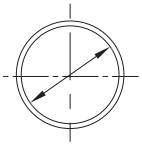
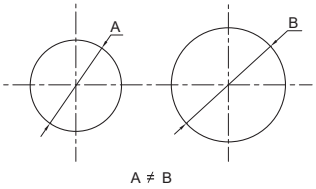
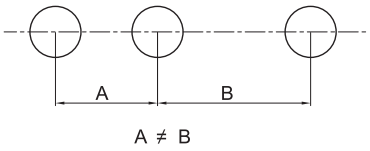
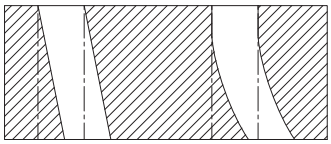
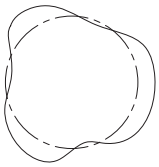
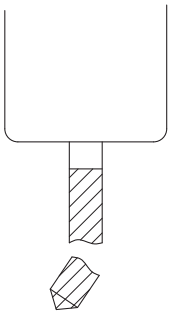
Technical information

C

Trouble shooting – solid carbide drills

Error	Reason	Countermeasure
Oversized holes 	– Insufficient clamping of workpiece and/or tool – Large radial run out – Point relief is off centre	– Use precision clamping – Reduce spindle play – Check and adjust clamped drill
	– Asymmetric point angle – Large radial run out – Point relief is off centre	– Regrind drill – Check quality of regrinding
Irregular hole size 	– Asymmetric point angle – Large radial run out – Point relief is off centre – High wear	– Use precision clamping – Reduce spindle play – Check and adjust clamped drill
	– Insufficient clamping of work piece and/or tool – Large radial run out – Point relief is off centre – High wear	– Use precision clamping – Reduce spindle play – Check and adjust clamped drill
	– Feed rate too high	– Reduce feed rate
	– Insufficient coolant	– Increase amount of coolant or change coolant supply
Low position accuracy 	– Insufficient clamping and spindle positio-ning – Large radial run out of spindle	– Improve positioning of machine – Use precision clamping – Calibrate spindle – Check and adjust clamped drill
	– The feed direction is not vertical to the workpiece surface	– Adjust feed rate vertically to workpiece surface
	– Tool isn't aligned with centre of spindle (lathe machines	– Centre the tool
Bad drill run out 	– High tool wear	– Regrind drill
	– Poor drill accuracy	– Improve positioning of drill
	– Asymmetric point angle – Large radial run out – Point relief is off centre	– Regrind drill – Check quality of regrinding
	– Insufficient tool stability	– Improve stability of tool
	– Uneven workpiece surface – Tool isn't aligned with centre of spindle (lathe machines)	– Before boring align the workpiece horizontally to the drill or pre-machine the workpiece
Inaccurate hole (roundness) 	– Asymmetric point angle – Large radial run out – Point relief is off centre – High wear	– Regrind drill – Check quality of regrinding
	– Insufficient clamping of work piece and/or tool – Large radial run out of spindle	– Use precision clamping – Calibrate spindle – Check and adjust clamped drill
	– Clearance angle too large	– Regrind the drill
	– Insufficient tool stability	– Improve tool stability

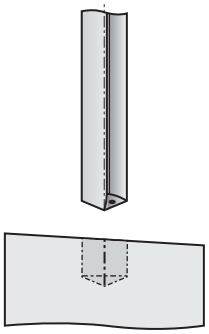
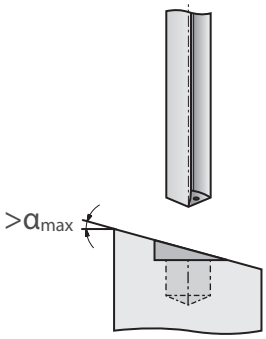
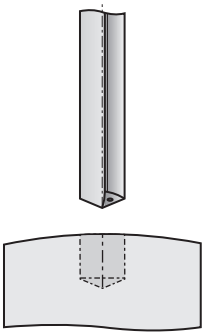
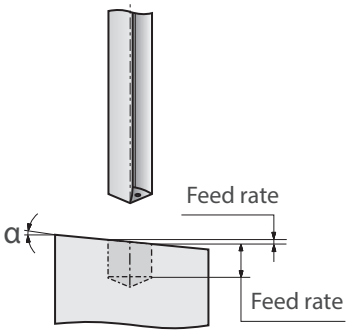
Trouble shooting – solid carbide drills

