matched to each other (synchronised) and their relative accuracy.

Comment:
All current milling chucks can be used for thread milling. The special chucks for thread tapping and thread forming are shown below.

Important types of tool adaptors for taps and thread formers

Quick change chuck with axial compensation

Advantages:

- for use in synchronous and non-synchronous machines
- compensation of axial and radial position deviations
- solid design

Disadvantages:

- more complicated technology than fixed chucks
- miscutting cannot be prevented, because the tool must guide itself

Quick change chucks are available in the standard product range from Walter.

Synchro chuck with minimum compensation

Advantages:

- compensation of axial forces resulting in a marked increase in the tool life
- combination of advantages from both fixed chucks and floating chucks

Disadvantages:

- more expensive to purchase than fixed chucks
- only for use on synchronous machine tools

Synchro chucks with minimum compensation are available in the standard product range from Walter.

Important types of tool adaptors for taps and thread formers

Tapping attachment

Advantages:
- for use in synchronous and non-synchronous machines
- protects the spindle, because the direction of rotation of the chuck can be reversed
- very short cycle times, because the spindle does not need to be accelerated or decelerated; for this reason it is of particular interest for mass production

Disadvantages:
- complicated technology
- high maintenance costs
- torque support required
- high procurement costs

Shrink-fit chucks, fixed collet chucks, Weldon chucks (from left to right)

Advantages:
- simple, cost-effective and solid design
- shrink-fit chuck: very high concentricity

Disadvantages:
- only for use on synchronous machine tools
- minimum pitch differences cause axial forces which act on the tool flanks and reduce the tool life

Shrink-fit chucks, collet chucks and Weldon chucks are in the standard product range from Walter.

Synchronous machining for tapping and forming threads

To reduce the process times in thread tapping and thread forming, manufacturers are increasingly favouring higher rotation speeds and cutting speeds (HSC). The synchronous machining approach is recommended especially for achieving high cutting speeds.

Synchronous tapping requires a machine that can synchronise the rotary motion of the main spindle with the feed motion. The threading tool does not guide itself using its geometry, but is controlled solely by the feed rate and the spindle rotation speed of the machine. Nowadays, most machining centres are suitable for synchronous machining.

Basically, all taps and thread formers can be used synchronously. Nevertheless, the tool range from Walter Prototyp known as Synchrospeed has been designed specifically for synchronous machining. The key characteristics of these tools are their extremely high flank clearance angle, as well as their extra short threading section. Tools in the Synchrospeed family can only be used synchronously. In contrast, the tools in the Eco family achieve very good results both synchronously and conventionally.

Synchronous taps are compatible with conventional Weldon chucks as well as collet chucks (where possible with square drive). Both fixtures have the disadvantage of being unable to compensate for the axial forces that are generated.

A better alternative is the Protoflex C tapping chuck with minimum compensation. Protoflex C is a tapping chuck for machining centres with synchronous control logic. It guarantees a precisely defined minimum compensation and is matched to the geometry of Synchrospeed tools.

What is so special about Protoflex C?
Unlike conventional synchro tapping chucks, the Protoflex C design is based on a precision-machined flexor with high spring rate, which compensates for position deviations in the micron range both radially and axially. The patented microcompensator is made from a special alloy originally developed for NASA and is characterised by a long service life and is maintenance-free. Conventional synchro chucks use plastic parts for this purpose, but these lose their flexibility over time,. Microcompensation is then no longer provided.

The Protoflex C tapping chuck helps to considerably reduce the pressure forces that act on the flanks of the tap. This results in:

- greater process reliability thanks to the reduced risk of breakage, particularly where dimensions are small
- a longer tool life due to less friction
- improved surface quality on the flanks of the thread

For customers using the Protoflex C tapping chuck, this means extremely high productivity while simultaneously reducing the tool costs, and this is true for both thread tapping and thread forming.

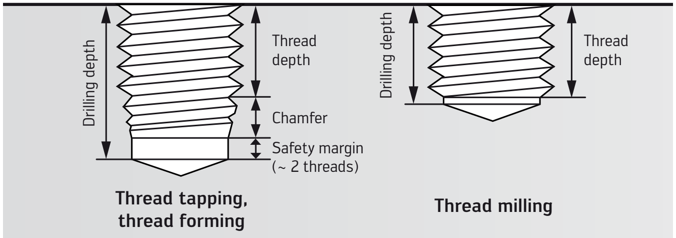
Protoflex C synchronous tapping chuck

Flexor with minimum compensation

Notes on the core hole

Depth of the core hole

Drilling depth ≥ usable thread depth (+ chamfer length) + safety margin



Comment:
Any existing tip on the threading tool must be taken into account when calculating the required depth of the core hole. Here a distinction must be made between a full point and a reduced point. In contrast to taps and thread formers, thread mills have neither a chamfer area or a tip, which makes it possible to have threads that almost go to the bottom of the hole. Miscutting is excluded from the milling process which is why an additional axial safety margin is not necessary.

Diameter of the core hole for thread tapping and thread milling

Rule of thumb:
Hole diameter = nominal diameter - pitch

Example size M10
Hole diameter = 10.0 mm – 1.5 mm = **8.5 mm**

Diameter of the core hole for thread forming

Rule of thumb:
Hole diameter = nominal diameter – f x pitch
– tolerance 6H: f = 0.45
– tolerance 6G: f = 0.42

Example size M10
Hole diameter = 10.0 mm – (0.45 x 1.5 mm) = 9.325 mm = **9.33 mm**

Special notes on thread forming

Comment:
The recommended diameter of the core hole is marked on the shank of Walter Prototyp thread formers.

When selecting the drilling and boring tool, the permissible tolerances for the core hole listed in the table below must also be noted to ensure a reliable forming process and a suitable tool life.

Thread pitch	Tolerance of pilot drill diameter
≤ 0.3 mm	± 0.01 mm
> 0.3 mm to < 0.5 mm	± 0.02 mm
≥ 0.5 mm to < 1 mm	± 0.03 mm
≥ 1 mm	± 0.05 mm

Based on these tolerances which in contrast to thread cutting are tighter, thread forming is not always more economical than thread tapping.

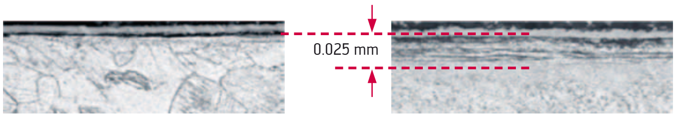
Practical tip:
In thread forming, the core diameter of the thread is created during the forming process and is therefore dependent on the flow characteristics of the material. In contrast to this, the core diameter for thread tapping and thread milling is already determined by the core hole. After the forming process, it is therefore absolutely essential to gauge the thread core diameter. The tolerances of the internal thread core diameter are listed on page 116.

Comment:
The product range from Walter Titex is aligned with the pilot hole diameters for tapping and forming threads.

Increased edge zone hardening

Often the production of threads is seen as a stand-alone process. This is not advisable because the preceding drilling operation has a significant impact on the subsequent threading process.

When the core hole is drilled, the edge zone of the workpiece is effected mechanically and thermally. The resulting structural changes can be seen in the two photomicrographs:



New drill:
edge zone is nearly unchanged

Worn drill:
influence of the edge zone

The hardness of the edge zone is significantly greater using a worn drill than using a new tool. Using higher cutting parameters when drilling leads to increased hardness of the edge zone. Even though the increased hardness only occurs within a very small distance to the hole surface, this causes a significant reduction in the tool life of the threading tool (compare the example below).

Summary:

- The tool life of the threading tool is reduced as the hardness of the edge zone increases.
- The hardness of the edge zone escalates as wear on the drilling or boring tool increases. High cutting parameters or rounded cutting edges also have an effect on the hardness of the edge zone.

Example: Material C70, tool diameter 8.5 mm, drilling depth 24.5 mm

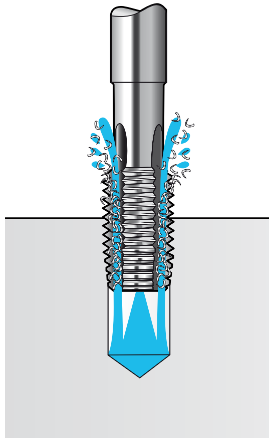
	Worn drill	New drill
Edge zone hardness	450 HV	280 HV
Edge zone width	0.065 mm	≈ 0
Tool life of tap	70 threads	> 350 threads

Practical tip:
If problems occur with the tool life, in addition to considering the process used to produce the threads, give consideration to the preceding drilling process and the drilling or boring tool itself.

Basic types

Blind hole

Short-chipping materials
Straight-fluted taps do not transport chips. For this reason, they can only be used with short-chipping materials or short threads.

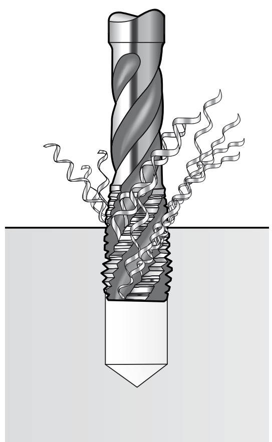


> **Comment:**
> The chips accumulate at the bottom of the hole if internal cooling is not used. If the safety margin has been measured too tight, the tool may run up against the chips and break.

Deep threads are possible with straight-fluted tools if the tap has an axial coolant supply, because the chips are flushed out against the feed direction. A prerequisite for this is that the chips are broken off short (e.g.: Paradur® HT, thread depth up to 3.5 x D_N).

