Contents

Drilling

A

B

C

D

E

F

G

H

A

B

C

D

E

F

G

H

Drilling

The drilling process

Drilling ...

... covers the methods of making cylindrical holes in a workpiece with metal cutting tools. Drilling is associated with subsequent machining operations such as trepanning, counterboring, reaming and boring. Common to all these processes is a main rotating movement combined with a linear feed. There is a clear distinction between short hole and deep hole drilling, the latter being a specialist method for making holes that have depths of many times (up to 150 times the diameter – see seperate catalogue.)

With the development of modern tools for short hole drilling, the need for preparatory and subsequent machining has changed drastically. Modern tools have led to solid drilling being carried out in a single operation, normally without any previous machining of centre and pilot holes. The hole quality is good, where subsequent machining to improve the measurement accuracy and surface texture is often unnecessary.

The **drilling** process can in some respects be compared with turning and milling but the demands on chipbreaking and the evacuation of chips is critical in drill-

ing. Machining is restricted by the hole dimensions, the greater the hole depth, the more demanding it is to control the process and to remove the chips. Short holes occur frequently on many components and high material removal rate is a growing priority along with quality and reliability.

Solid drilling is the most common drilling method, where the hole is drilled in solid material to a predetermined diameter and in a single operation.

Trepanning is principally used for larger hole diameters since this method is not so power-consuming as solid drilling. The trepanning tool does not machine the whole diameter, only a ring at the periphery. Instead of all the material being removed in the form of chips, a core is left round the centre of the hole – consequently, this method is for through-hole applications.

Counterboring is the enlargement of an exisiting hole with a specifically designed tool. This machines away a substantial amount of material at the periphery of the hole.

Reaming is the finishing of an exisiting hole. This method involves small working allowances to achieve high surface finish and close tolerances.
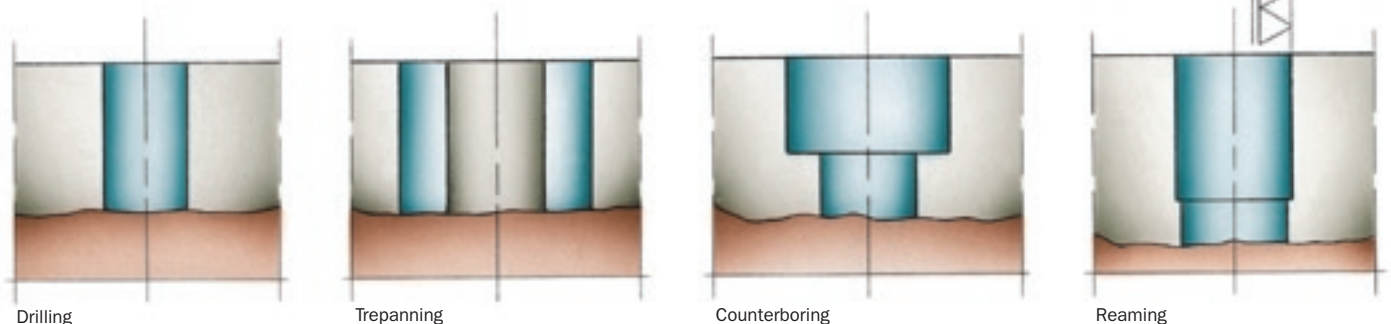
Drilling

Trepanning

Counterboring

Reaming

Cutting data

The **cutting speed**, or surface speed **(vc)** in for drilling is determined by the periphery speed and can be calculated from the spindle speed (n) which is expressed in number of revolutions per minute. During one revolution, the periphery of the drill will describe a circle with a circumference of π x Dc, where Dc is the tool diameter. The cutting speed also varies depending upon which cutting edge across the drill-face is being considered. A machining challenge for drilling tools is that from the periphery to the centre of the drill, the cutting speed declines in value, to be zero at the centre. Recommended cutting speeds are for the highest speed at the periphery.

The **feed per revolution (fn)** in mm/rev expresses the axial movement of the tool during one revolution and is used to calculate the penetration rate and to express the feed capability of the drill.

The **penetration rate** or feed speed (vf) in mm/min is the feed of the tool in relation to the workpiece expressed in length per unit of time. This is also known as the machine feed or table feed. The product of feed per revolution and spindle speed gives the rate at which the drill penetrates the workpiece.

The **hole depth (L)** is an important factor in drilling as is the radial cutting depth (ap) and **feed per tooth (fz)** for calculations.

Machining holes

Holes are either made or finish machined. Most workpieces have at least one hole and depending upon the function of the hole, it needs machining to various limitations. The main factors that characterize a hole from a machining point of view are:

- **diameter**
- **depth**
- **quality**
- **material**
- **conditions**
- **reliability**
- **productivity**
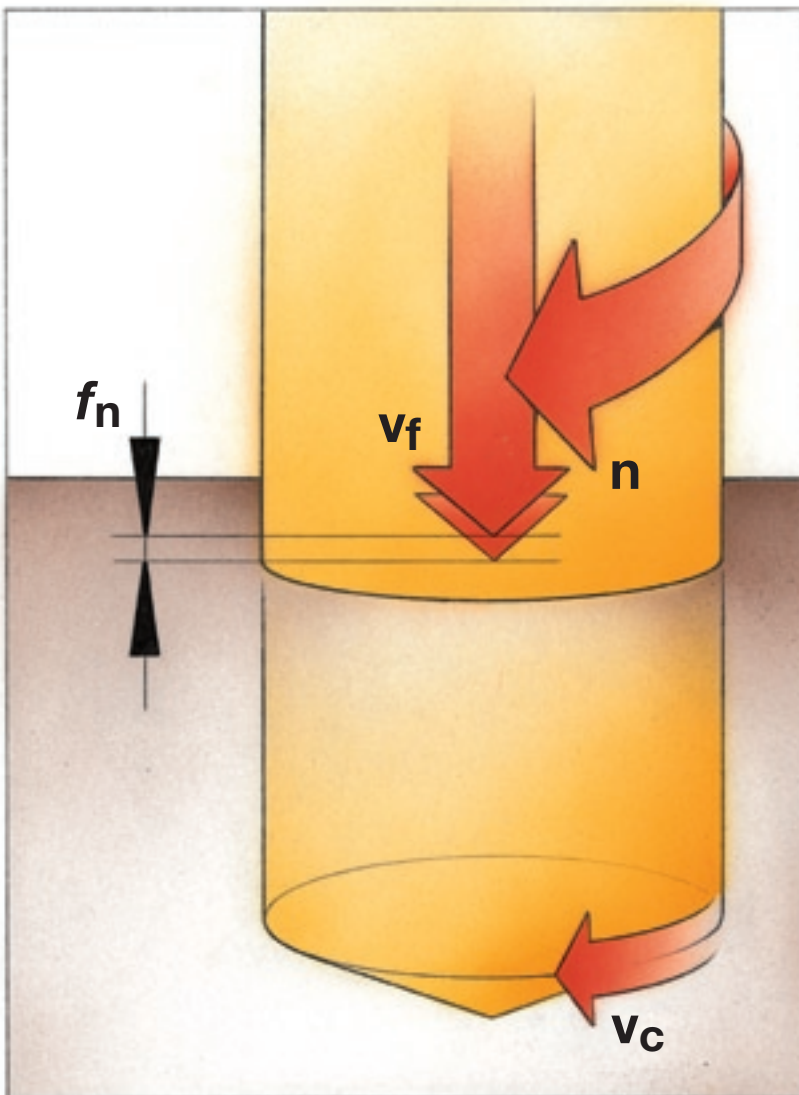
Cutting speed, penetration rate, spindle speed and feed per revolution.
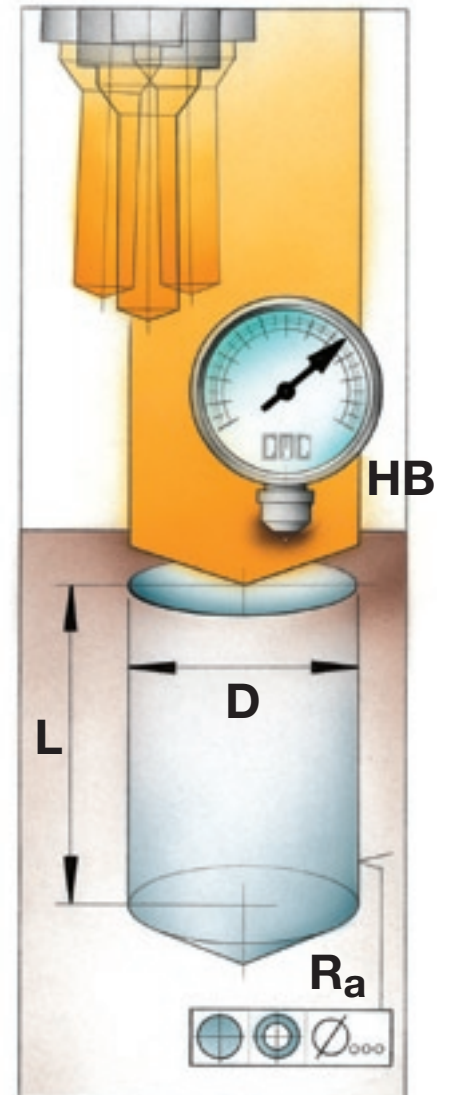
Main hole-machining factors.

SANDVIK
Coromant

Cutting forces and power

To produce a hole requires a certain amount of energy. Cutting forces act on the drill as it penetrates through the workpiece removing metal and generating a certain amount of power.

To start with, the power required in drilling varies with the type of workpiece material when calculating how much is required.

A specific cutting force for the material in question also needs to be established.

The **specific cutting force** value **(kc)** in N per square-mm has been worked out and tested for most materials and is available in a table relating to the effective rake angle of the tool and the average chip thickness. It is defined as the tangential cutting force needed for a chip with a certain cross-section (one square mm) or the effective cutting force divided by the theoretical chip area. Values are indicated for a certain feed per tooth. Steel normally has a specific cutting force some three times that of non-ferrous alloys, and a HRSA has a value of up to around twice that of steel.

In addition to the material factor, the **power (Pc)** in kW required for a drilling operation depends upon the diameter, feed rate and cutting speed. A formula is indicated for calculating the approximate power requirement for a certain operation and this can then be checked to ensure that the machine tool in question copes with the application. Most holes with a moderate diameter are no problem for modern machines but for large diamters with depths of several times the diameter, it is wise to check the power.

Torque (Mc) in Nm is another value which may be critical for some large-diameter drilling operations, especially trepanning, as regards the total drilling moment that the drill is subjected to during machining. The feed, diameter and material are the main factors that affect the torque value (see formula). The torque is the sum of the moments on each cutting edge and the product of the tangential force and radius from the centre.

The **feed force (Ff)** in N is usually the most important in driling from a performance point of view. This is the axial force acting on the drill as it penetrates the material. It needs to be considered in order to ensure that the spindle power and strength is sufficient for the drilling operation. Applying an excessive feed force can affect the hole quality, tool reliability and stall the machine. On the other hand, applying a sufficient feed force is important for the cutting action and also from producitivity point of view.