Error	Reason	Countermeasure
Bad surface quality	– Bad drill regrinding	– Improve regrinding
	– Insufficient amount of coolant or coolant method	– Change coolant supply – Increase amount of coolant
	– Insufficient clamping – Large radial run out of spindle	– Use precision clamping – Calibrate spindle
	– Feed rate too high	– Reduce feed rate
	– High wear of cutting edge – High welding	– Regrind drill – Use a coated drill
	– Bad chip removal	– Chose a suitable drill (with an accordingly flute, helical angle etc.) – Adjust cutting speed (reduce feed rate, etc.)
Bad cylindricity 	– No symmetrical point angle – Large radial run out (drilling) – Centre insert is off centre – Large cutting edge wear – Feed rate too low	– Regrind drill – Check regrind – Increase feed rate
Breakage of drill 	– Insufficient clamping of tool and/or workpiece	– Improve stability of tool and clamping of workpiece
	– Clearance angle too small	– Use a drill with bigger clearance angle or regrind
	– Feed rate too high	– Reduce feed rate
	– Excessive wear	– Regrind drill
	– Chip jamming	– Chose a suitable drill (considering flute geometry, helical angle, etc.) – Adjust cutting speed – Reduce feed rate
	– Drilling in uneven surfaces	– Increase rigidity of drill and clamping of machine and workpiece – Use drill with sharp centre insert – Pre-drill a centre hole – Create a straight surface (e.g. with solid carbide milling cutter) – Use a guide bush or bush plate
Chipping on the drill 	– Hard surface or blow holes	– Check material and chose suitable grade – Change cutting conditions (cutting speed, feed rate or machining method)
	– Feed rate too high	– Reduce feed rate
	– Insufficient coolant	– Improve/increase coolant supply
Chipping on the cutting edge 	– Poor clamping – Large radial run out	– Use a more precise clamping device – Adjust the spindle
	– Cutting speed and feed rate too high	– Reduce cutting speed and feed rate
	– Clearance angle too large	– Use a drill with smaller clearance angle or regrind

Trouble shooting – solid carbide drills

Error	Reason	Countermeasure
Excessive wear 	– Overdue regrinding	– Regrind in time
	– Drill tip not in centre position	– Adjust drill with centre of spindle
	– Cutting speed too high	– Reduce cutting speed
	– Cutting angle not suitable	– Chose right cutting angle
	– Material not suitable	– Chose suitable material
	– Insufficient cooling	– Use suitable cooling
Wear and chipping on point relief	– Feed rate too high	– Reduce feed rate
	– Cutting angle not suitable	– Chose right cutting angle
	– Material not suitable	– Chose suitable material
	– Clearance angle too small	– Regrind drill
Breakage on margin	– Guide bush too large	– Change guide bush
Built up edge on margin	– High wear and heat	– Regrind drill
	– Insufficient cooling	– Change cooling method
	– Wrong coolant	– Change coolant
	– Workpiece material is too soft	– Use drill with smaller clearance angle
High vibrations	– Clearance angle too large	– Regrind drill
	– Drill stability too low	– Improve stability
Swarf clogs the drill	– Long chips – Chip removal not fluent	– Optimise cutting data – Change drill or adjust machine
One-side wear	– Drill tip not centred	– Adjust drill with centre of spindle
	– Poor clamping	– Improve drill clamping – Check concentricity

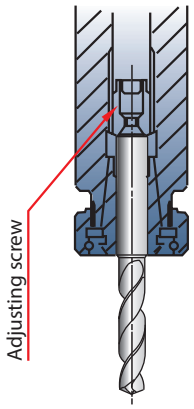
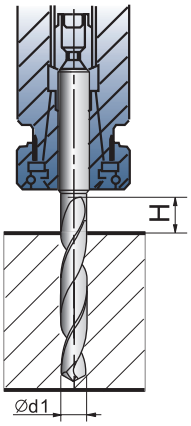
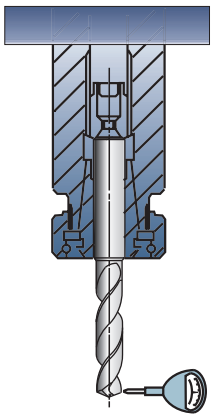
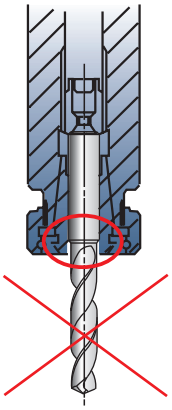
Trouble shooting – PC series

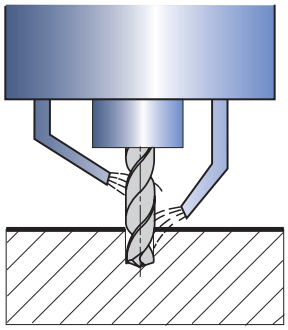
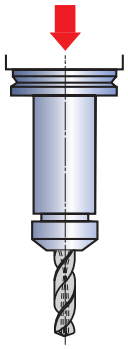
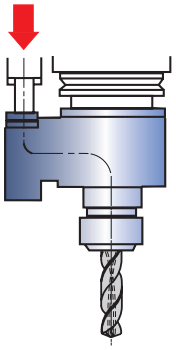
Machining	Recommendation
Sloped surface 	– Inclined surfaces should be pre-machined (chamfering). 
Inclined surface 	– Reduce feed rate accordingly. 