In comparison to helical tools, straight-fluted taps have a longer tool life.

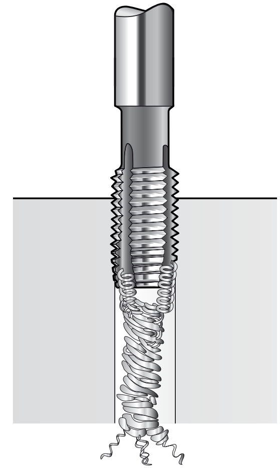
Some straight-fluted tools can also be used for through holes in materials with good chip breaking properties (e.g. Paradur® Eco CI).



Long-chipping materials

Right-hand spiral taps transport chips back towards the shank. The tougher the material to be machined is (producing longer chips) and the deeper the thread, the greater the helix angle required.

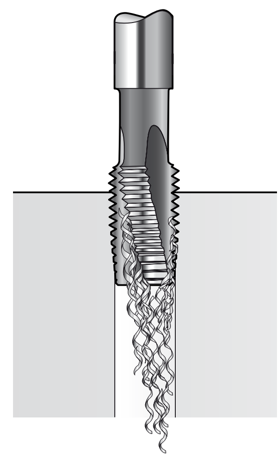
Through hole



Long-chipping materials

Taps with a spiral point transport the chips forward in the feed direction.

Taps with a spiral point are the first choice when producing through hole threads in long-chipping materials.

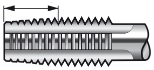
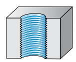
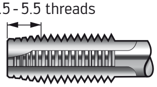
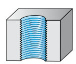
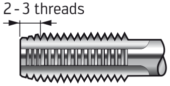
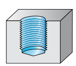
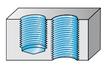
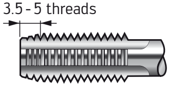
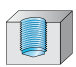
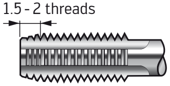
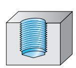
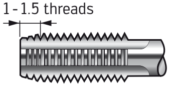
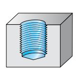


Left-hand spiral taps
(and taps with a spiral point) transport the chips forward in the feed direction.

Tools with left-hand spiral are practical only if chips cannot be removed reliably with a spiral point. Tool example: Paradur® N of the type 20411 and 20461

Chamfer forms based on DIN 2197

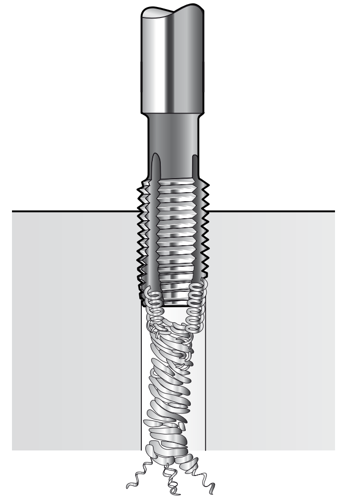
Please note:
- longer chamfers increase the tool life
- longer chamfers reduce the cutting edge load which gains importance as the material strength increases
- short chamfers enable threads to almost reach the bottom of the hole
- longer chamfers increase the required torque

Form	Threads per chamfer	Execution and application		
A	6 - 8 threads 	straight-fluted		short-chipping materials
				short through hole thread in medium and long-chipping materials
B	3.5 - 5.5 threads 	straight-fluted with a spiral point		medium and long-chipping materials
C	2 - 3 threads 	right-hand helical		medium and long-chipping materials
		straight-fluted		short-chipping materials
D	3.5 - 5 threads 	left-hand helical		long-chipping materials
		straight-fluted		short-chipping materials
E	1.5 - 2 threads 	right-hand helical		short thread run-out in medium and long-chipping materials
		straight-fluted		short thread run-out in short-chipping materials
F	1 - 1.5 threads 	right-hand helical		very short thread run-out in medium and long-chipping materials
		straight-fluted		very short thread run-out in short-chipping materials

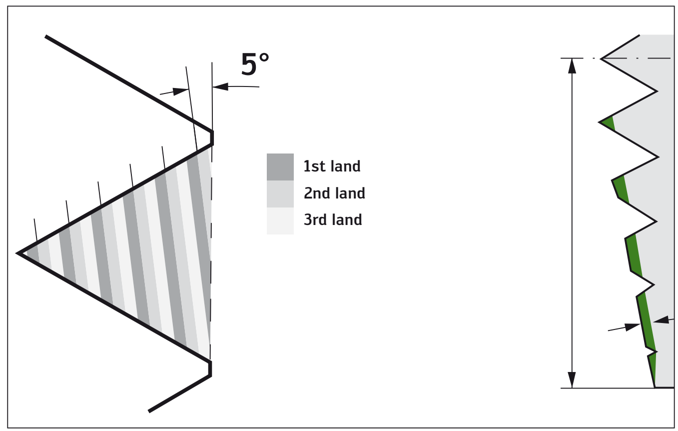
Chip cross sections

For through hole threads, usually longer chamfer forms are used.

Long chamfer (e.g. form B) results in:
- longer tool life
- high torque
- small chip cross-section
- low strain on the chamfer teeth



Form B

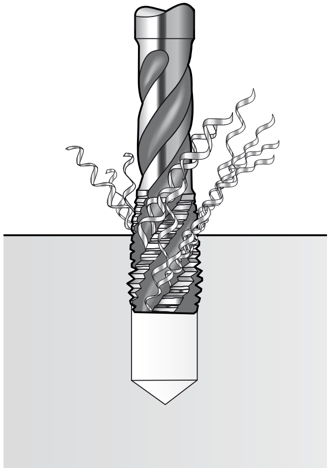


Chip cross sections

For blind hole threads, shorter chamfer forms are usually selected. This is justified not only by the fact that the thread should often reach the bottom of the hole.

The shearing of the chip in the blind hole thread presents a particular problem. If the chip becomes too thin, it simply flattens during reverse action and can no longer be cut through. The chip becomes trapped between the component and chamfer flank face. This may break the tool and this is why long chamfers in form A, B and D are not suitable for blind hole threads, as these forms produce thin chips.

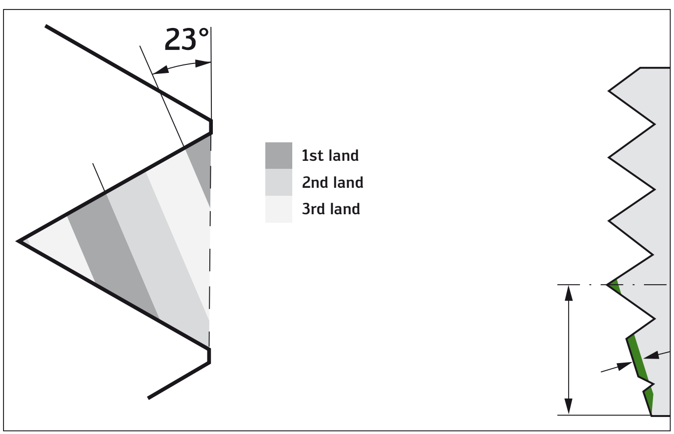
An advantage of short chamfers is that fewer chips are produced. In addition, the larger chip cross section is favourable for chip transport.



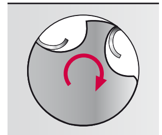
Short chamfer (e.g. form E) results in:

– low torque
– large chip cross section
– increased strain on the chamfer teeth
– shorter tool life
– optimised chip transport

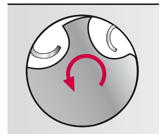
Form E



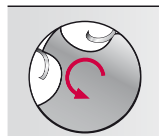
Cutting process for blind hole threads



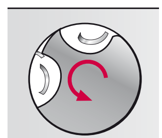
The tap has been cutting and now comes to a stop. At this very moment, all cutting edges in the chamfer are still in the process of forming a chip.



The tool begins to reverse. The chips remain where they are for the time being. The reverse torque at this point is virtually zero.



The chips come into contact with the back of the trailing land of the tap. The reverse torque now increases sharply. The chip has to be shorn off. As the chamfer of the tap has a clearance angle and withdraws from the thread axially when it backs out of the hole, it is inevitable that the purchase point will no longer be directly at the root of the chip. For this reason, the chip would require a certain amount of stability (thickness) to be cut.



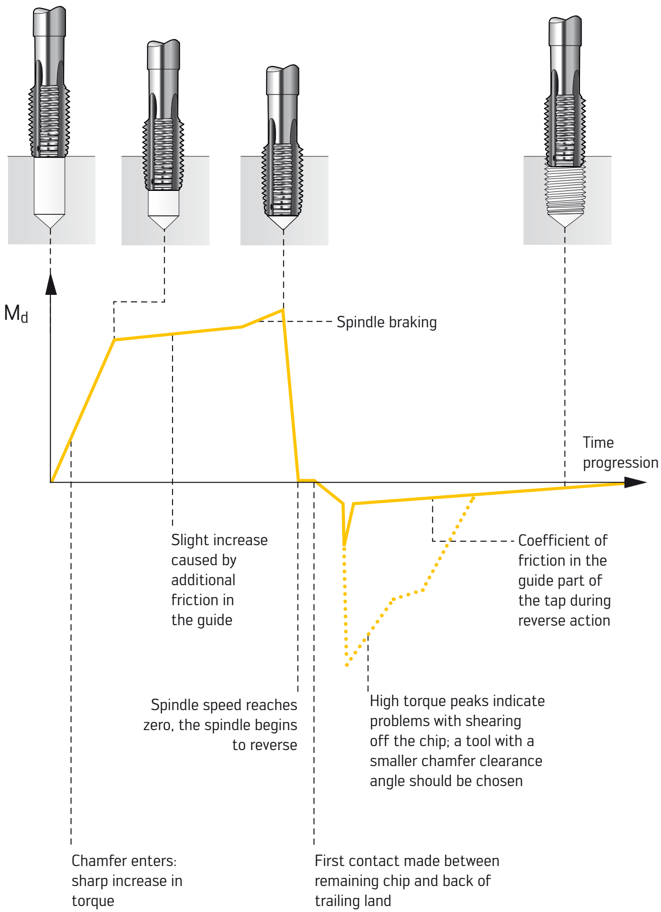
The chip has been shorn off and reverse torque decreases to the friction between the guide and the cut thread.