The feed force can be calculated from the provided formula and is related to the diameter of the drill, feed and material being drilled. The **cutting edge angle** of the drill **(κ_r)**, of the cutting edges, also influences the feed force. The point angle of the drill is (φ).
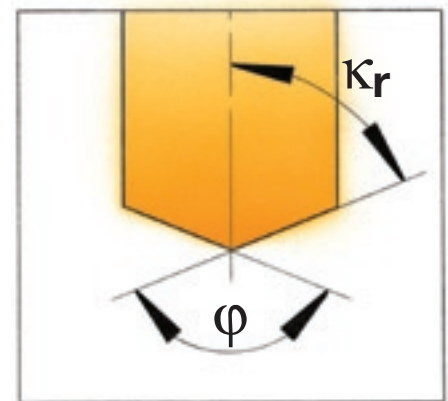
Point angle and cutting edge angle.
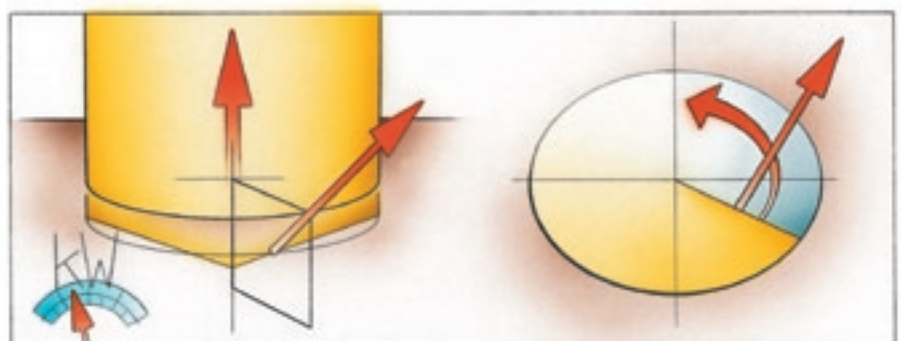
Feed force and torque.

Chip control and cutting fluid ...

... are important factors in drilling. Generating suitable chip forms and sizes and evacuating them are vital to the succes of any drilling operation. Without satisfactory performace in this regard, all drills will rapidly become ineffective due to clogging up the hole. Cutting speeds and feeds are high with modern drills but this has only been made possible through efficient evacuation of chips with cutting fluid.

Most short hole drills have two chip channels through which the chips are evacuated. With modern machines and drilling tools, this is be done very effectively by supplying cutting fluid internally through the tool coolant holes. The cuting fluid is ejected at the point of the drill during machining to lubricate the drill and flush out chips throught the channels.

Chip information is influenced by the workpiece material, tool geometry, cutting speed, feed and to some extent the choice of cutting fluid. Generally, increased feed and/or reduced cutting speed produces shorter chips. The chip length and form can be said to be acceptable if the chips can be flushed out reliably.

The **rake angle** (γ_E) of the drill varies along the cutting edge and decreases from the periphery towards the centre of the drill, such as with solid and brazed cemented carbide twist-drills. Since the cutting speed also drops from the periphery towards the centre, the cutting edge will work ineffectively at the point of the drill. As the point of the drill presses and scrapes the material rather than cuts it, plastic deformation tends to occur where the rake angle is negative and the cutting speed low. This pressure gives rise to a relatively high axial-force component. If the machine is weak in relation to the size of hole to be drilled, and the generated feed force, the machine spindle may deflect and, as a result, oval holes may be produced.

Drilling with modern cemented carbide drills enables high material removal rates to be achieved and large volumes of chips to be flushed out with **cutting fluid**, supplied internally under high pressure. The required **pressure (M_{pa})** and **flow** (l/min) are primarily dependent on the hole diameter but are also affected by the machining conditions and the workpiece material.

When cutting fluid is supplied internally, rotating drills require higher cutting fluid pressure than non-rotating drills due to the drop in pressure caused by the effect of the centrifugal force. In order to avoid having to compensate with very high pressures on the cutting fluid supply, a **volume compensator** can be used. But a certain drop in pressure in the conductive system must also be taken into account for non-rotating drills and with external cutting fluid supply.

The pressure in the system should be checked as well as the flow, so that the latter is at least at the level which is recommended for the drill and that there is a good margin at the tank.The cutting fluid flow should be measured at the cutting edge of the drill as this is where the recommended values apply. **Minimum flow and pressure values are recommended relative to drill type and diameter.**
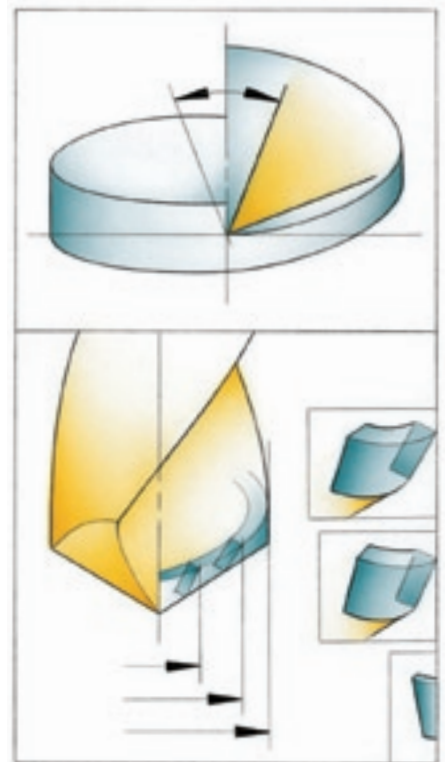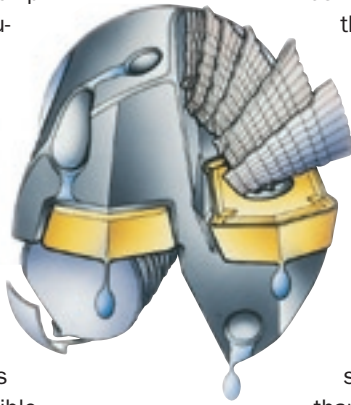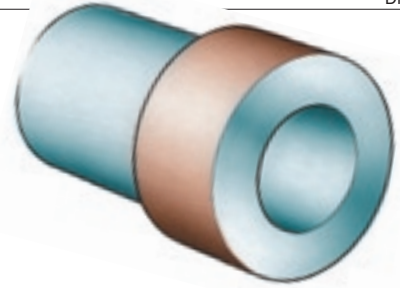