Inclination angle	Max. feed rate
1°	80%
2°	50%
3°	30%

Solid carbide drills

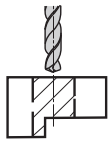
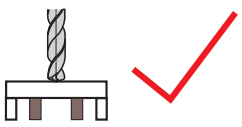
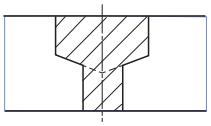
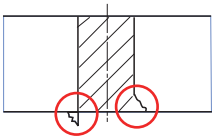
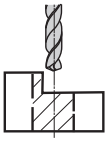
Operation notes

Correct drill clamping	Max. drilling length	Radial run-out	Wrong drill clamping
			
Use precision collets	H = 1,5 x d1	Radial run-out <0.02 mm	Don't clamp on the drill flutes.

External coolant method	Internal coolant method	
		
The coolant liquid should shoot to the end and the centre of the drill as shown in the figure.	Coolant pressure is about 0.5–1 mpa (coolant pressure is 2–3 mpa when the diameter is less than 5 mm), coolant volume: 1.5–4 L/min	

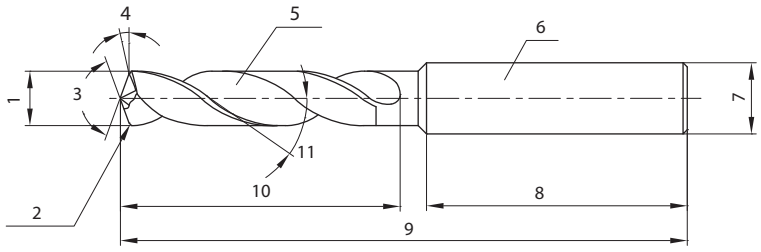
Handling of coolants:

1. Small chip particles and dust can cause jamming in the oil hole. A fine mesh filter should be used.
2. Dirt and dust particles will adhere to the oil hole and lead to unsmooth coolant flow. Regularly change the coolant. Please ensure proper coolant supply.

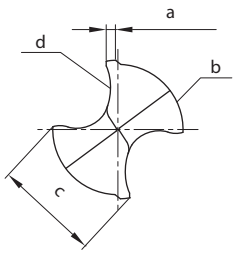
Interrupted cutting	Thin work pieces	Stepped holes	Burrs and work piece chippings on exit
 Reduce the feed rate when drilling interrupted cut.	  If bending occurs, add a supporter.	 – First drill the larger hole, then the smaller hole. – We can offer multiple step and chamfer drills on request.	 – Reduce the feed rate approx. by half when the drill exits. – Use a drill with a different point angle.
 Machine a countersink with an end mill prior to drilling.			

Solid carbide drills

Terminology

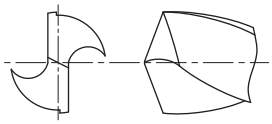
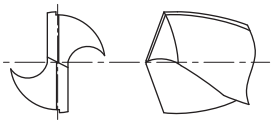
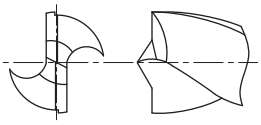


1. Drilling diameter
2. Chamfer
3. Point angle
4. Clearance angle
5. Chip pocket
6. Shank
7. Shank diameter
8. Shank length
9. Overall length
10. Flute length
11. Helical angle



a. Margin width
b. Body clearance
c. Land width
d. Primary cutting edge

Cutting edge type

Shape	 (Conical)	 (Dual flats)	 (Centring tip)
Features	– The flank face is conical and the clearance angel increases toward the centre of drill. – Wide applications, commonly used both for soft and hard materials.	– The flank face is dual flats, to facilitate cutting and initial entering. – Often used for small diameter drills.	– This shape has two-stage point angles for perfect centring capabilities and reduces burrs. – It is the first choice for drilling thin plate.