Comment:
Through hole taps cannot be used for blind hole machining, because these have a higher chamfer clearance angle and the chip might not be sheared off, but instead get jammed between the chamfer and the thread. This could lead to spalling on the chamfer and, in extreme cases, tap breakage.

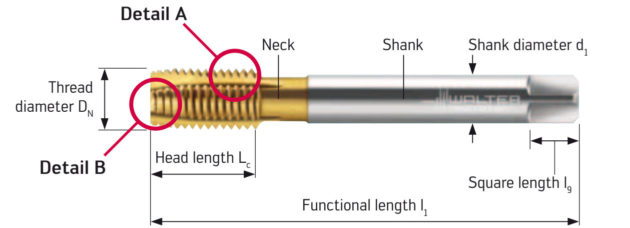
The chamfer clearance angle of blind hole taps is always smaller than that of through hole taps, because blind hole taps must shear off the chip root during reverse action.

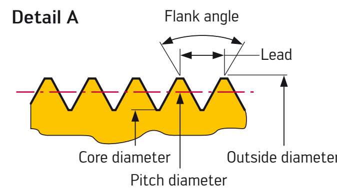
Cutting process for blind hole threads

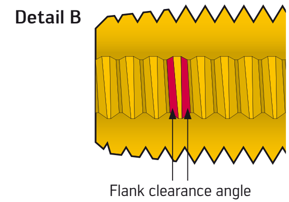
Torque curve during the blind hole thread tapping process

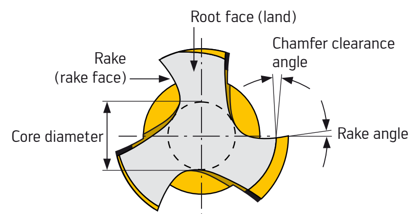


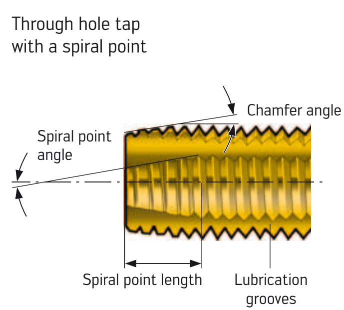
Angles and characteristics on the tap

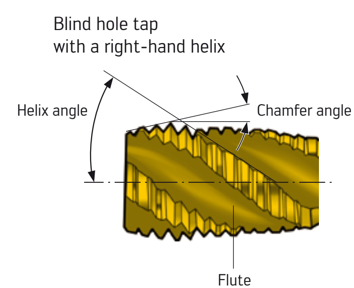








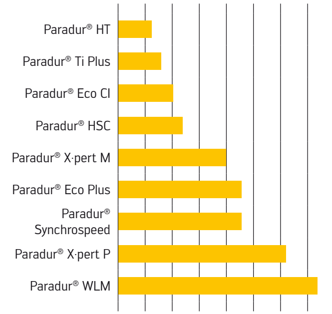




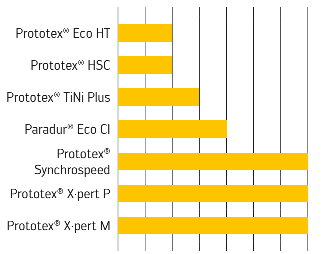
Comparison of geometry data

A smaller **rake angle:**
- increases the stability of the cutting edges (fractures around the chamfer may occur with large rake angles)
- normally produces chips in a more controlled manner
- produces poorer surfaces on the component
- increases the cutting forces and the cutting torque
- is required for machining harder materials
- increases the tendency to compress the material to be machined, i.e. the tap cuts less cleanly and therefore produces slightly tighter threads

Rake angle of blind hole tools

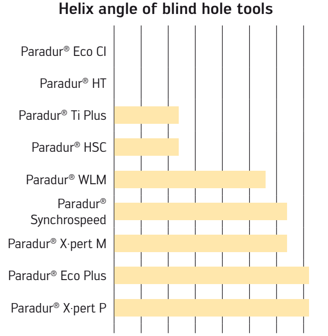


Rake angle of through hole tools



A larger **helix angle:**
- supports chip removal
- reduces the stability of the tool and this limits the maximum cutting torque
- reduces the stability of the teeth
- reduces the tool life

Helix angle of blind hole tools

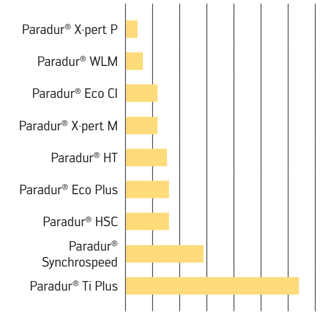


Flank clearance angle:

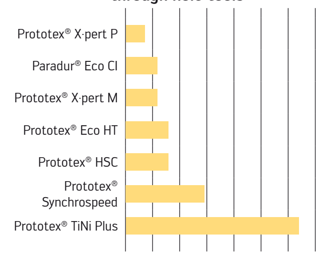
The flank clearance angle must be matched to the material to be machined. Materials with a higher tensile strength and materials that tend to cause jamming require a larger flank clearance angle. The guidance characteristics of the tool worsen as the clearance angle is increased, which is why miscutting occurs in soft materials if floating chucks are used.

> **Practical tip:**
> **Check the flank clearance angle**
> A tap should screw easily into the previously-cut thread without any recutting. If this is not possible, a tool type with a larger flank clearance angle should be selected.

Flank clearance angle of blind hole tools



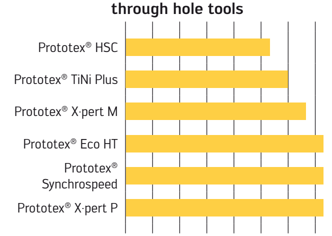
Flank clearance angle of through hole tools



Spiral point angle:

The spiral point angle is limited by the chamfer length and number of flutes, because with a larger spiral point angle, the land width in the first thread of the chamfer is reduced. This causes the stability of the cutting edge to decrease (the risk of fractures around the chamfer increases). An increased spiral point angle facilitates chip removal in the feed direction. If the spiral point angle is too small, chip removal becomes problematic. Left-hand helical tools provide a solution for this.

Spiral point angle of through hole tools



Chamfer clearance angle:

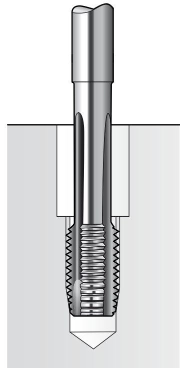
Through hole taps have approx. 3-times as large a chamfer clearance angle as blind hole taps. See page 80 for the reason for this.

Special features of thread tapping

Recessed and deep blind hole threads

– where possible use straight-fluted taps with axial coolant supply or blind hole taps with a steep helix angle and a bright or vaporised rake:
  - Paradur® HT (straight-fluted)
  - Paradur® Synchrospeed with Tin/vap coating (helical)

– for stainless steels and in general we recommend thread forming as a problem solver; spiral taps are absolutely essential for tapping threads in stainless steels:
  - Thread forming: Protodyn® S Eco Inox
  - Thread tapping: Paradur® X·pert M



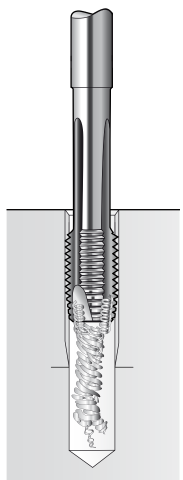
Threads with significantly deeper core hole than thread depth

– use through-hole taps with a modified spiral point:
  - reduce the radial relief of the chamfer to the value of a blind hole tap
  - shorten the chamfer length to approx. 3 threads

Advantage: longer tool life than blind hole taps with a high helix angle
Disadvantage: chips remain in the bore

– for short-chipping materials such as GG25, straight-fluted tools without a spiral point can also be used:
  - Paradur® Eco CI

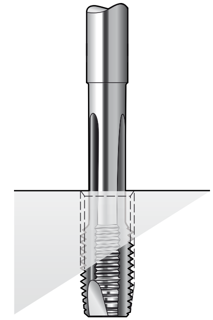
– of course, blind hole taps with a high helix angle can also be used for this application



Inclined thread lead-out

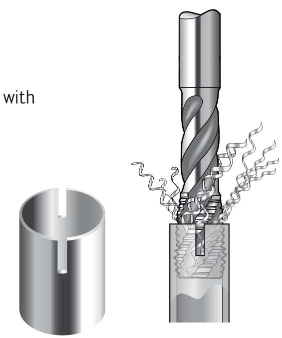
– use taps with a very long guide and maximum stability (e.g. Prototex® X·pert P, Prototex® X·pert M)
  - Inclinations of up to 30° are relatively unproblematic

– alternative: Thread milling



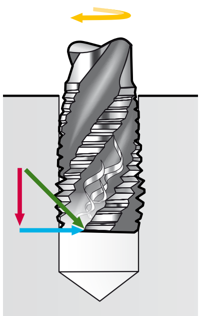
Slotted threads

– slotted threads should be machined with tools with a high helix angle:
  - Paradur® X·pert M
  - Paradur® X·pert P
  - Paradur® Eco Plus

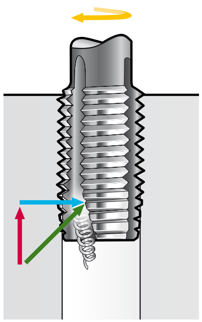


Process forces in thread tapping

Workpiece-related axial forces occur when cutting threads. Right-hand helical taps are subject to an axial force in the feed direction. On taps with a spiral point, this force acts against the feed direction.