Rake angle of drill.
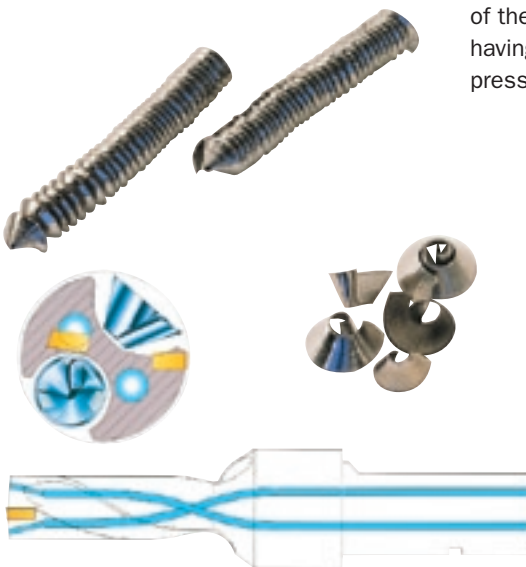
Chip formation, chip evacuation and cutting fluid supply.

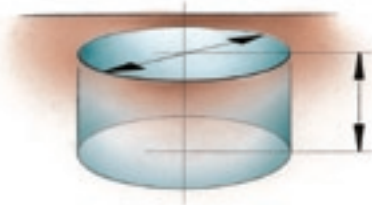Suitable cutting fluid supply is critical in drilling.

Drill selection procedure

① **Define your hole diameter, depth and quality requirements**
Consider also production economy and machining reliability aspects.

② **Select the type of drill**
Choose a drill for roughing and/or finishing holes. Check that the drill is suitable for the workpiece material, hole quality demands and that it provides the best hole economy.

③ **Choose the drill grade and geometry**
If an indexable insert drill has been selected, inserts have to be selected seperately. Find the right inserts for the drill diameter and choose recommended geometry and grades for the workpiece material. For solid or brazed cemented carbide drills, select suitable grade.

④ **Select the shank style**
Many drills are available with different mounting options. Find the style compatible with the machine.
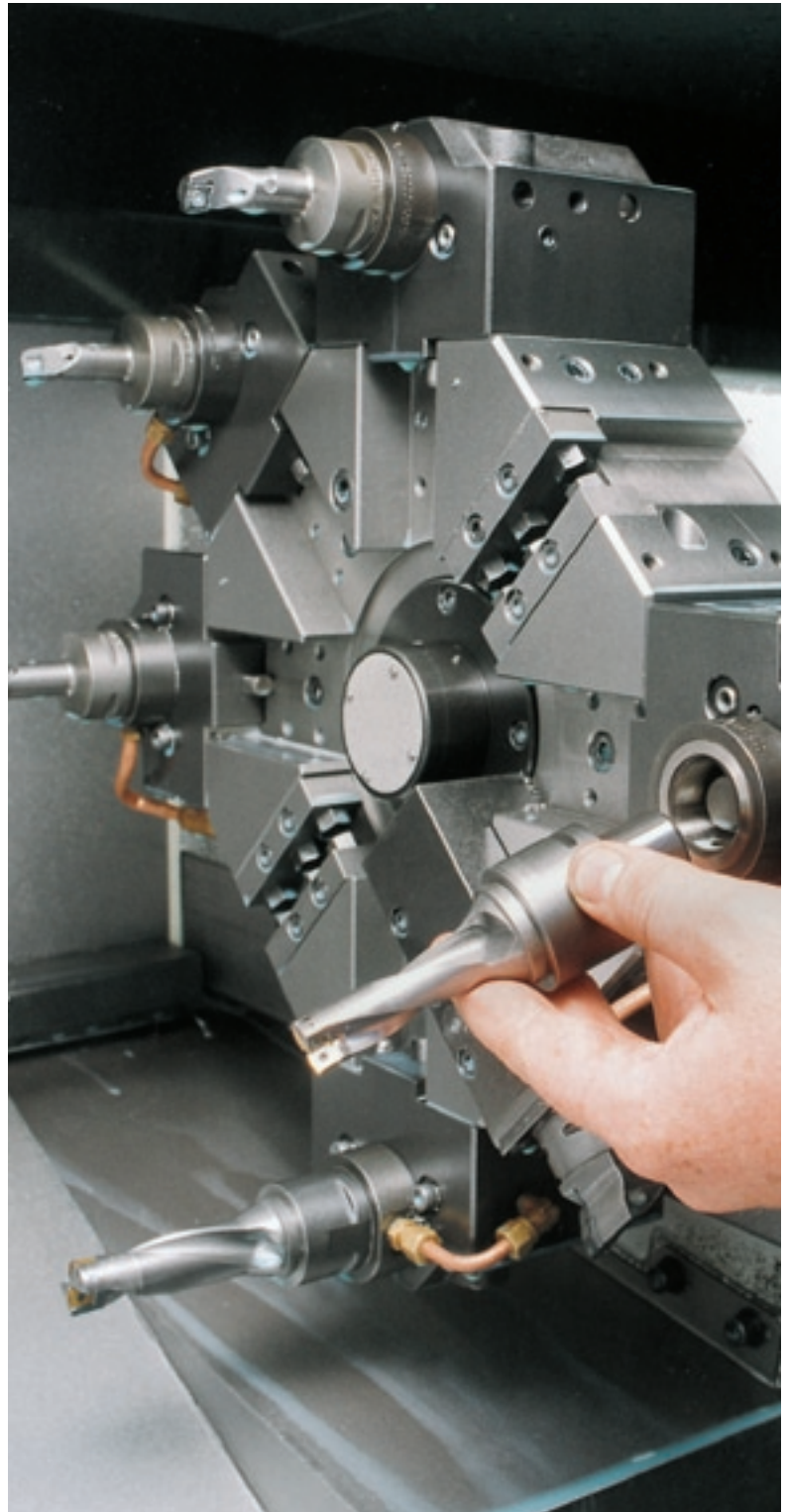
Coromant Capto integrated drill.