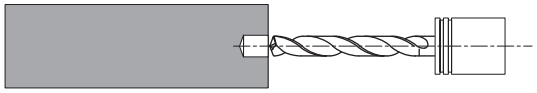
Solid carbide drills

Drill specification and cutting parameters

Chip pocket	The chip pocket ensures that the chips are removed out of the hole during processing.
Helical angle	The helical angle describes the pitch of the flute. It's specified according to the to be machined material. hardened materials — small ← helical angle → large — tough materials
Cutting edge length or spiral length	The cutting edge length needs to be specified according to the drilling depth, guide bushing length and the whole regrinding length. The larger the helical angle, the lower the stability. Since it greatly influences the tool life, it should be as small as possible. The recommended min. spiral length is the drilling depth plus 1.5 times of the hole diameter.
Point angle	Generally the point angle is 140°, for special applications it should be set differently. tough materials, easy to machine — small ← point angle → large — hardened materials and high-performance drilling
Core diameter	The core diameter is an important factor and influences the stability and the chip flow. low axial cutting force, low stability, for easy to machine materials — small ← core diameter → large — high axial cutting force, high stability, for hardened materials or cross holes
Chamfer width	The chamfer width influences the guidance and friction of the drill during machining. low friction and bad drill guidance — small ← chamfer width → large — high friction and good drill guidance
Back taper	The drill diameter is slightly reduced from cutting edge to shank to reduce friction during machining.
Body clearance	The area behind the chamfer width. The body clearance is necessary to reduce friction during machining.

Deep hole drilling

1 Preparation of the pilot hole with 1534SP03C*

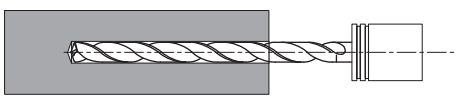


– Point angle of pilot drill must be bigger than SL drill.
– Diameter of pilot drill must be 0.01–0.04 mm bigger than SL drill.
– The pilot hole should be 1–3×D.

2 Entering the pilot hole with SL drill

– Entering the pilot hole with low cutting speed. (VC: 20–30 m/min)
– Stop 1–3 mm before end of pilot hole. (Vf = 0)
– Increase cutting speed up to recommended parameter and then start drilling at feed rate.

3 Manufacturing the deep hole



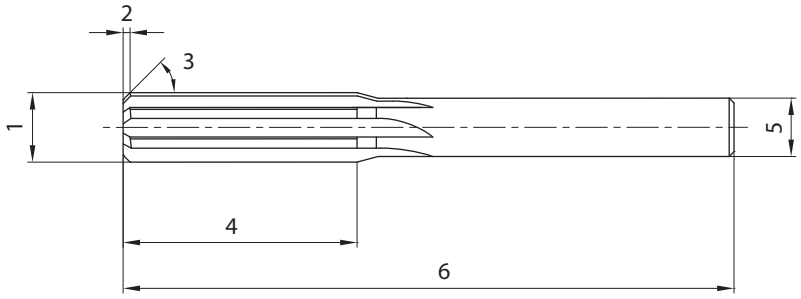
– Drilling with suitable cutting speed and feed rate.
In case of cross holes feed rate should be reduced to 0.05 mm/rev..

4 Pulling out the drill

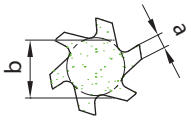
– After reaching the required depth reduce the cutting speed (VC: 20–30 m/min) and pull out the drill at high feed rate. (Vf: 2000 mm/min)

Solid carbide reamers

Terminology

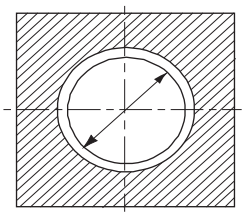
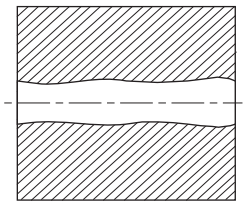
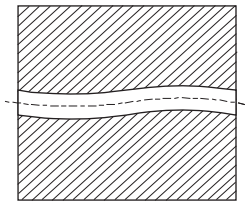
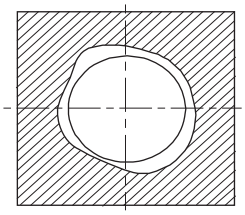
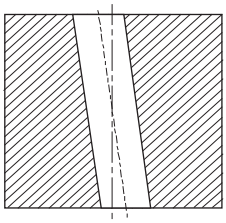
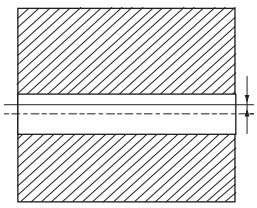


1.Nominal diameter
2.Chamfer length
3.Entry angle
4.Cutting edge length
5.Shank diameter
6.Total length



a.Cutting edge thickness
b.Core diameter

Reaming is semi-finishing and finishing of a previously formed hole within a narrow tolerance for higher surface quality, perfect roundness, cylindricity, etc.. To achieve a precisely reamed hole, the right choice of reamer and reamer diameter is important. In addition to that, the bore tolerance, the material and the machining conditions need to be taken into account. Furthermore the bore quality is strongly influenced by the radial run-out of the cutting tool.