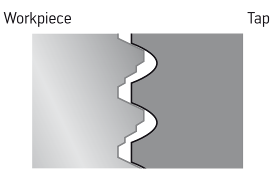
Process forces in right-hand helical taps



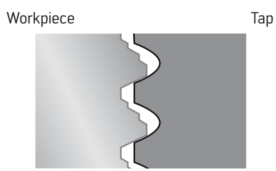
Process forces in taps with a spiral point

Direction of rotation
Cutting force
Axial force
Radial force

If floating chucks are used, these axial forces can cause the thread to be cut too large – this is known as axial miscutting. The tendency toward axial miscutting is increased if tools with a high helix angle and a large flank clearance angle are used in soft materials or if the cutting edge treatment is inappropriate.



Axial miscutting of the thread caused by right-hand helical tools: miscutting of the bottom face of the flank



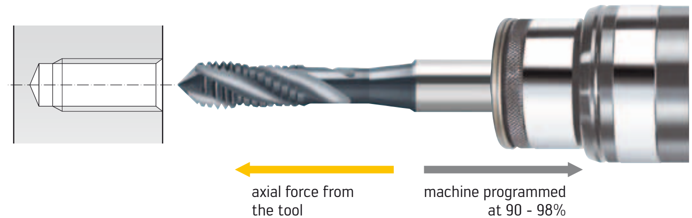
Axial miscutting of the thread caused by left-hand helical taps or taps with a spiral point: miscutting of the top face of the flank

For additional information on miscutting and for countermeasures, see page 91 (Problems and solutions for thread tapping).

Programming the feed if floating chucks are used

If tapping chucks with length compensation are used, the workpiece-related axial forces which occur during machining must be taken into account.

Spiral blind hole taps create an axial force in the feed direction. This force must be countered with minus programming.

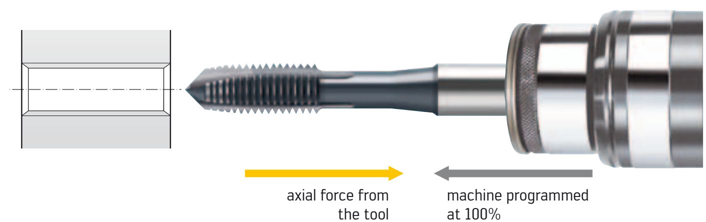


The usual feed values for this application lie between 90 and 98% of the theoretical feed. The theoretical feed rate can be calculated using the following formula:

$$v_f = n \times p$$

n = rotation speed; p = thread pitch

The conditions are reversed with **left-hand helical tools** and with **taps with a spiral point,** causing the axial forces to act against the feed direction.



Programming the theoretical feed rate is recommended here.

Modifications

	Negative chamfer (Secur chamfer)	Shortened chamfer	Reduced helix in the chamfer	Inclined thread	Bright rake
					
Chip formation	Chips are rolled more tightly, shorter chips	Chips are rolled more tightly, less chips	Chips are rolled more tightly, shorter chips	No change	Chips are rolled more tightly, shorter chips
Tool life	increases	decreases sharply	uncoated: decreases coated: increases	increases	decreases
Thread quality	decreases	decreases	uncoated: decreases sharply coated: remains unchanged	remains unchanged	decreases
Chip thickness	remains unchanged	increases	remains unchanged	remains unchanged	remains unchanged
Torque	increases	decreases	decreases	decreases	remains unchanged
Application example	Avoidance of bird nesting in structural steels such as St52, C45, etc.	Threads nearly to the bottom of the hole, better chip control	Optimised chip formation in steels and aluminium	Problems with fractures or weld formations in the guide	Optimised chip formation in steels, machining crankshafts
Standard tools with appropriate modification	Paradur® Secur Paradur® HSC Prototex® HSC	All tools with chamfer form E/F	Paradur® Ni 10 Paradur® HSC	Paradur® Eco Plus Paradur® X·pert M Paradur® Synchrospeed	All uncoated tools as well as Paradur® Synchrospeed (TiN-vap)

increases   remains unchanged   decreases   decreases sharply

Problems and solutions

Chip control:

Chip control is a major topic when tapping blind holes, particularly with deep blind holes in tough, long-chipping materials. Problems with chip control can be seen in snarl chips, randomly occurring torque peaks, tooth fractures in the guide and/or total breakage.

Remedy:

Standard taps can be modified* or new designs can be created to optimise chip control:

- regrind a reduced helix to achieve short chips
- reduce the rake angle to achieve more tightly rolled chips
- if tools with a shallow helix or straight flutes are used, the above-mentioned measures can be combined and complemented with a supply of axial coolant which helps to flush the short chips out; in mass production in particular, this is a proven method for increasing process reliability and productivity
- regrind the rake, and grind a bright reduced helix; this produces chips which can be better controlled
- replace the TiN/TiCN coatings with THL, because THL has better chip formation characteristics; use of bright or vaporised tools instead of coated
- shorten the chamfer (re-engineer) – fewer and thicker chips are produced
- reduce the number of flutes (new design); the chip thickness increases and the stability of the tool is increased
- use a tool with a negative chamfer (e.g. Paradur® Secur)

In general, the following is true:
The higher the material strength and the lower the elongation at fracture of the material, the greater the chip control is. Chip control is most difficult with soft structural steels, low alloy steels and stainless steels with a low tensile strength.

The more interference to chip formation from the aforementioned measures results in a worsening of the quality of the thread surface. For this reason, it is essential to match the measures with the customers requirements.

- thread forming or thread milling: materials in which chip control is difficult while tapping blind holes can in most cases be produced through forming in a non-chipping process. If thread forming is not permitted, thread milling can be used as a problem solver. This process produces short chips.

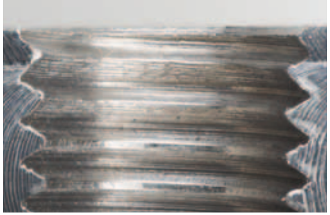
Example of fractures due to chip control problems

Miscutting:

The geometry of taps is customised to certain applications. If used improperly, taps can produce threads that are too large – this is known as miscutting.

Comment:
Miscutting during thread forming, thread milling and synchronous thread cutting is largely excluded.

Miscutting is most likely to occur with more highly spiralled blind hole taps. The axial force in the feed direction created due to the helix angle can pull the tap more quickly into the hole than at a rate which corresponds than the actual pitch – this is referred to as the “corkscrew” effect and is known as **axial miscutting**. Through-hole taps are subject to geometry-related axial forces against the feed direction, which similarly may lead to **axial miscutting**. The tendency toward axial miscutting is increased if taps with a large flank clearance angle are used in soft materials or if the cutting edge treatment is inappropriate.

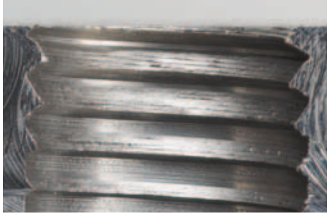


Axially miscut blind hole threads

Taps that miscut for the reasons mentioned above systematically produce threads that are too large. Sporadic miscutting may occur if single-sided radial forces act on the tool due to chip packing or because of weld formations on the material – this is known as **radial miscutting**.

Remedy:

- synchronous machining
- use tools which have been adapted to the material
- choose a suitable coating (against radial miscutting)
- optimise chip control (against radial miscutting)
- use a tap with a smaller helix angle
- use a tap which has been specially treated:
  - Paradur® X·pert P; Paradur® Eco Plus
  - Prototex® X·pert P; Prototex® Eco HT
- Thread milling
- Thread forming



Axially miscut through hole threads

* The modifications are explained fully and clearly shown on pages 88 - 89.

Problems and solutions

Thread surface:

The thread surface is determined by:
- the production process: cutting, forming, milling
- the wear on the tool
- the geometry
- the coating
- the material to be machined
- the coolant and its availability in the operating area of the tool

Comment:
In thread cutting and thread forming, there is almost no possibility to influence surface finish quality using the cutting data. In contrast to this, the cutting and feed rates can be selected independently of each other for thread milling.