Selecting drills

Modern cemented carbide drilling tools are very efficient and have come a long way from out-dated, high speed twist-drills - which are still in use in many machine shops. As a consequence, the cost per machined hole has fallen dramatically. For solid cemented carbide drills, tool-life is around 20 times better than for HSS and the cutting speed capacity several times higher with the same feed rate. The basic concept of the twist-drill as such is still with us but in a very refined form. The drill-point geometries used today have vastly improved the cutting action of the conventional chisel-edge and tool materials have lifted performance and extended tool-life.

Solid carbide and brazed twist-drills drill operate at lower cutting speeds/higher feeds in relation to machine and operation while indexable insert drills use high cutting speeds/low feeds.

The **modern cemented carbide twist-drills** are application orienated towards above all two directions:

- precision holes, giving closer tolerances and surface finishes than indexable insert drills

- smaller diameter holes, where the indexable inserts drills are not a practical solution.

The **solid cemented carbide twist-drill** range of Coromant Delta-C covers diameters from 1.5 to 20 mm.

The **brazed cemented carbide twist-drill** Coromant Delta covers diameters from 9.5 to 30.4 mm.

Hole tolerances for these drills can be within IT8 and finishes within Ra 1 micron depending upon drill length, tool holding and conditions. The drill shank tolerance is h6. Cemented carbide grade options are available for all materials, including TwinGrade compound grade for stainless steel drilling. Two different grades are sintered together to provide high speed capability for the periphery and low speed capability for the centre of the tool.

Thanks to the high bending stiffness of cemented carbide, it is possible to apply tool lengths of 8 times the diameter in stable conditions (Tailor Made) and 12 to 14 times as specials.

The **cemented carbide indexable insert drill** provides high machining productivity, versatility and long, reliable tool-life. Today's drills are not just fast roughing drills. They are capable of making holes even more rapidly than the first generations of these drills but they are also capable of finishing holes to a better level and keeping within closer tolerances than previously, not just from the solid drilling operation but also from boring and the Wiper insert technology.

The **indexable insert drills CoroDrill 880. Coromant U-drill and T-Max U** (including the trepanning tool version) cover the diameter range 12 to 110 mm as standard.

The achievable hole tolerances with the new CoroDrill 880 drill have been almost halved to + 0.25 mm and with a moderate feed, the surface finish possible is Ra 0.5 micron.

Production economy has been improved considerably with the new CoroDrill 880 where penetration rates are up to twice as high.

Cemented carbide drills.

Coromant Delta C

Coromant Delta

CoroDrill 880

SANDVIK
Coromant

Drill diameter points to type of drill

One of the first things to be established when choosing a hole-making tool is whether an indexable insert drill or a regrindable drill should be used. The diameter of the hole is the first factor. Indexable insert drills cannot be used for small diameter holes (smaller than 12 mm) so these applications need solid or brazed cemented carbide drills.

Small-diameter holes
Solid cemented carbide drills, such as the CoroDrill Delta C, are available in different versions, covering a diameter range from 0.3 to 20 mm.

Where it is possible to machine at high spindle speeds the properties of cemented carbide should be utilized in order to achieve increased productivity. When the stability of the set-up is really poor, to the extent that it puts the relaibility of the carbide drill at risk, a high speed steel drill can be an alternative choice.

When the diameter of the hole is within the range covered by both CoroDrill Delta C and Coromant Delta, the latter – a brazed carbide drill – may often be the best choice. Coromant Delta offers closer tolerances with respect to both size and surface finish, low cutting forces and high cutting data in the ISO K area.

Medium-sized hole diameters
The diameter range designated as medium sized hole diameters is the range in which indexable insert drills and brazed carbide drills (Coromant Delta) overlap.

When close tolerances are required, and/or the hole depth restricts the use of indexable insert drills, then Coromant Delta is usually the best choice. With the introduction of CoroDrill 880, however, the borderline has shifted with indexable insert drills now having the capability of a finishing tool.