 Diameter tolerance/Allowance	 Cylindricity	 Straightness
 Roundness	 Vertical deviation	 Off centre

Trouble shooting – solid carbide reamers

Problem	Solution
Oversized hole	– Reduce the diameter of the reamer. – Check concentricity of the reamer and hole. – Check the radial run-out of the reamer. – Check the shank of the reamer for scratches. – Select a suitable coolant. – Adjust the cutting parameters.
Hole too small	– Increase the diameter of the reamer. – Reduce the cutting speed. – Reduce allowance. – Regrind or replace the reamer. – Ensure sufficient cooling.
Poor hole roundness and straightness	– Guarantee concentricity of the reamer chamfer. – Reduce overhang. – Check radial run-out after the reamer is clamped. – Adjust concentricity of the reamer and hole. – Check and ensure drill geometry.
Poor surface quality	– Reduce the cutting speed. – Ensure correct reaming allowance. – Check the cutting chamfer length of the reamer for wear and built-up edge. – Ensure stability of the machine, tool holder and reamer. – Chose the reamer according to the application. – Check the hole allowance.
Poor bore quality	– Pull out the reamer in cutting direction. – Reduce the cutting speed. – Use reamers with more teeth. – Check for concentricity and radial run-out. – Improve coolant supply. – Chose the optimal coolant lubrication.
Reamer breakage and thermal damage	– The guide chamfer is insufficient. Check the drill and drilling axis. – Adjust machining allowance. – Ensure sufficient coolant supply. – Adjust the cutting speed and feed rate. – Improve the stability of the machine, the tool holder and the cutting tool. – Change or regrind the cutting tool if the cutter wear is too high.
Damage on reamer shank	– Check clamping sleeve and tool holder for damage.
Short tool life	– Check coolant supply. – Change from straight fluted to helical fluted reamers. – Check all factors affecting machining precision.
Scratched hole surface	– Check the cutting edge for built-up edges and if necessary correct the cutting data. – Improve clamping of the workpiece.
Trumpet-shaped entry hole	– Improve clamping of the workpiece. – Check radial run-out of the clamped reamer. – The centre of the reamer may not be aligned with the centre of the hole. Adjust concentricity.

Solid carbide thread formers

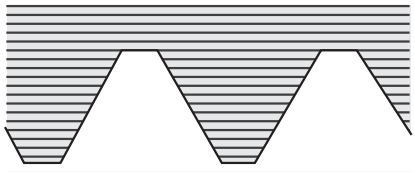
What is thread forming?

The material fibres aren't severed but compressed at the base of the thread. This is why no material is lost unlike when thread cutting.

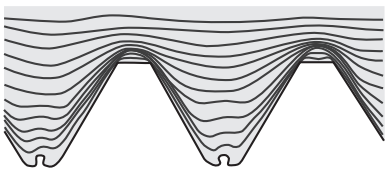
Advantages of thread forming:
- The thread is formed much more precisely.
- The thread is more resilient.
- The threads have a very smooth surface.
- Higher rotation speeds and feed rates possible than in thread cutting.
- Longer tool life increases the productivity.

Disadvantages of thread forming:
- Higher requirements on the hole tolerance.
- Can't be used as hand tool.
- Greater heat build-up than in thread drilling.
- Limited material choice.
- Often the use of a release agent is necessary.

Thread formers should be used in materials with good cold formability. Next to steel, stainless steel and aluminium alloys, these include light metals and light metal alloys with a yield strength of 1200 N/mm^2. Basically, all long-chipping materials are suitable.



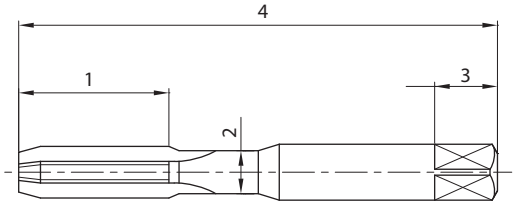
Fibre orientation after thread cutting



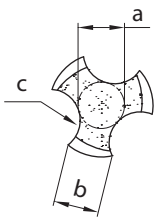
Fibre orientation after thread forming

Solid carbide taps

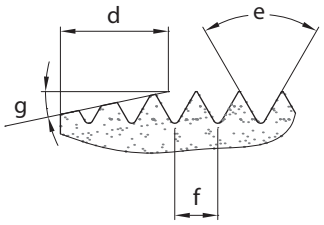
Terminology



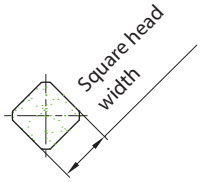
1.Thread length
2.Neck diameter
3.Square head length
4.Total length