Optimisation of the thread surface during thread cutting:
- use thread forming or thread milling instead of thread cutting
- increase the rake angle
- decrease the chip thickness by using a longer chamfer or an increased number of flutes (with blind hole taps this nevertheless worsens chip formation)
- as a rule, TiN and TiCN produce the best surfaces in steel (bright tools or CrN and DLC layers produce the best surfaces in Al)


Tap with TiCN layer in AlSi7


Tap with DLC layer in AlSi7

- concentrate emulsion or use oil instead of emulsion
- supply lubricant directly to the operating area
- replace the tool with a new one earlier

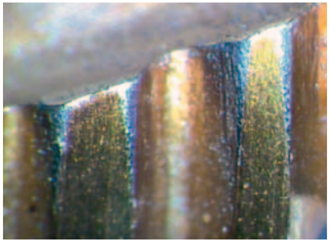
Some of the suggested measures might lead to an improvement in the surface quality, but are accompanied by a worsening in chip control – which is problematic with deep blind holes in particular. Here again a compromise that takes the customer's requirements into account must be chosen.

Wear:

A high level of hardness ensures a high resistance to wear and thus a long tool life. An increase in the hardness normally leads, however, to reduced toughness.

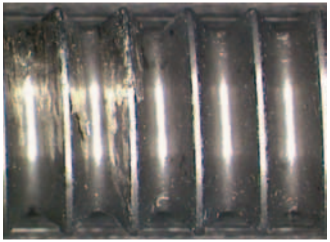
If small dimensions and highly spiralled tools are used, a high level of toughness is required, because otherwise total breakage can occur.

The hardness of the tool can normally be increased without difficulty for thread formers, straight-fluted and low-spiralled tools, as well as for machining abrasive materials with a low tensile strength.


Example of abrasive wear

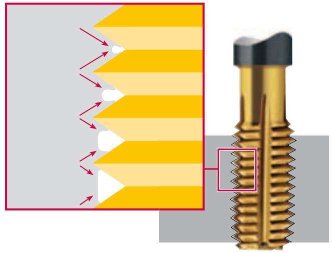
Weld formations on the tool:

Special coatings and surface treatments are recommended as a problem solver dependent on the material to be machined:
- Al and Al alloys: bright, CrN, DLC, WC/C
- soft steels and stainless steels: vap
- soft structural steels: CrN

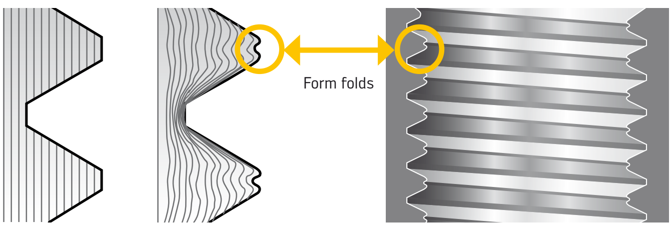

Example of weld formations

Process principles

Thread forming is a non-chipping process that uses cold forming to produce internal threads. Displacement of the material forces the material to yield. This produces a compressed thread profile. The flutes that are required in thread tapping can be omitted which increases the stability of the tool.



Both the pull-out strength under static load as well as the fatigue strength under dynamic load increases significantly due to cold work hardening used in combination with the uninterrupted grain profile of formed threads (compare picture at the bottom right). In contrast, the interrupted grain profile is used in thread tapping and thread milling (compare the picture on the bottom left).



Please note that in the area of the crest on formed threads, there is always a form fold. For this reason, thread forming is not permitted in all industries. Specific restrictions are listed adjacently.

- food industry and medical technology (germ formation in the area around the form fold)
- automatic component screw connections (screw may jam in the form fold)
- not permitted in the aircraft industry

Thread forming is predestined for mass production – for example in the automotive industry. Extremely reliable processes can be performed based on the non-chipping production of threads in combination with higher tool stability from the closed polygon profile. Moreover, in contrast to thread tapping, higher cutting parameters can often be achieved at the same time as achieving a longer tool life. In comparison with thread tapping, thread forming requires a torque that is approx. 30% higher.

Comment:
Compared to thread tapping and thread milling, the tolerance of the core hole is tighter in thread forming. Thread forming is therefore not always the more efficient option in all cases. Examining individual cases is therefore absolutely essential. Refer to pages 70 - 71 for the formulas required to calculate the core holes.

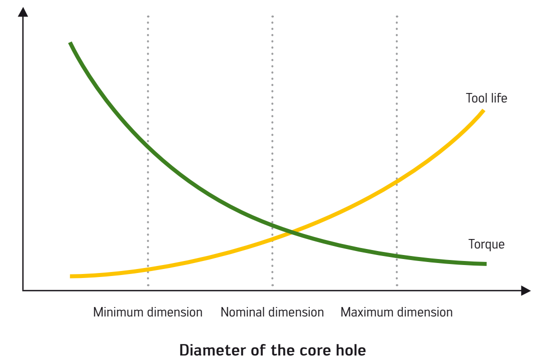
The different chamfer forms are useful in different applications:
- Form D, 3.5 - 5.5 thread: Through hole threads
- Form C, 2 - 3.5 thread: Blind hole and through hole threads
- Form E, 1.5 - 2 thread: Deep hole threads

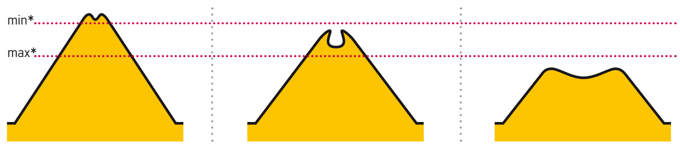
Approx. 65% of all machined materials in industry are formable. The limits are illustrated below:
- brittle materials with elongation at fracture lower than 7% such as:
  - GG
  - Si alloys with an Si content > 12%
  - short-chipping Cu-Zn alloys
  - thermosetting plastics
- thread pitch > 3 mm (forming at pitches ≤ 1.5 mm is particularly cost-efficient)
- tensile strength > 1,200 - 1,400 N/mm^2

Typical materials used in thread forming are:
- Steel
- Stainless steel
- Soft copper alloys
- Aluminium wrought alloys

Influence of the pilot hole diameter

The pre-drilled diameter of the core hole has a large influence on the thread forming process. On the one hand, the required torque and the tool life of the thread former are affected, but on the other hand, the formation of the thread is also effected. The graphic illustrates these relationships clearly.

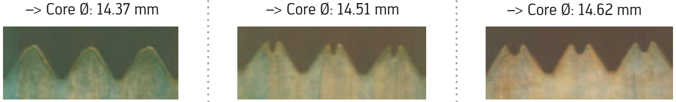




* Tolerance of the core diameter produced as per DIN 13-50

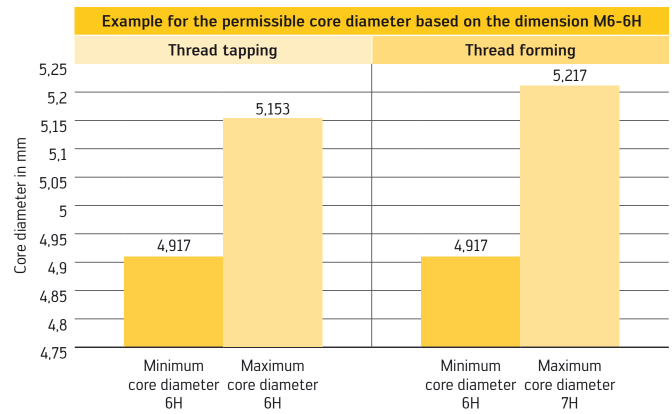
Example: M16 x 1.5-6H, 42CrMo4; Rm = 1100 N/mm^2

Pilot drill Ø: 15.22 mm –> Core Ø: 14.37 mm	Pilot drill Ø: 15.3 mm –> Core Ø: 14.51 mm	Pilot drill Ø: 15.34 mm –> Core Ø: 14.62 mm



Comment:
Dependency on the pilot drill diameter and thread core diameter:
If the core hole is drilled larger by 0.04 mm, the thread core diameter (after forming) enlarges by at least 0.08 mm – hence at least by a factor of 2.

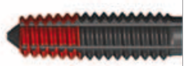
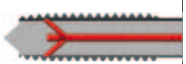
Larger core diameters are permitted for threads formed according to DIN 13-50 than for thread tapping. For example, for a thread formed with tolerance class 6H, the minimum thread core diameter must comply with tolerance class 6H, however the maximum thread core diameter is based on tolerance class 7H. This correlation is shown by way of example in the diagram below.



Practical tip:
Especially in mass production, it is worth optimising the pilot drill diameter. The following applies:
The pilot drill diameter selected should be as large as possible, but only as small as necessary.
The larger the pilot hole diameter:
– the higher the tool life will be
– the lighter and more reliable the forming process will be
– the lower the required torque will be
Ensure that the thread remains true to gauge.

The recommended pilot hole diameters can be found in the table on page 116.

Modifications

	Diagram	Action	Side effect
Chamfer form D		longer tool life	slightly increased cycle time
Chamfer form E		threads almost to the bottom of the hole and slightly shortened cycle time	decreased tool life
Radial coolant outlets		improved cooling and lubrication conditions (for deep threads and demanding materials)	higher tool costs
Lubrication grooves on the shank		better cooling and lubrication conditions (not as efficient as radial coolant outlets)	–
Increased total length		machining of areas that are difficult to access	–
Coatings and surface treatments		coating matched to the specific application	potentially higher tool costs

Problems and solutions

In general, thread forming is extremely reliable. The full advantages of thread forming are achieved if thread tapping is used to produce deep blind holes in soft and tough materials in which problems with chip removal are more likely to occur. For this reason, thread forming can truly be seen as a “problem solver”.
It is a fortunate technical coincidence that precisely the materials that most frequently have problems with chipping, such as St52, 16MnCr5, C15, can be formed well.

Thread forming is also advantageous if a very high surface finish quality is required. The depths of surface roughness of formed threads are normally much lower than those of cut threads.