When the initial penetration surface is not flat, or the hole is predrilled or cross drilling has to be undertaken, then indexable insert drills are often the only option. These will provide the lowest cost per machined component, since there are inserts that can be changed and no regrinds. This cost advantage should be particularly noted when machining large volumes of components.
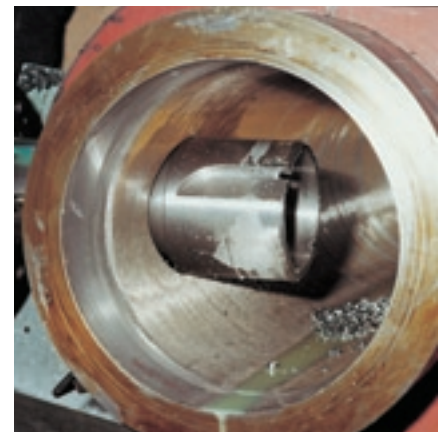
Large-diameter holes
Only indexable insert drills are available for what is designated here as large diameter holes and the choice of tool is primarily concerned with choosing the insert geometry and grade. When the machine power is limited, trepanning drills are used instead of solid drills.
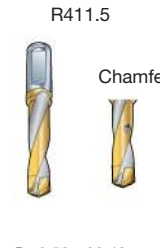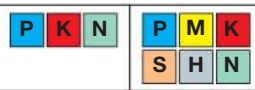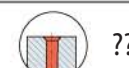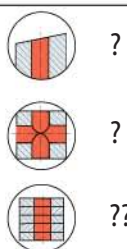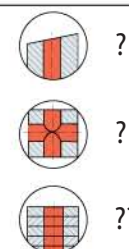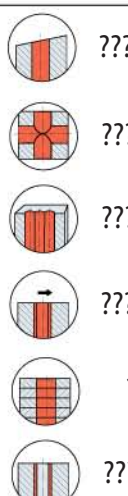
The combination of insert geometry/grade is established between the peripheral insert and central insert to provide optimal performance.

Very large diameter indexable insert drills with several cartridges have peripheral inserts, internal peripheral inserts, centre inserts and internal centre inserts.


Solid carbide drills to indexable insert trepanning tools cover hole diameters from 0.3 to 110 mm for short hole drilling.

	CoroDrill Delta C			Step / chamfer	Coromant Delta	CoroDrill 880
	R840		R850	R841	R411.5 Chamfer	
	D_c 1.5 – 20.00 mm		D_c 5.00 – 14.00 mm	D_c 3.00 – 16.00 mm	D_c 9.50 – 30.40 mm	D_c 14 – 29.5 mm
Drill depth	**2 – 5 × D_c**		**2 – 7 × D_c**	**2 – 3 × D_c**	**3.5 – 5 × D_c**	**2 – 4 × D_c**
Material	P K N	P M K / S H N	N	P M K / S H N	P M K / H N	P M K / S H N
Hole tolerance	IT8-10			IT8-10	IT8-10	
Surface finish Ra	1–2 µm			1–2 µm	1–4 µm	1–5 µm
General drilling	???			???	???	???
Step & chamfer				??? / ???	??	
Versatility	? / ? / ??				? / ? / ??	??? / ??? / ??? / ??? / ? / ???

*) By presetting.

Material

P	Steel	??? = Very good	Surface with angle	Radial adjustment
M	Stainless steel		Cross hole	Stackdrilling
K	Cast iron	?? = Good		
N	Aluminium		Plunge drilling	Trepanning
S	Super alloys	? = Fair		
H	Hardened steel			

SANDVIK
Coromant

Coromant U	Coromant U Step & chamfer	T-MAX U stackdrill	T-MAX U ≥60 mm Solid	T-MAX U ≥60 mm Trepanning	Plunge drill
R416.2	R416.21	R416.01	R416.9	R416.7	R416.22
D_c 12.7 – 58 mm	$D_{c1}/D_{c2}/D_{c3}$ 12.7 – 58.9 mm	D_c 27 – 59 mm	D_c 60 – 80 mm	D_c 60 – 110 mm	D_c 12.7 – 35 mm
2 – 5 × D_c	**≤2.3 × D_c**	**2.5 × D_c**	**2.5 × D_c**	**2.5 × D_c**	**4 × D_c**
P M K S H N	P M K S H N	P M K S H N	P M K S H N	P M K S H N	P M K S H N
IT13 IT11 *)	IT13 IT11 *)	±0.2	±0.2	±0.2	±0.20
1–5 µm	1–5 µm	2–7 µm	2–7 µm	2–7 µm	1–5 µm
●●●	●●●	●●	●●●	(icon) ●●●	
	(icon) ●●● (icon) ●●●				
(icon) ●●● (icon) ●●● (icon) ●●● (icon) ●●● (icon) ● (icon) ●●●		(icon) ●●●	(icon) ●●● (icon) ●●● (icon) ●●● (icon) ●●●	(icon) ●●●	(icon) ●● (icon) ●● (icon) ●●● (icon) ●● (icon) ● (icon) ●●

*) By presetting.