a.Core diameter
b.Cutting edge thickness
c.Chip pocket



Chamfer and thread profile



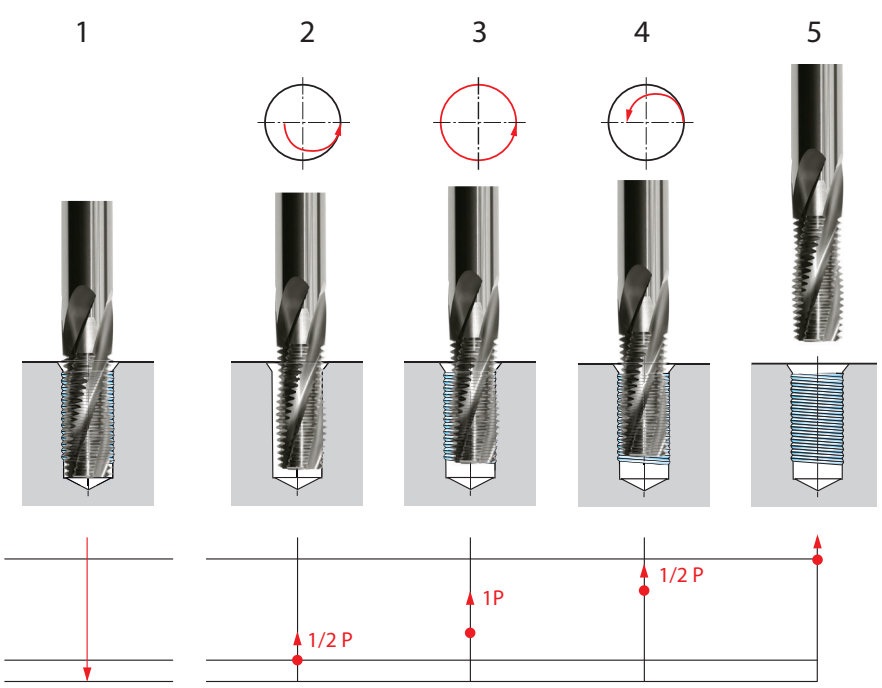
d.Chamfer length
e.Thread profile angle
f.Pitch
g.Chamfer angle

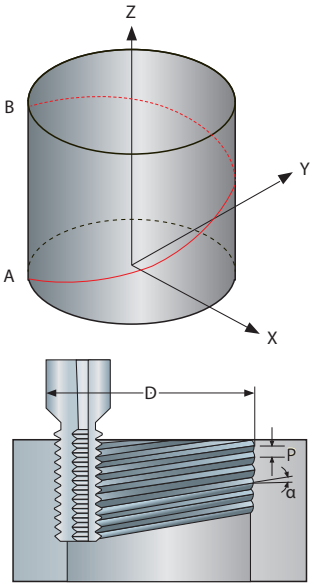
Chip space and application

Chip space type	Features	Application
 Helical flute tap	– Helical flute – No chips inside the hole – Good entering performance – Simple centring	– For long-chipping materials – Suitable for blind holes – Usage in holes with groove
 Straight flute tap	– Straight flute – Stable cutting edge – Easy regrinding	– For hard machining – For short-chipping materials – For through holes and blind holes – For wear material

Solid carbide thread milling cutters

Solid carbide thread milling cutters with cylindrical shank – example





α Helix angle
D Diameter
p Pitch

Nonstandard – solid carbide drills

Name/Company: **Address:** **Tel.:** **Fax:** **E-Mail:**	ZCC-CT **Wanheimer Str. 57** **40472 Düsseldorf, Deutschland** **Fax:** +49-(0)211-989240-111 **E-Mail:** technik@zccct-europe.com Scan for PDF