Despite the advantages that are achieved through the non-chipping production of threads, there are also specific points about thread forming that must be noted in order to guarantee a reliable process:

- the pilot drill diameter has a tight tolerance (e.g. M6 ± 0.05 mm) compared to cutting threads
- no chips from drilling are permitted to remain in the core hole; this can be ensured using a twist drill with internal cooling or using a thread former with axial coolant outlets; in the latter case, the thread former should be positioned over the core hole for a short period with coolant on before forming starts
- the required torque for forming threads is higher than it is for tapping threads; the chuck setting value is therefore to be increased where required
- more attention must be devoted to the coolant and the supply of coolant during forming; the effects of briefly running dry are greater than with cutting threads. This has to do with the higher surface pressure acting on the formed edges and the fact that the lubrication grooves used in forming have a narrower cross-section than the flutes of taps. The smaller lubrication grooves give the thread former greater stability, which is also required due to the increased torque. Larger lubrication grooves would cause the formed edges to crack easily due to the higher forces applied. Detailed information on correct cooling and lubrication can be found on page 60.
- the coefficient of friction of each coating is reduced as the temperature increases; higher forming speeds can therefore lead to longer tool life
- well-known automotive manufacturers often stipulate that the threads must comply with a specific thread overlap standard tools cannot always guarantee this with reliability

Comment:
Walter Prototyp is able to meet the special profile requirements of automotive manufacturers reliably.

Problems and solutions

Borderline cases for thread forming:

It is difficult to set clear material strength limits with forming, because there are always exceptions where limits have been exceeded successfully – or not even reached at all.

– Tensile strength
The limit is approx. 1200 N/mm² depending on the material and the lubrication conditions. Nevertheless, there are notable cases in which forming could be performed successfully on stainless steel using HSS-E thread formers and on hard-to-cut Inconel 718 using solid carbide thread formers. Both materials had a tensile strength of approx. 1450 N/mm².

– Elongation at fracture
In general, a minimum value of 7% is specified for the elongation at fracture. Nevertheless, there are also notable cases here too, in which for example GGG-70 has been formed with an elongation at fracture of about 2%. However, in this case tiny cracks in the flanks were clearly evident which the user could accept. In such cases, an increased strength due to forming should not be assumed.

– Pitch and thread profile
With pitches larger than 3 to 4 mm, the limits for the above-mentioned tensile strengths must be corrected downwards. Thread types with steep flanks (e.g. 30° trapezoidal threads) must be examined as an isolated case.

– Si content
AlSi alloys can be formed if the silicon content is not greater than 10%. Nevertheless, there are also notable cases in which the Si content was 12 - 13%. However, this lowered the surface finish quality and the pull-out strength of the thread.

– Form folds
The unavoidable form folds occurring on the crest of the thread may become problematic if automated processes are used to screw in bolts. The first thread pitches are sometimes threaded into the form fold.
Formed threads in components used in the food industry and medical technology are also avoided, because it is not possible to reliably eliminate contamination in the form fold caused by washing.

Comment:

Walter Prototyp is able to design special tools in which the form folds can be closed under specific conditions. There are notable cases in which customers who initially were against using thread forming decided to permit it for this reason.


Thread profile made with a standard former


Thread profile made with a special former

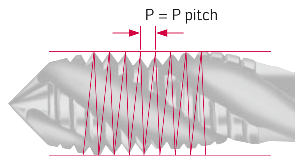
– Aerospace industry
Thread forming is not permitted in the aerospace industry. Changes to the structure that occur during thread forming or welding are avoided in general.

Process principles

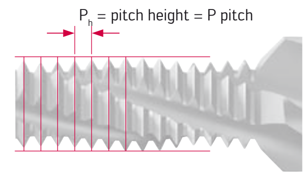
Basic aspects of thread milling:

– a machine tool with a 3D CNC control system is required (more or less a standard today)
– conventional thread milling to a depth of 2.5 x D_N is possible, orbital thread milling to a depth of approx. 3 x D_N
– higher costs compared to thread tapping
– milling threads with a small pitch and a large dimension is often quicker than if thread tapping and thread forming is used

In contrast to thread tapping and thread forming, the pitch is produced in thread milling by the CNC control system.



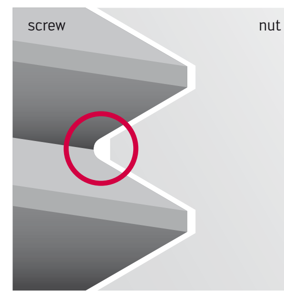
Thread tapping: The thread pitch P is produced by the tap/thread former.



Thread milling: The thread pitch P is produced by the CNC control system (circular program).

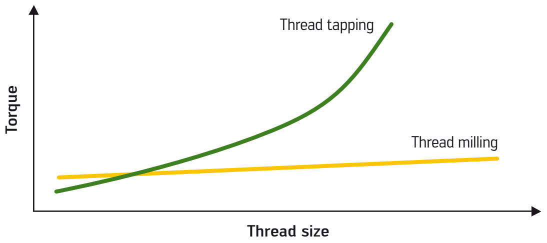
Theoretically, an internal thread milling cutter can also be used to produce an external thread. The threads produced in this way do not comply with the standard, because the external threads are rounded to minimise the notch effect in the core and the external diameter produced is too small.

However, because the thread ring gauge tests the thread on the pitch diameter, the gauge accuracy is maintained.



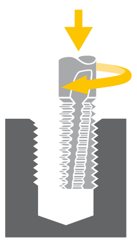
Process principles

In contrast to thread tapping and thread forming, the required torque for thread milling only increases moderately as the thread dimension increases. This means large threads can also be produced on machines with less drive power.

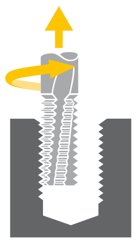


Thread milling is an extremely reliable production process.
Short chips are normally produced, which is why chip removal is unproblematic. Moreover, special chucks are not required for thread milling, and nearly all standard milling chucks can also be used for thread milling.

There are two fundamentally different milling processes:



Up-cut milling
from top to bottom in right-hand threads)
Up-cut milling is preferred when machining hardened materials, or as a remedy against conical threads.

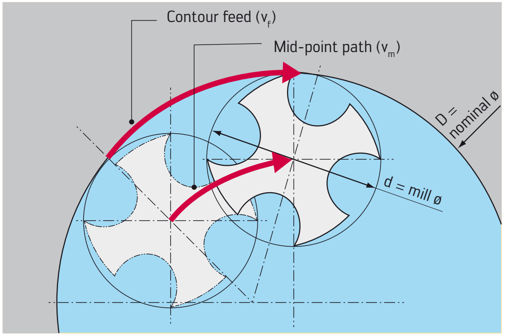


Synchronous milling
(from the bottom to the top in right-hand threads)
Synchronous milling increases tool life and prevents chatter marks, while promoting thread conicity.

Comment:
Walter GPS automatically determines the right process for the relevant application and takes into account the specific details relating to the tool and the machine.

Feed rate correction
Because thread milling uses a circular path, and the cutting edge therefore travels through a longer path than the tool centre, a distinction must be made between contour feed and tool centre feed.
Because the tool feed is always based on the tool centre point, the milling feed must be reduced.

Comment:
The relationship is precisely the other way around when milling external screw threads.



Walter GPS automatically reduces this when the CNC program is created. Some CNC control systems also reduce the feed automatically for the same reason. Reduction of the feed rate on the circular path must then be deactivated in the CNC program using the appropriate G command. The cycle time calculated by the GPS can be compared with the actual cycle time in order to determine whether the machine automatically corrects the feed.

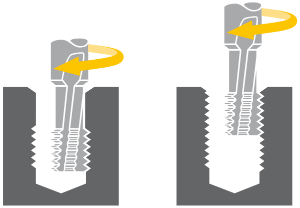
Practical tip:
The program can be tested during feed-in without operational engagement in order to determine whether the machine tool corrects the feed automatically. A comparison of the actual cycle time with the time determined by Walter GPS shows whether the feed must be adjusted in the CNC program.

Process principles

The cuts can be made in a number of passes in order to reduce the radial forces acting on the tool:

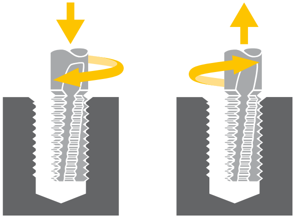
Comment:
Ensure that the thread mill is always moved by a multiple of the pitch when making axial cutting passes.

Axial passes

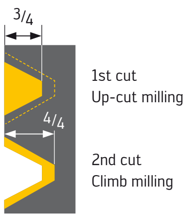


1st cut 2nd cut

Radial passes



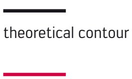
1st cut 2nd cut

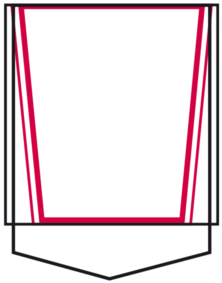


Advantages:
– longer thread depths can be produced
– reduced risk of tool breakage
– thread milling is possible even with a relatively unstable clamping arrangement
– counteracts conical threads

Disadvantages:
– increased tool wear
– higher production time

The cutting forces normally deflect a thread mill less at the shank than they do at the front cutting edge. This results in conical threads. With a conventional thread mill, it is therefore necessary to calculate a conicity of approx. 1/1000 mm for each mm of thread depth when machining steel. This is due to the radial forces acting on the thread mill.





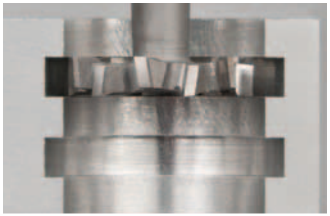
To counteract this physical law, the geometric design of thread mills is slightly conical. Nevertheless, under difficult machining conditions it may be necessary to find a remedy using one of the following measures:
– multiple radial cutting passes
– run all radial cuts in the opposite direction
– make a non-cutting or spring pass without additional infeed at the end of the process

Comment:
As an alternative, orbital thread mills (TMO) can be used to produce cylindrical threads right to the bottom of the hole.

The above-mentioned measures may increase the cycle time, but they are unavoidable in some cases if true to gauge threads cannot be guaranteed in any other way.
This conicity makes achieving true to gauge threads particularly difficult with tight tolerance threads as well as with materials that are difficult to machine (e.g. Inconel).

Profile distortion

Diagonal milling in the inclination angle causes a distortion of the thread profile of the tool to be transferred onto the component. This so-called profile distortion is shown below using a clear example.



No inclination – no profile distortion

Inclination P = 12 – profile is distorted

Comment:
The closer the milling cutter diameter approaches the thread nominal diameter and the higher the thread pitch, the more pronounced the profile distortion is.

To produce true to gauge threads, the following rules must be followed:

Metric threads:
Milling cutter diameter ≤ $^2/_3$ x thread nominal diameter

Fine metric threads:
Milling cutter diameter ≤ $^3/_4$ x thread nominal diameter

Example of profile distortion in a M18 x 1.5 thread

Thread mill diameter in mm	Flank offset due to profile distortion in mm
16	0,0386
14	0,0167

Theoretically, any size thread can be produced with small thread mills. However, the tool life is decreased as the thread dimension increases, and the stability of the tool and the length of the cutting edges are limiting factors.

Comment:
Because of profile distortion, special threads and threads with small flank angles need to be assessed for technical feasibility.

CNC programming

CNC programming with Walter GPS
Generally it is recommended to create the CNC program using Walter GPS. This makes perfect sense because, in contrast to preprogrammed machine cycles, GPS includes the stability of the tool in the calculation, and a reduction in the cutting data or a radial cutting pass is provided if any tool is overloaded.

Comment:
It is advantageous to make a radial cutting pass at a constant feed per tooth, instead of selecting a cut and decreasing the feed per tooth. At a low feed per tooth, the wear on the cutting edge in particular is disproportionately high.

Walter GPS enables even inexperienced users to create a thread milling program for 7 different control systems easily and reliably. In contrast to the previous CCS, handling has been greatly simplified. In addition, the most cost-effective strategy for producing threads is recommended automatically.

Each line in the program has comments so that the machine movements are always understandable (different languages can be selected). The example below is of a CNC program for milling an internal thread on a control system complying with DIN 66025.

Comment	Code
Tool radius presetting	;Tol. 6H: R='Rprg'-0.045 mm
Tool call in	01 M6 T
Selection of working plane	02 G90 G17
Spindle on	03 S5640 M3
2 mm above workpiece surface on centerline of thread	04 G00 X0.000 Y0.000 Z2.000
Start incremental programming	05 G91
Move to required depth on centerline of the predrilled hole	06 G00 Z-17.375
Set approach path for entry loop	07 G41 G01 X0.000 Y3.750 F1450
Move to the contour starting point	08 G03 X0.000 Y-8.750 Z0.375 I0.000 J-4.375 F176
Thread milling	09 G03 X0.000 Y0.000 Z1.500 I0.000 J5.000 F363
Move out of the contour	10 G03 X0.000 Y8.750 Z0.375 I0.000 J4.375
Reset to centerline	11 G40 G01 X0.000 Y-3.750
Retract from thread	12 G00 Z15.125
Start absolute programming	13 G90

CNC programming

The programming radius “Rprg.”
The programming radius, abbreviated to Rprg., is an important variable for setup. The Rprg. is calculated based on the pitch diameter of the thread mill and enables true to gauge threads to be produced instantly. Approximating the correction value can be omitted. The Rprg. can be read from the tool shank and is to be entered in the tool table of the CNC control system when creating the CNC program during setup of the machine.

The Rprg. is defined so that when it is used in the CNC program, the mathematically smallest dimension for the thread tolerance is attained. If the CNC program is created using GPS, a correction dimension is displayed which can be used to attain the tolerance centre of the selected thread tolerance. The correction dimension must be subtracted from the Rprg., then the corrected Rprg. is to be entered into the CNC control system.

During the course of the tool’s life, the cutting edges become worn, the tool is forced back more strongly, and the threads are too narrow. This wear can be compensated for by reducing the Rprg. and true to gauge threads can still be produced. Correction increments in the order of 0.01 mm are recommended. In comparison with large tools, it is often not possible to correct the Rprg. of small tools, because the radial forces increase and this increases the risk of tool breakage. If the tools are to be reground, it is recommended to do this after 80% of the tool life has been reached.

Modifications

Diagram	Modification	Effect
	countersink and facing step	countersinking and facing step in one tool
	cooling grooves on the shank	systematic cooling without weakening of the tool cross-section in the cutting area
	radial coolant outlets	systematic cooling for through hole threads
	threads removed	reduced cutting forces but longer machining time, because two passes are required
	deburring cutter	removal of the incomplete thread pitch at the thread run-in area without an additional operation
	first thread profile lengthened on the face side	chamfering of the core hole
	grinding of the neck (necking)	enables axial cutting passes to be made – practical for deep threads

Problems and solutions

		Problem					
		Chatter marks	Low tool life	Cutting edge breakaway	Conical threads	Tool breakage	Accuracy to gauge
Cutting data/strategy/adjustments	f_z in [mm/tooth]	+	+	🔍	–	–	
	v_c in [m/min]	–	–	🔍		🔍	
	Programming			🔍		🔍	🔍
	Synchronous run	✓	✓				
	Reverse rotation				✓		✓
	Cutting pass	✓		✓	✓	✓	✓
	Programming radius [Rprg.]						–
	Cooling		+	+			
Workpiece	Clamping arrangement	🔍	+	+	🔍	🔍	🔍
	Pilot drill diameter	🔍	+	🔍	🔍	🔍	+
	Chip removal		+	+		🔍	
Tool	Stability/geometry	🔍	+	+	🔍	🔍	+
	Projection length	–	–	–	–	–	–
	Helix angle	+			+		
	Coating		🔍				
	Concentricity	🔍	🔍	🔍		🔍	🔍

Key:
🔍 investigate – reduce + improve/increase ✓ use is preferred

TMO – specialists for complex tasks:
Tools from the TMO family can often be used as a problem solver, for example, if deep threads must be produced, hardened materials are to be machined or if conventional thread mills create conical threads. Further information available on pages 36 and 102 - 105.

Conical threads:
Explanations and solutions to problems can be found on pages 102 - 105.

Comment:
The use of tools from the TMO family are a very good option for producing cylindrical threads.

Cooling and lubrication:
Problems related to cooling and lubrication as well as the corresponding remedial measures are described on page 59.

Hard machining:
- specially designed only for use with tools that are suitable for hard machining (TMO HRC and thread mill Hart 10)
- machining in reverse rotation where possible (see Walter GPS recommendation)
- select the largest, permissible pilot drill diameter
- if problems with the cylindricity of the threads occurs, make a non-cutting pass or use tools from the TMO HRC family
- do not use lubricant, remove the hard chips from the bore using an air blast or MQL

Formulas

Speed		
n [min^{-1}]	$n = \frac{v_c \times 1000}{d_1 \times \pi}$	[min^{-1}]
Cutting speed		
v_c [m/min]	$v_c = \frac{d_1 \times \pi \times n}{1000}$	[m/min]
Feed rate		
v_f [mm/min]	$v_f = p \times n$	[mm/min]

Core diameter for thread tapping and thread milling

M ISO metric coarse pitch thread

Designation as per DIN 13	Internal thread core diameter (mm)		Drill Ø (mm)
	6H min	6H max	
M 2	1,567	1,679	1,60
M 2,5	2,013	2,138	2,05
M 3	2,459	2,599	2,50
M 4	3,242	3,422	3,30
M 5	4,134	4,334	4,20
M 6	4,917	5,153	5,00
M 8	6,647	6,912	6,80
M 10	8,376	8,676	8,50
M 12	10,106	10,441	10,20
M 14	11,835	12,210	12,00
M 16	13,835	14,210	14,00
M 18	15,294	15,744	15,50
M 20	17,294	17,744	17,50
M 24	20,752	21,252	21,00
M 27	23,752	24,252	24,00
M 30	26,211	26,771	26,50
M 36	31,670	32,270	32,00
M 42	37,129	37,799	37,50

MF ISO metric fine pitch thread

Designation as per DIN 13	Internal thread core diameter (mm)		Drill Ø (mm)
	6H min	6H max	
M 6 x 0,75	5,188	5,378	5,25
M 8 x 1	6,917	7,153	7,00
M 10 x 1	8,917	9,153	9,00
M 10 x 1,25	8,647	8,912	8,75
M 12 x 1	10,917	11,153	11,00
M 12 x 1,25	10,647	10,912	10,75
M 12 x 1,5	10,376	10,676	10,50
M 14 x 1,5	12,376	12,676	12,50
M 16 x 1.5	14,376	14,676	14,50
M 18 x 1.5	16,376	16,676	16,50
M 20 x 1.5	18,376	18,676	18,50
M 22 x 1,5	20,376	20,676	20,50

UNC Unified Coarse Thread

Designation acc. to ASME B 1.1	Internal thread core diameter (mm)		Drill Ø (mm)
	2B min	2B max	
Nr. 2-56	1,694	1,872	1,85
Nr. 4-40	2,156	2,385	2,35
Nr. 6-32	2,642	2,896	2,85
Nr. 8-32	3,302	3,531	3,50
Nr. 10-24	3,683	3,962	3,90
$^{1}/_{4}$ -20	4,976	5,268	5,10
$^{5}/_{16}$ -18	6,411	6,734	6,60
$^{3}/_{8}$ -16	7,805	8,164	8,00
$^{1}/_{2}$ -13	10,584	11,013	10,80
$^{5}/_{8}$ -11	13,376	13,868	13,50
$^{3}/_{4}$ -10	16,299	16,833	16,50

UNF Unified Fine Thread

Designation acc. to ASME B 1.1	Internal thread core diameter (mm)		Drill Ø (mm)
	2B min	2B max	
Nr. 4-48	2,271	2,459	2,40
Nr. 6-40	2,819	3,023	2,95
Nr. 8-36	3,404	3,607	3,50
Nr. 10-32	3,962	4,166	4,10
$^{1}/_{4}$ -28	5,367	5,580	5,50
$^{5}/_{16}$ -24	6,792	7,038	6,90
$^{3}/_{8}$ -24	8,379	8,626	8,50
$^{1}/_{2}$ -20	11,326	11,618	11,50
$^{5}/_{8}$ -18	14,348	14,671	14,50

G Pipe thread

Abbreviation according to DIN EN ISO 228	Internal thread core diameter (mm)		Drill Ø (mm)
	min	max	
G $^{1}/_{8}$	8,566	8,848	8,80
G $^{1}/_{4}$	11,445	11,890	11,80
G $^{3}/_{8}$	14,950	15,395	15,25
G $^{1}/_{2}$	18,632	19,173	19,00
G $^{5}/_{8}$	20,588	21,129	21,00
G $^{3}/_{4}$	24,118	24,659	24,50
G 1	30,292	30,932	30,75

Thread forming core diameters

M metric ISO coarse pitch thread, tolerance 6H

Designation as per DIN 13	Internal thread core diameter as per DIN 13 - 50 (mm)		Pilot drill Ø (mm)
	6H min	7H max	
M 1,6	1,221	-	1,45
M 2	1,567	1,707	1,82
M 2,5	2,013	2,173	2,30
M 3	2,459	2,639	2,80
M 3,5	2,850	3,050	3,25
M 4	3,242	3,466	3,70
M 5	4,134	4,384	4,65
M 6	4,917	5,217	5,55
M 8	6,647	6,982	7,40
M 10	8,376	8,751	9,30
M 12	10,106	10,106	11,20
M 14	11,835	12,310	13,10
M 16	13,835	14,310	15,10

MF metric ISO fine thread, tolerance 6H

Designation as per DIN 13	Internal thread core diameter as per DIN 13 - 50 (mm)		Pilot drill Ø (mm)
	6H min	7H max	
M 6 x 0,75	5,188	5,424	5,65
M 8 x 1	6,917	7,217	7,55
M 10 x 1	8,917	9,217	9,55
M 12 x 1	10,917	11,217	11,55
M 12 x 1,5	10,376	10,751	11,30
M 14 x 1,5	12,376	12,751	13,30
M 16 x 1.5	14,376	14,751	15,30

Hardness comparison table

Tensile strength Rm in N/mm²	Brinell hardness HB [Brinell HB]	Rockwell hardness HRC	Vickers hardness HV	PSI
150	50		50	22
200	60		60	29
250	80		80	37
300	90		95	43
350	100		110	50
400	120		125	58
450	130		140	66
500	150		155	73
550	165		170	79
600	175		185	85
650	190		200	92
700	200		220	98
750	215		235	105
800	230	22	250	112
850	250	25	265	120
900	270	27	280	128
950	280	29	295	135
1000	300	31	310	143
1050	310	33	325	150
1100	320	34	340	158
1150	340	36	360	164
1200	350	38	375	170
1250	370	40	390	177
1300	380	41	405	185
1350	400	43	420	192
1400	410	44	435	200
1450	430	45	450	207
1500	440	46	465	214
1550	450	48	480	221
1600	470	49	495	228
		51	530	247
		53	560	265
		55	595	283
		57	635	
		59	680	
		61	720	
		63	770	
		64	800	
		65	830	
		66	870	
		67	900	
		68	940	
		69	980	

Torque setting for tapping chucks

Recommended values for torque adjustment of tapping chucks

Thread type	Size [mm]	Inclination [mm]	Torque setting value for cutting threads [Nm]	Fracture torque of tap [Nm]	Torque setting value for forming threads [Nm]
M, MF	1	≤ 0,25	0,03*	0,03	0,07*
M, MF	1,2	≤ 0,25	0,07*	0,07	0,12
M, MF	1,4	≤ 0,3	0,1*	0,1	0,16
M, MF	1,6	≤ 0,35	0,15*	0,15	0,25
M, MF	1,8	≤ 0,35	0,24*	0,24	0,3
M, MF	2	≤ 0,4	0,3*	0,3	0,4
M, MF	2,5	≤ 0,45	0,5	0,6	0,6
M, MF	3	≤ 0,5	0,7	1	1
M, MF	3,5	≤ 0,6	1,2	1,6	1,5
M, MF	4	≤ 0,7	1,7	2,3	2,4
M, MF	5	≤ 0,8	3	5	4
M, MF	6	≤ 1,0	5,5	8,1	8
M, MF	8	≤ 1,25	12	20	17
M, MF	10	≤ 1,5	20	41	30
M, MF	12	≤ 1,75	35	70	50
M, MF	14	≤ 2,0	50	130	75
M, MF	16	≤ 2,0	60	160	85
M, MF	18	≤ 2,5	100	260	150
M, MF	20	≤ 2,5	110	390	160
M, MF	22	≤ 2,5	125	450	170
M, MF	24	≤ 3,0	190	550	260
M, MF	27	≤ 3,0	220	850	290
M, MF	30	≤ 3,5	320	1100	430
M, MF	33	≤ 3,5	350	1600	470
M, MF	36	≤ 4,0	460	2300	650
M, MF	39	≤ 4,0	500		
M, MF	42	≤ 4,5	700		
M, MF	45	≤ 4,5	750		
M, MF	48	≤ 5,0	900		
M, MF	52	≤ 5,0	1000		
M, MF	56	≤ 5,5	1300		

Basis for the above-mentioned table: Material 42CrMo4, tensile strength 1000 N/mm², thread depth 1.5 x D_N. Using the conversion table, the values can be carried over to other materials.

With dimensions marked with a *, the torque required to produce a thread with a depth of 1.5 x D_N exceeds the fracture torque of the tool. Remedy: produce the thread in several operations.

Conversion for other materials

Material	Factor
Soft steel	0,7
Steel 1200 N/mm²	1,2
Steel 1600 N/mm²	1,4
Stainless steel	1,3
GG/GGG	0,6
Aluminium/copper	0,4
Ti alloys	1,1
Ni alloys	1,4

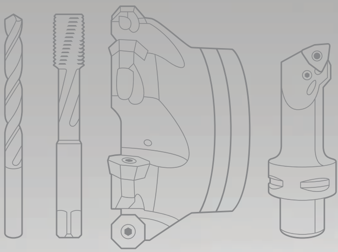
The table is used to set the torque of tapping chucks, insofar as these can be set. If the torque is set too high, there is a risk of tool breakage. If the torque is set too low, the tool can become jammed during machining, however the machine continues to run. If at this point the pressure compensation is not sufficient, the tool is destroyed and the machine can be damaged.

Notes

Walter AG

Derendinger Straße 53, 72072 Tübingen
Postfach 2049, 72010 Tübingen
Germany

www.walter-tools.com



Walter GB Ltd.
Bromsgrove, England
+44 (1527) 839 450, service.uk@walter-tools.com

Walter Wuxi Co. Ltd.
Wuxi, Jiangsu, P.R. China
+86 (510) 8241 9399, service.cn@walter-tools.com

Walter AG Singapore Pte. Ltd.
+65 6773 6180, service.sg@walter-tools.com

Walter Korea Ltd.
Anyang-si Gyeonggi-do, Korea
+82 (31) 337 6100, service.kr@walter-tools.com

Walter Tools India Pvt. Ltd.
Pune, India
+91 (20) 3045 7300, service.in@walter-tools.com

Walter (Thailand) Co., Ltd.
Bangkok, 10120, Thailand
+66 2 687 0388, service.th@walter-tools.com

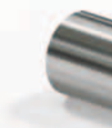
Walter Malaysia Sdn. Bhd.
Selangor D.E., Malaysia
+60 (3) 8023 7748, service.my@walter-tools.com

Walter Australia Pty. Ltd.
Hallam, Victoria 3803, Australia
+61 3 8793 1000, service.au@walter-tools.com

Walter Tooling Japan K.K.
Nagoya, Japan
+81 (52) 723 5800, service.jp@walter-tools.com

Walter USA, LLC
Waukesha WI, USA
+1 800-945-5554, service.us@walter-tools.com

Walter Canada
Mississauga, Canada
service.ca@walter-tools.com



Printed in Germany 632 4002 (11/2012) EN