Material

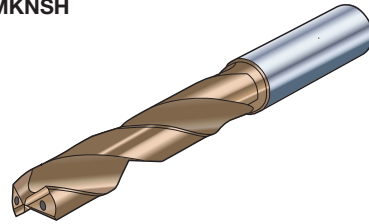|			
P Steel	●●● = Very good	(icon) Chamfer drilling	
M Stainless steel			
K Cast iron	●● = Good	(icon) Step drilling	
N Aluminium			
S Super alloys	● = Fair	(icon) Boring	
H Hardened steel			

Solid and Brazed cemented carbide drills

CoroDrill Delta-C drill R840 GC1220   PMKNSH
- Diameter range 0.5 – 20.00 mm
- Drill depth 2-7 x D
- Cyl./WN shanks
- First choice in general drilling
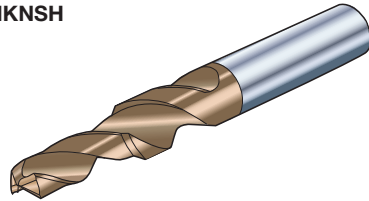
Tailor made options

CoroDrill Delta-C chamfer drill R841 GC 1220   PMKNSH
- Diameter range 3.35 – 17.50 mm
- Drill depth 2-3 x D
- Cyl. Shank
- Drill & chamfer
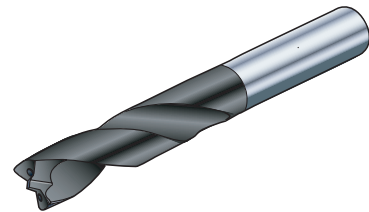
Tailor made options

CoroDrill Delta-C drill R850 N20D   N
- Diameter range 5.0 – 14.00 mm
- Drill depth 2-7 x D
- Cylindrical shank
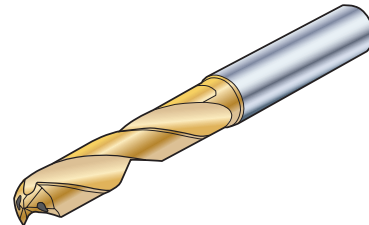- Unique geometry specially designed for drilling Aluminium

Tailor Made options

CoroDrill Delta-C drill R842 GC1210   K
- Diameter range 3.0 – 16.0 mm
- Drill depth 5 x D
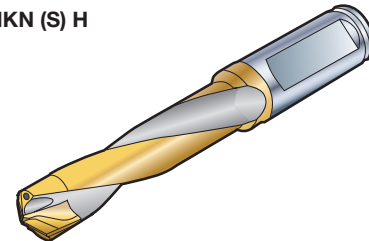- Cylindrical shank

Coromant Delta drill R411.5 P20/K20   PMKN (S) H
- Diameter range 9.50 – 30.40 mm
- Drill depth 2-5 x D
- Cylindrical with flat/CWN shanks
- Superior hole tolerance and surface finish
- Suitable for unstable conditions

Tailor made options
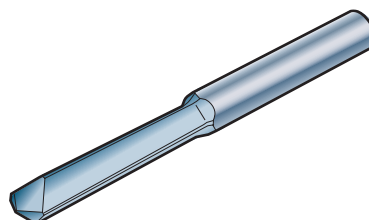
Hard Cut drill HC 2...6   H
- Diameter range 2.0 – 6.00 mm
- Drill depth 5 x D
- Cylindrical shank
- For removal of broken taps or drilling in difficult materials

SANDVIK
Coromant

Indexable insert drills

CoroDrill 880
• Diameter range 20 – 29.5 mm
• Drill depth 2 – 4 x Dc
• Different shank types

Coromant U drill R416.2
• Diameter range 12.7 – 58 mm
• Drill depth 2 – 4 x diameter
• Different shank types

Coromant U Socket head cap screw drill
• Standard diameter for screw sizes M12. M14. M16 and M20
• Drill depth 2 x D
• Cylindrical shank with flat (ISO 9766)

Coromant U drill, step and chamfer
• Available as Tailor Made
• Diameter range 12.7 – 57 mm
• Three tools in one
• Different shank types

Coromant U Plunge drill
• Suitable for rough opening of deeper cavities
• Diameter 12.7 – 35 mm
• Drill depth 4 x D
• Cylindrical shank with flat (ISO 9766)
• Engineered special options diameter
  range 12.7 – 58 mm, 2 – 6 xD

T-Max U – Left hand drill
• Diameter range 17.5 – 58 mm
• Drill depth 2.5 x diameter
• Coromant Whistle Notch shank

T-Max U stack drill
• Problem solver for drilling stacked components
• Diameter range 27 – 59 mm
• Drill depth 2.5 x diameter
• Coromant Whistle Notch shank

T-Max U – ≥ 60 mm drill
• Exchangeable cartridges
• Diameter range 60 – 80 mm
• Drill depth 2.5 x diameter

T-Max U trepanning drill
• Suitable when machine power is a limitation for solid drilling
• Exchangeable cartridges
• Diameter range 60 – 110 mm, drill diameter over 110 mm
  available on request
• Drill depth 2.5 x diameter
• Special inner cartridge can be used for drilling stacked
  components
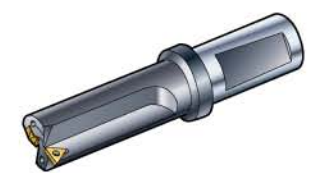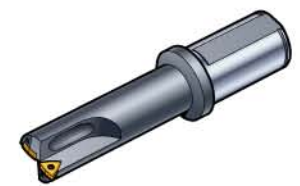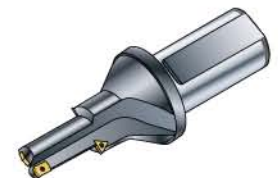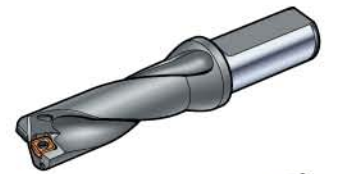
A
B
C
D
E
F
G
H

Application of drills

Solid cemented carbide twist-drills

CoroDrill Delta-C

R840: first choice for general drilling (1.5 – 20 mm diameters)

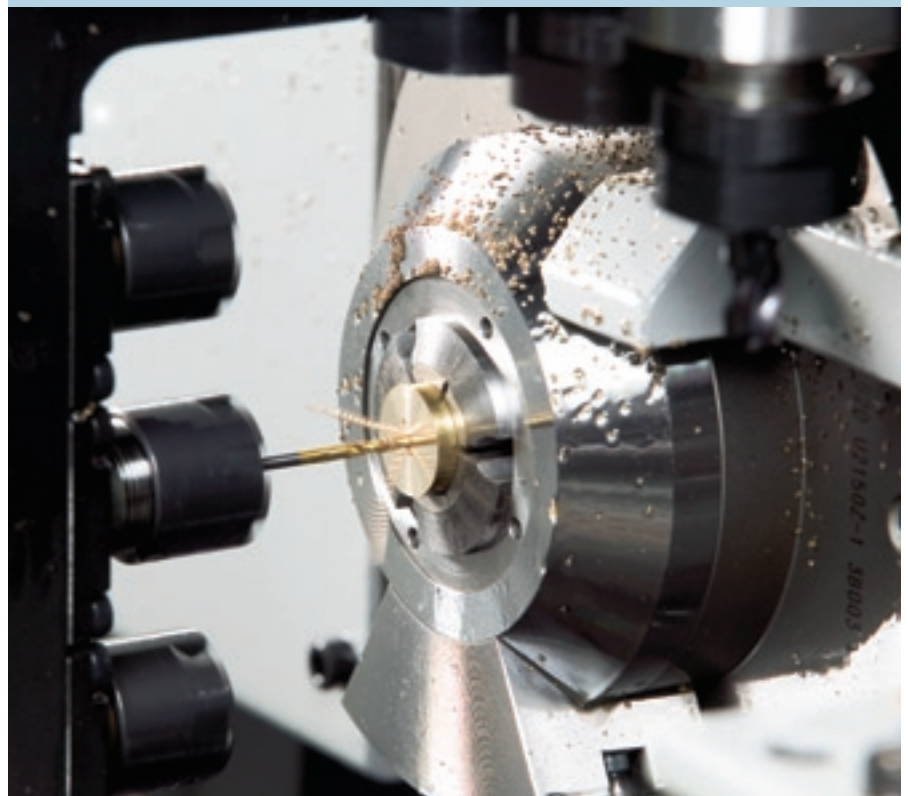**R850: for drilling in aluminium (5 – 14 mm diameters)**

R841: for step and chamfer options (3 – 16 mm diameters)

Hole depths: up to 7 times the drill diameter, depending on type and application

Workpiece materials: all types (R850 for Al)

Hole tolerance achievable: up to IT8

Surface finish achievable: up to Ra 1 micron

Recommendations for successful drilling
– CoroDrill Delta-C

Maximum stability throughout the entire system of spindle, tool holding and drill is essential. Instability puts heavy demands on the rigidity of the entire tool system. Solid carbide is the stiffest possible tool material and, more than any other type of drill, is capable of high speed production of high-quality holes. However, when machining conditions are unstable or there are tough demands on the tool, precautions should be taken or an alternative considered. To fully make use of Coromant Delta-C drill capabilities, the machine tool should be rigid, in good condition and work piece clamping should be secure.

Vibration, even at a very low frequency, has a negative effect on tool-life and production security, in that cutting edges may develop chipping rather than flank wear and thus generate poor finish and rapid breakdown. Good quality torque transmision and coolant supply are also success factors.

A rotating drill is preferred. However, if using the drills stationary, such as on a lathe, the centre of the drill should be checked to make sure it is aligned with the centre of the spindle.

In the case of application on special-purpose machines the use of bushings is not recommend.

Minimum tool run-out is essential in drilling. One of the main criteria for successful use of solid carbide drills is lowest possible run-out. It is recommended not to exceed a maximum TIR (Total Indicator Readout) of 20 microns for the drill and chuck in order to achieve the possible hole tolerance within down to IT8 and also to achieve the best tool-life.

The nominal run–out of the drill, in relation to the shank (measured in a V-block), should not exceed 0.015 mm for the total length of the drill.

The smallest total run–out is provided by the CoroGrip power chuck with a Coromant Capto coupling and also with shrink fit holding tools.

TIR
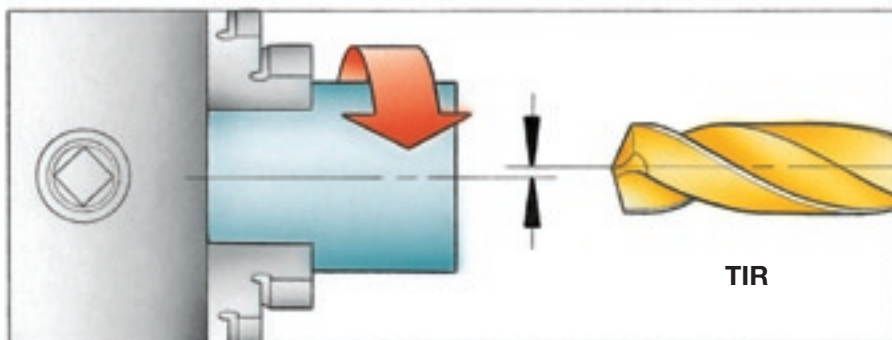
Good tool holding is the basis for good performance in drilling.

This catalogue has been split into smaller parts
to enhance downloading speeds.

If you want to view the next page
please click **HERE!**