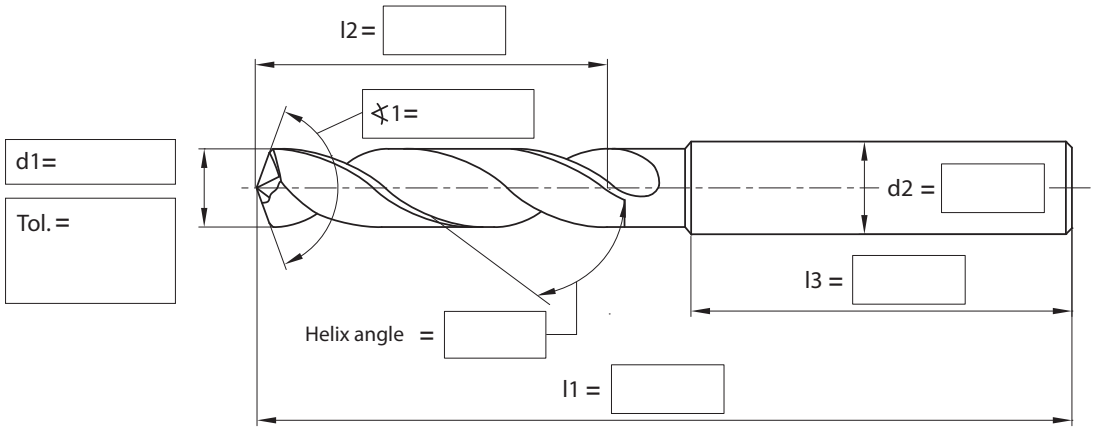
Material	
Material	
Tensile strength (N/mm^2)	
Hardness	

Twist drill series		
SU ☐	SH ☐	SL ☐
SUK ☐		SLK ☐
ST ☐	SC ☐	PA ☐

Cooling	
External	☐
Internal	☐

Tool holder type		
DIN6535		
Form HA ☐	Form HB ☐	Form HE ☐

Coating	
Yes ☐	No ☐

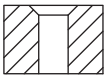
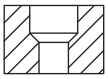


Remarks:	
Order quantity:	Expected delivery date:
Date:	Confirmation:

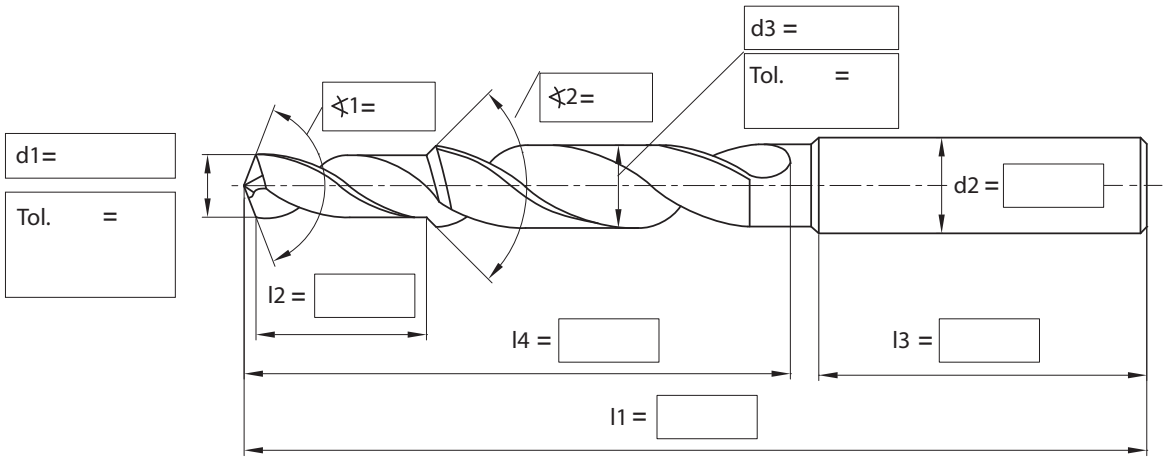
Nonstandard – solid carbide step drills

Name/Company: **Address:** **Tel.:** **Fax:** **E-Mail:**	ZCC-CT **Wanheimer Str. 57** **40472 Düsseldorf, Deutschland** **Fax:** +49-(0)211-989240-111 **E-Mail:** technik@zccct-europe.com Scan for PDF

Material	
Material	
ensile strength (N/mm²)	
Hardness	

Machining information	
 Chamfering ☐	 Stepped hole ☐

Cooling		Tool holder type			Twist drill series		Coating	
	External ☐	DIN6535			SU ☐	ST ☐	Yes	☐
	Internal ☐	 Form HA ☐	 Form HB ☐	 Form HE ☐	SUK ☐		No	☐
					PC ☐			



Remarks:	
Order quantity:	Expected delivery date:
Date:	Confirmation:

Nonstandard – solid carbide three-lips drills

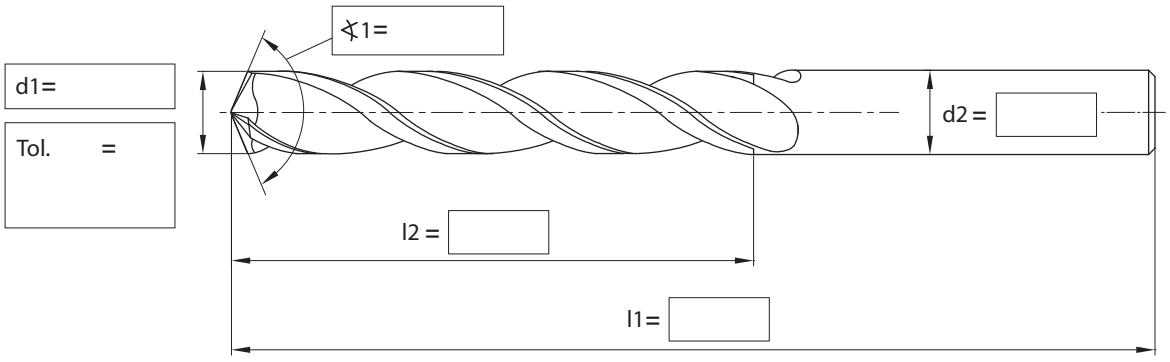
Name/Company: **Address:** **Tel.:** **Fax:** **E-Mail:**	ZCC-CT **Wanheimer Str. 57** **40472 Düsseldorf, Deutschland** **Fax:** +49-(0)211-989240-111 **E-Mail:** technik@zccct-europe.com Scan for PDF

Material	
Material	
ensile strength (N/mm^2)	
Hardness	

Tool holder type		
DIN6535		
Form HA ☐	Form HB ☐	Form HE ☐

Cooling	
External	☐
Internal	☐

Coating			
Yes	☐	No	☐



Remarks:	
Order quantity:	Expected delivery date:
Date:	Confirmation:

Nonstandard – solid carbide reamers

Name/Company: **Address:** **Tel.:** **Fax:** **E-Mail:**	ZCC-CT **Wanheimer Str. 57** **40472 Düsseldorf, Deutschland** **Fax:** +49-(0)211-989240-111 **E-Mail:** technik@zccct-europe.com Scan for PDF

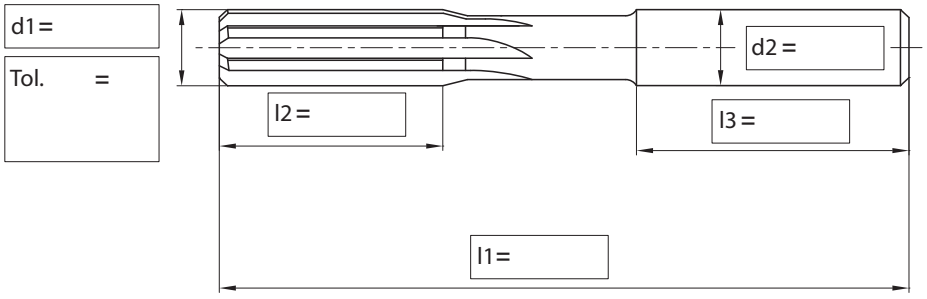
Material	
Material	
Tensile strength (N/mm^2)	
Hardness	

Coating	
Yes	☐
No	☐

Cooling	
External	☐
Internal	☐

Helix angle	
Right	☐
Straight	☐
Left	☐

Machining information	
Through hole	☐
Blind hole	☐



Remarks:	
Order quantity:	Desired delivery date:
Date:	Signature:

Nonstandard – solid carbide taps and thread formers

Name/Company: **Address:** **Tel.:** **Fax:** **E-Mail:**	ZCC-CT **Wanheimer Str. 57** **40472 Düsseldorf, Deutschland** **Fax:** +49-(0)211-989240-111 **E-Mail:** technik@zccct-europe.com Scan for PDF

Material	
Material	
Tensile strength (N/mm²)	
Hardness	

Coating	
Yes	☐
No	☐

Cooling	
External	☐
Internal	☐

Helix angle

Straight ☐

Right ☐

Machining information

Through hole ☐ Blind hole ☐

Thread profile

Thread angle = 60°

p

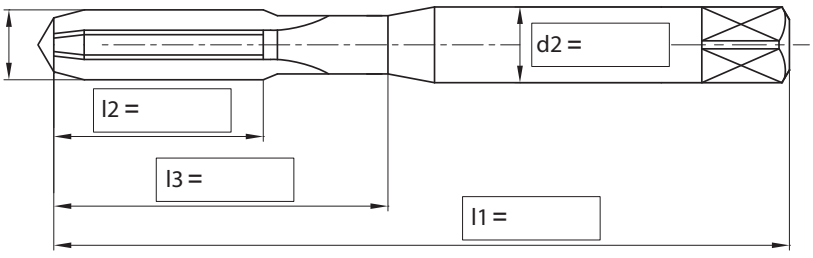
Pitch p = ☐

Tool type

Tap ☐

Thread former ☐

Thread size ☐



Remarks:	
Order quantity:	Desired delivery date:
Date:	Signature: