

DRILLING

Introduction E 2

APPLICATIONS

Getting started E 3

General drilling E 10
Step / chamfer drilling E 24
Dedicated methods E 30
Trouble shooting E 44

PRODUCTS

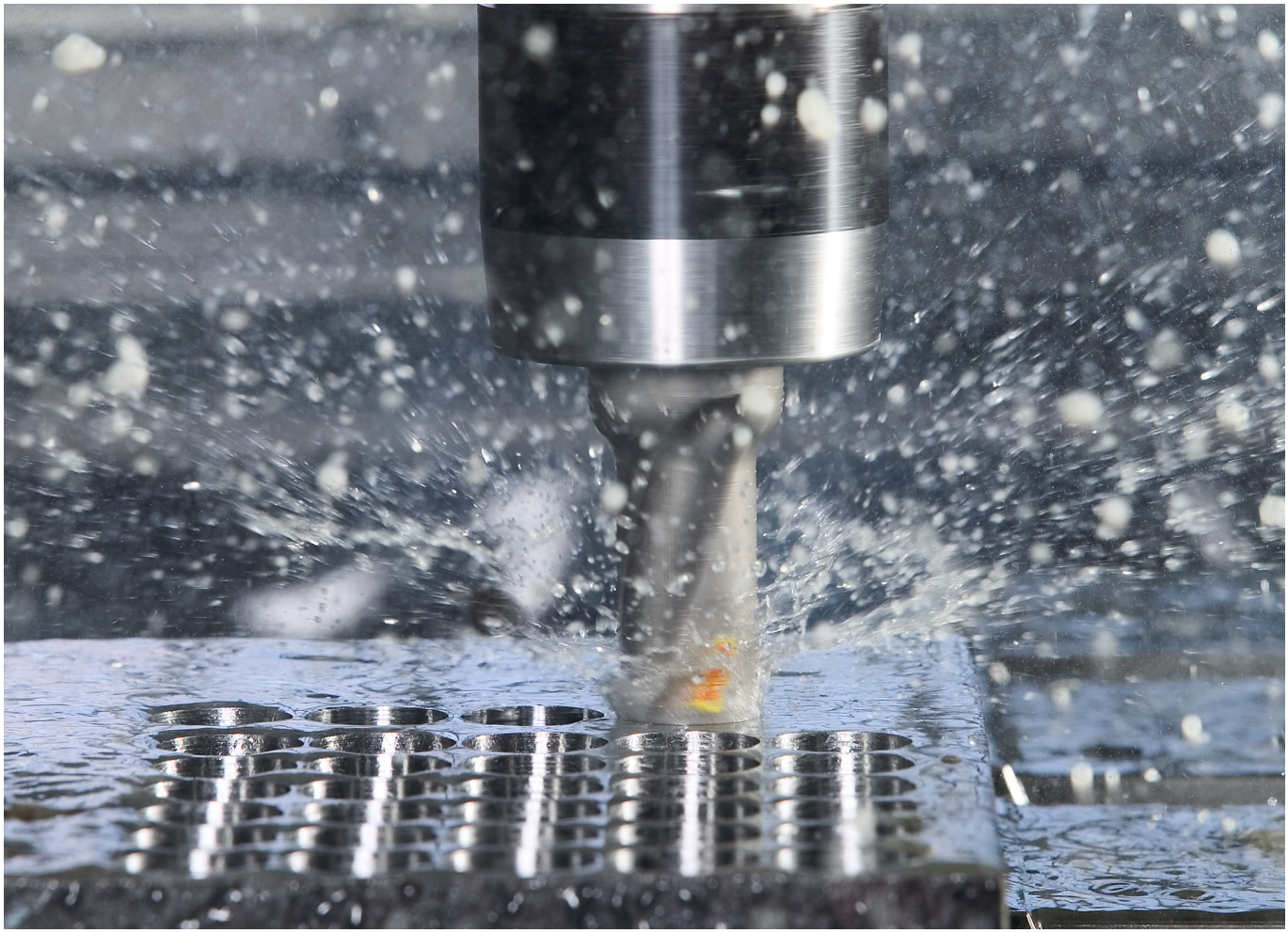
Indexable insert drills

CoroDrill® 880 E 50
CoroDrill® 805 E 54
Other drills E 55

Solid and brazed carbide drills

CoroDrill Delta-C® E 56
CoroDrill® 854 and CoroDrill® 856 E 60
Coromant Delta® E 62

Grade information E 66



Introduction

Most holes are made with drilling tools – solid carbide, indexable insert or brazed carbide, because drilling is the most productive method to make a hole.

High technology CoroDrill tools allow drilling to be carried out in one operation, without first machining centre or pilot holes. Hole quality is excellent; subsequent finishing is often unnecessary.

CoroDrill 880 is the most widely used indexable insert drill in the world, employed in a wide range of applications.

CoroDrill Delta-C, solid carbide drills, have all-round capability but are also optimized for specific materials and applications.

Trends

Machines and machining methods

- Higher spindle speeds – justify the use of solid carbide drills vs HSS-drills.
- Multi-task machining and advanced NC-controlled systems – use the CoroDrill 880 for more than conventional drilling. Drill deep holes in one set-up using CoroDrill 805 and DoroDrill Delta C.
- High pressure coolant – improves chip evacuation and tool life.

Components and materials

Environemental concerns increase the demand for lighter and stronger components for extended use in corrosive environments. Drilling more high alloyed, high-strength and non-corrosive materials means that more optimized drills and inserts are also required.

Getting started

Hole making methods

Drilling

General drilling – Sandvik Coromant has a wide choice of drills covering diameters from 0.30 to 110 mm and hole depths of up to 15 x D_c. See page E 10.
Step/chamfer drilling – Made with optimized step/chamfer drills or by programming a tool path using a standard tool. See page E 24.
Dedicated methods – Radial adjustment, boring, circular ramping, plunge drilling, trepanning, stack drilling. See page E 30.

Boring and reaming

Boring is defined as a method for enlarging or improving the quality of an existing hole. Both roughing and finishing tools are available in a wide diameter range. The multi-edge reamer is a highly productive tool for precision holes. See Boring, Chapter F.

Milling

A milling cutter with helical or circular milling can be used instead of drills or boring tools. The method is less productive but can be a good alternative when:

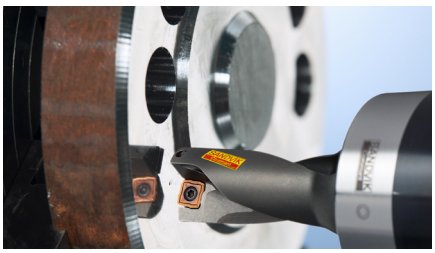
- machine power is limited and/or no coolant is available
- chip breaking/evacuation is difficult to obtain by drilling
- an absolutely flat bottom hole is required
- there is limited space in the tool magazine

See Milling, Chapter D.

Deep hole drilling

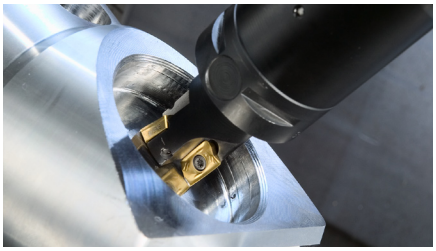
Sandvik Coromant has a wide programme of STS and Ejector drills and gun drills for drill depths up to approx. 150 x D_c. See catalogue and application guide for deep hole drilling, C-1202:1.

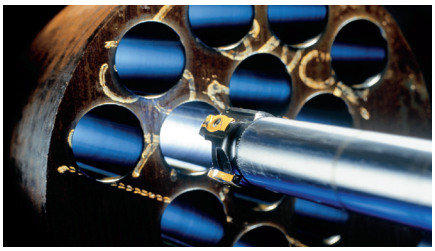




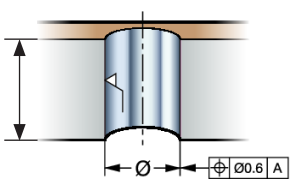




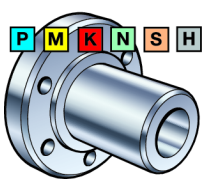




Choice of method
Three different areas should be considered to determine the best method and tooling solution.

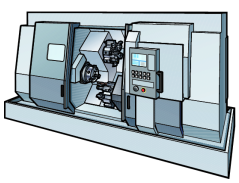


1. Hole dimensions and quality





2. Component material, shape and quantity



3. Machine parameters

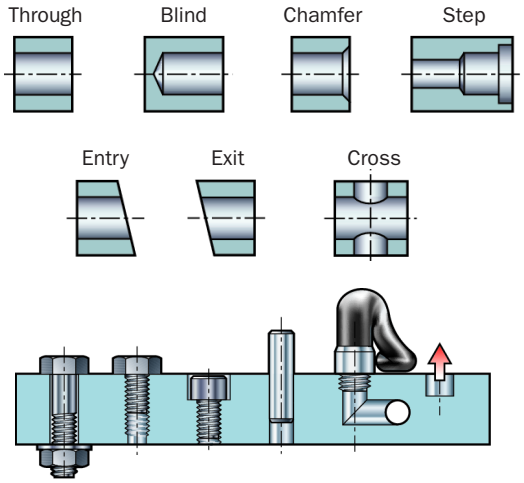
Initial considerations

1. The hole

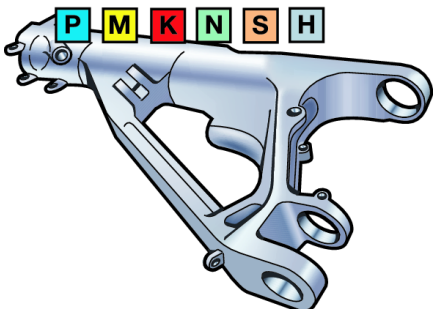
Start with the hole. The three most basic parameters are:

- diameter
- depth
- quality (tolerance, surface finish, straightness)

The hole type, and the required precision affect tool choice. Drilling can be affected by irregular or angled entry/exit surfaces and by cross holes. See page E 20.



Common hole types: Bolt, precision, coolant etc.



2. The component

After analysing the hole it is time to look at the component:

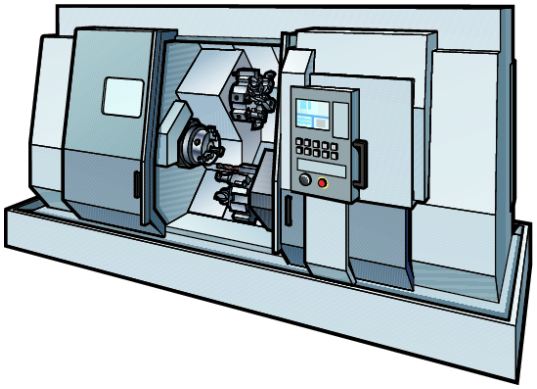
- Does the material have good chip breaking qualities?
- Is the component stable or are there thin sections that can cause vibrations?
- Is a tool extension needed to reach the surface where the hole shall be drilled?
- Can the component be fixed in place? What stability problems need to be considered?
- Is the component rotation symmetric around the hole, i.e. can the hole be machined with a non-rotating drill?

Batch size – mass production of holes, which justify an optimized, special tool to maximize productivity, or a single hole?

3. The machine

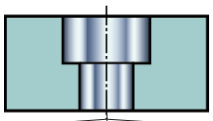
Some important machine considerations for holemaking:

- Stability, power and torque, especially for larger drills
- Chip evacuation is improved by:
  - a horizontal spindle
  - an internal coolant supply
  - a non-rotating drill
- Is spindle speed (rpm) fast enough for small diameters?
- Is coolant volume sufficient for large diameter drills?
- Is coolant pressure high enough for small diameter drills?

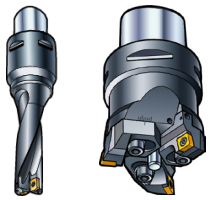


Choice of method – example

The hole to be produced:



Drilling & Boring



Advantages

- Simple standard tools
- Relatively flexible

Disadvantages

- Two tools, adaptors and basic holders
- Requires two tool positions

The standard option

Step drilling



Advantages

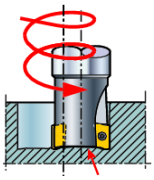
- Simple Tailor Made tools
- Fast way to make a hole

Disadvantages

- Requires more power and stability
- Less flexibility

High productivity, long series production

Milling, Circular ramping



Advantages

- Simple standard tools
- Very flexible
- Low cutting forces

Disadvantages

- Longer cycle times

Flexibility, short series production

Choice of drill type

Indexable insert drill, CoroDrill 880, should always be considered as the first choice. It normally produces holes at the lowest cost. It is a versatile tool that can do more than a conventional drill. The limitations vs solid carbide drills are hole tolerance and hole depths.

Typical applications

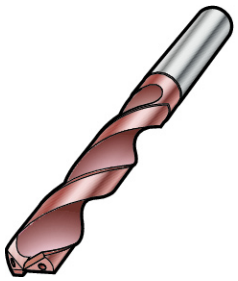
- Medium and large diameter holes
- Medium tolerance demands
- Blind holes requiring a ”flat” bottom
- Plunge drilling or boring operations



Solid carbide drill, CoroDrill Delta-C, operates at lower cutting speeds, but with a higher feed per revolution in comparison with indexable drills. The main advantage vs an indexable insert drill is that a closer hole tolerance can be achieved. Regrindable.

Typical applications

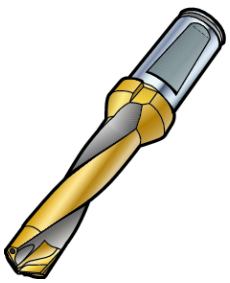
- Small diameter
- Close or precision tolerance holes
- Short to relatively deep holes



Braze-tip, cemented carbide drill, Coromant Delta, is a complementary choice with the CoroDrill Delta-C. Regrindable.

Typical applications

Complement to solid carbide for larger diameters or when process stability is poor – the steel part of the drill provides toughness

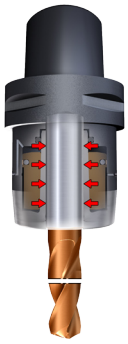


Tool holding

Productivity is not only influenced by grade and geometry, but also by the tool holding and the ability to clamp securely and accurately. For best stability and hole quality, use Coromant Capto, CoroGrip or HydroGrip holding tools. Always use the shortest possible drill and overhang.

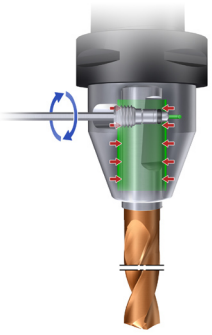
Coromant Capto is the only modular tooling system designed for all metal cutting operations, including all hole making methods. The same cutting tools and adaptors can be used in different applications and machines. This makes it possible to standardize, using one tooling system for the entire machine shop.

Precision chucks suitable for CoroDrill Delta-C are CoroGrip and HydroGrip.
See Tool holding/Machines, Chapter G.



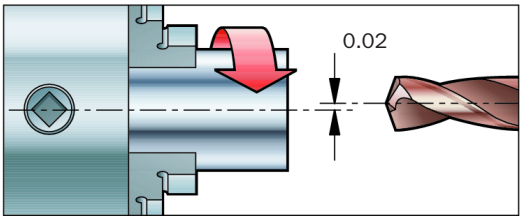
CoroGrip®



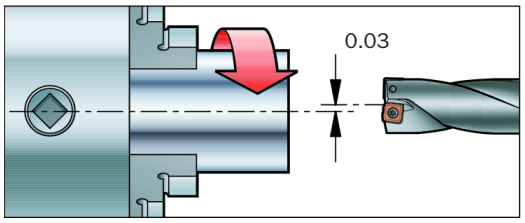


HydroGrip®

CoroDrill Delta C



CoroDrill 880



Drill run-out and alignment are very important for sucessful drilling.

Tool run-out

Minimum tool run-out is essential for successful drilling.

The run-out should not exceed the values in the figures at the left. Alignment must be parallel for:

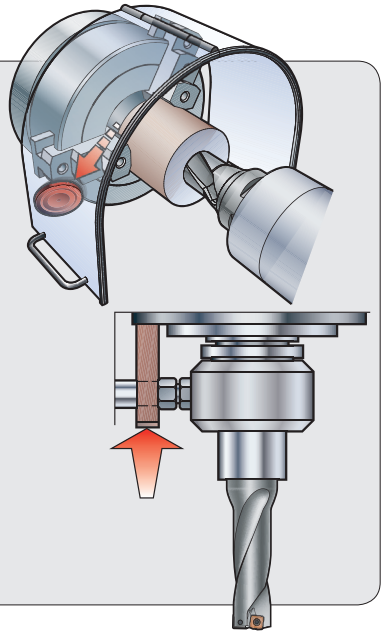
- close hole tolerance and straightness
- good surface finish
- long and consistent tool life

See Non-rotating drilling, page E 42.



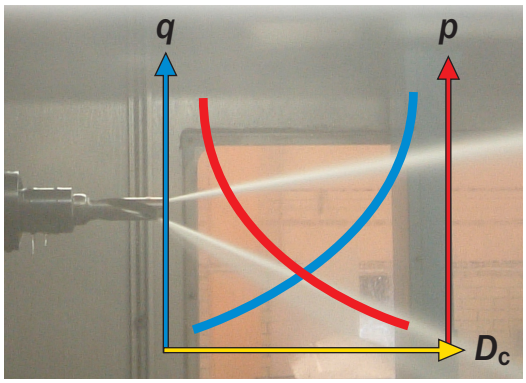
Safety precautions – danger points

When drilling through holes with an indexable insert drill, CoroDrill 880, a disc is formed, which can be ejected at high speed. Recommendation – enclose chuck or machine.



A rotation stop must be used for rotating drills. Chips in the coolant may cause slit seats to seize, housing to rotate, and supply tubing to be pulled around the housing; all of which can cause a serious accident. If a rotating connector has not been used for a while, check that holder rotates in the housing before machine spindle is started.

Cutting fluid



Pressure-diameter relationship in cutting fluid supply (pressure in red, diameter in yellow, volume in blue)

Chip evacuation, cooling and providing lubrication between the material and the tool are the primary functions of cutting fluid. All affect hole quality and tool life.

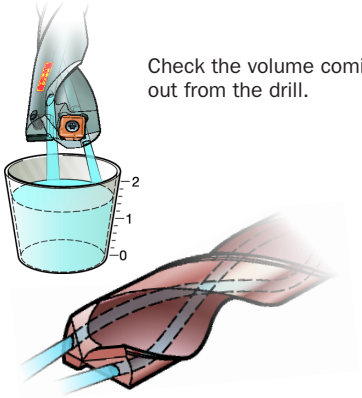
The minimum volume should be measured at the drill point, and can be checked using a stop-watch and a bucket.

The cubic capacity of the coolant tank should be 5-10 times greater than the volume of coolant that the pump supplies per minute. Coolant pressure usually falls between the pump and the drill. To check if the pressure is sufficient for a horizontal drill, the flow of coolant coming out of the drill should not have any downward drop for at least 30 cm.

- Soluble oil (emulsion) should always be used with EP (extreme pressure) additives, and the mixture of oil and water should be between 5-15% for best tool life. Drilling in stainless and heat resistant alloy materials benefits from a higher percentage mixture.
- High pressure coolant (~70 bar) improves chip evacuation and tool life in long-chipping materials like stainless steels.
- Neat oil improves lubrication and is beneficial when drilling stainless steels.
- Mist cutting fluid or minimal lubrication can be used, especially on aluminium.
- Dry drilling can be performed in short-chipping materials and at hole depths up to 3 times diameter, preferably in horizontal applications; however, tool life will be reduced.

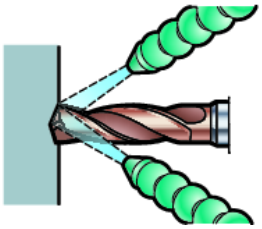
Note: Dry drilling is never recommended for stainless materials (ISO M and S) or for brazed carbide drills, i.e. Coromant Delta.

Internal coolant supply is preferred to avoid chip jamming, and should be used at hole depths 3 x D_c. External coolant supply is acceptable in short-chipping materials and can help to avoid edge built-up. The coolant nozzle must be correctly directed, see picture.



Check the volume coming out from the drill.

Internal coolant supply is always to be prefered.



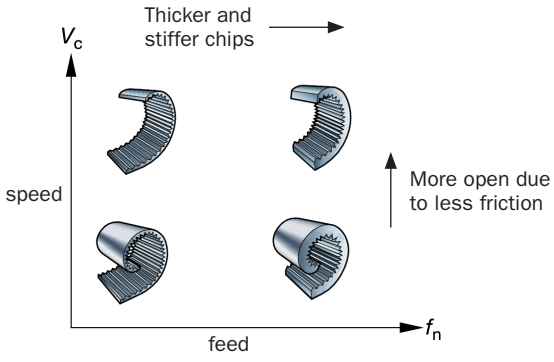
If external coolant is used, make sure the coolant is properly directed.

Chip control

Chip formation and chip evacuation is a critical issue in drilling. It depends on the workpiece material, choice of drilll/insert geometry, coolant pressure/volume, cutting data.

Chip jamming can cause radial movement of the drill and affect hole quality, drill life and reliability, or drill/insert breakages.

See page E 15.

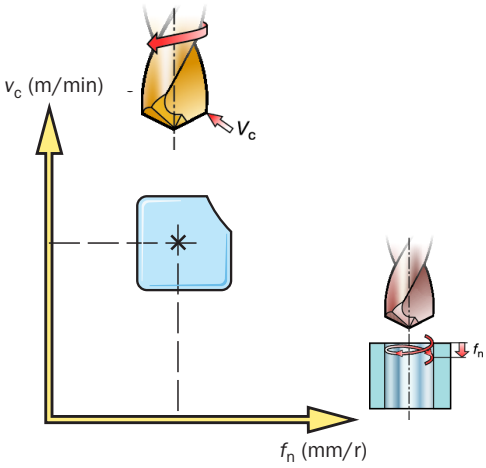


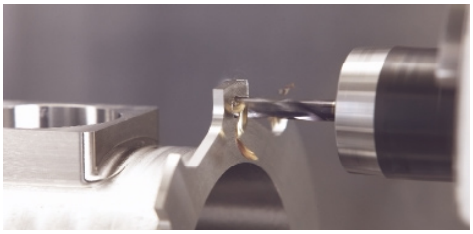
Cutting data

Effects of speed – v_c (m/min)
Cutting speed – the main factor, along with material hardness, that affects tool life and power consumption. Higher speed generates higher temperatures and increased flank wear.

- Higher speed is beneficial for chip formation in soft, long-chipping materials, i.e. low carbon steel
- Speed affects the power P_c (kW) and torque M_c (Nm)

Effects of feed rate – f_n (mm/r)
- Influences surface finish, but can also affect hole tolerance and straightness.
- Influences chip formation.
- High feed means less time spent cutting and less wear per drilled meter, but increased risk for drills/insert breakages.
- High feed affects the feed force, F_f (N), be considered when conditions are unstable.
- Feed affects power P_c (kW) and torque M_c (Nm).

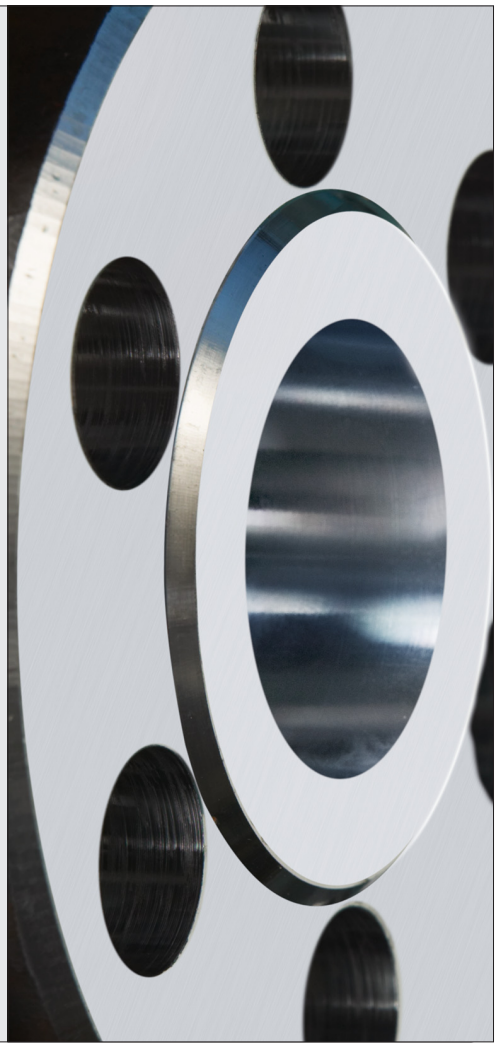




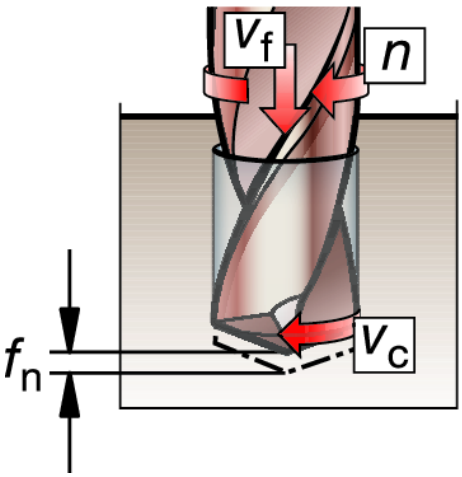
When drilling a thin/weak component, the feed/rev. (f_n) should be kept low.

How to achieve good hole quality

- **Chip evacuation**
  Make sure chip evacuation is satisfactory. Chip jamming affects hole quality and reliability/tool life. Drill/insert geometry and cutting data are crucial. See page E 15.
- **Stability, tool set-up**
  Use the shortest possible drill. Use a rigid and accurate tool holder with min. run-out. Make sure the machine spindle is in good condition and is aligned well. Rotate workpiece and drill to maintain straightness in deep holes. Ensure that component is fixed and stable. Establish correct feed rates for irregular, angular surfaces, and cross holes. See page E 20.
- **Tool life**
  Check insert wear and establish pre-determined tool life program. The most effective way to supervise drilling is by using a feed force monitor.
- **Maintenance**
  Change insert-clamping screw regularly. Clean the tip seat before changing insert. Use torque wrench and molycote. Don't exceed max. wear before regrinding solid/brazed carbide drills. See page E 65.



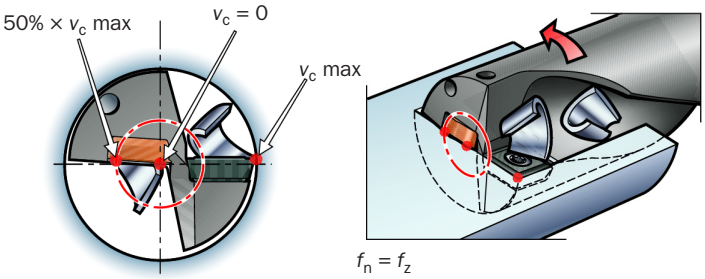
Drilling definitions



$$v_f = f_n \times n$$

$$v_c = \frac{\pi \times D_c \times n}{1\,000}$$

Productivity in drilling is strongly related to the penetration rate, v_f.

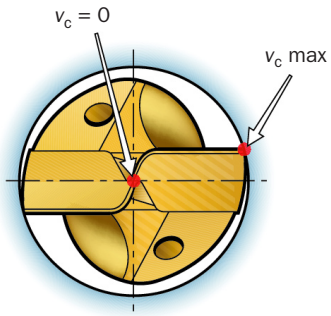


Indexable insert drill – one central and one peripheral insert

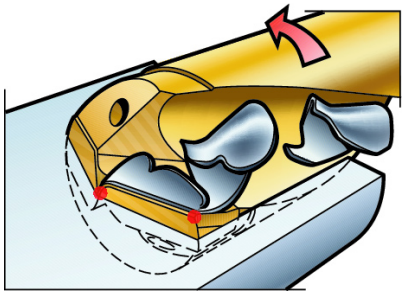
The central insert works from cutting speed zero to 50% of v_c max., and the peripheral insert from 50% v_c max. up to v_c max. The central insert forms a conical chip, while the peripheral forms a chip similar to that achieved using internal turning with a large depth of cut.

Solid and brazed carbide drills

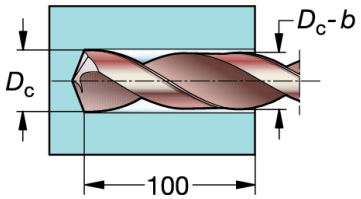
Two edges from the centre to the periphery.



50% × v_c max



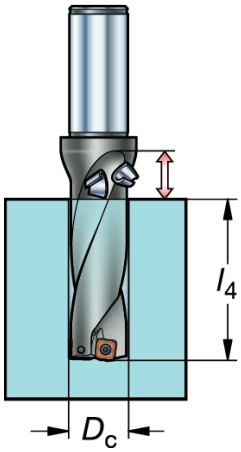
$f_n = 2 \times f_z$



Back taper

A solid or brazed carbide drill is ground slightly tapered on its outer diameter supply clearence, which prevents the drill from jamming in the hole.

Hole depth



l_4 max recommended hole depth

Calculating tool life

Tool life (TL) can be measured using distance, in meters, No. of holes, or minutes.

Theoretical example:
D_c 20 mm v_c = 200 m/min n = 3184 rpm
f_n = 0.20 mm/r, hole depth 50 mm
TL (meters): 15 meters
TL (No. of holes): 15 x 1000/50 = 300 holes
TL (min): 15 x 1000/v_f = 15 x 1000/(f_n x n)
= 15 x 1000 / (0.20 x 3184) = 23 min

The most common tool life criteria in drilling is flank wear. Tool life is dependent on:

- Cutting data
- Carbide grade and insert geometry
- Workpiece material
- Diameter (a small diameter drill travels a longer distance in a shorter time)
- Hole depths (many short holes means many entries/exits which decrease tool life).
- Stability

General drilling

Application overview

Conventional drilling

Choice of tools **E 12**

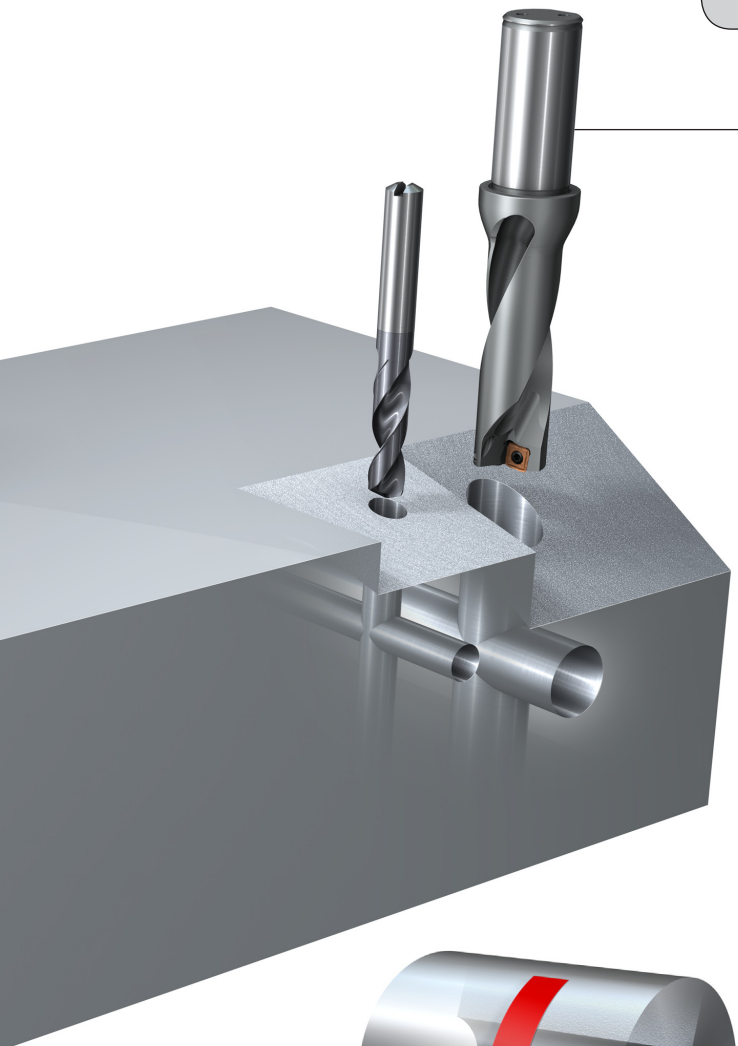
How to apply **E 15**



Irregular surface and cross hole drilling

Choice of tools **E 20**

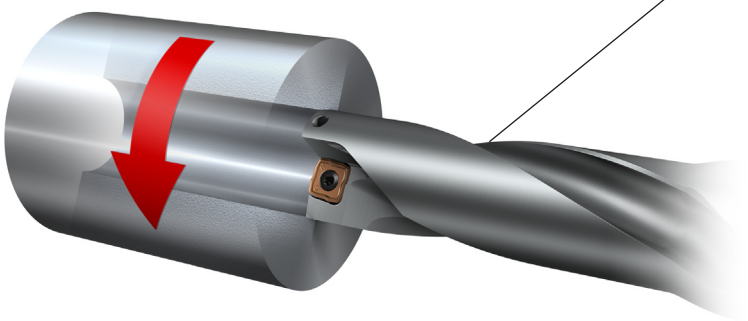
How to apply **E 21**



Non-rotating drilling

Choice of tools **E 42**

How to apply **E 42**



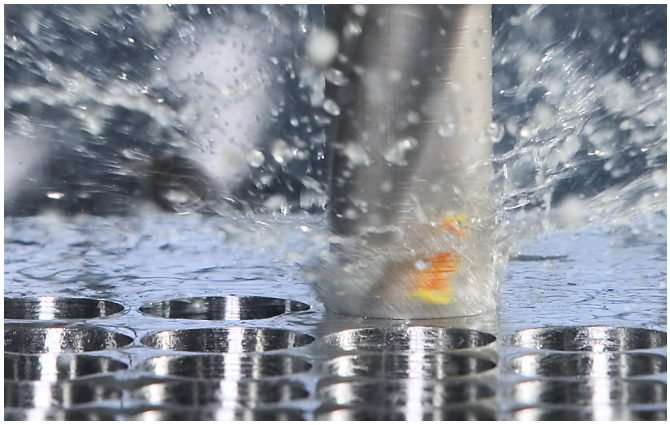
Drilling

Trouble shooting **E 44**

Conventional drilling

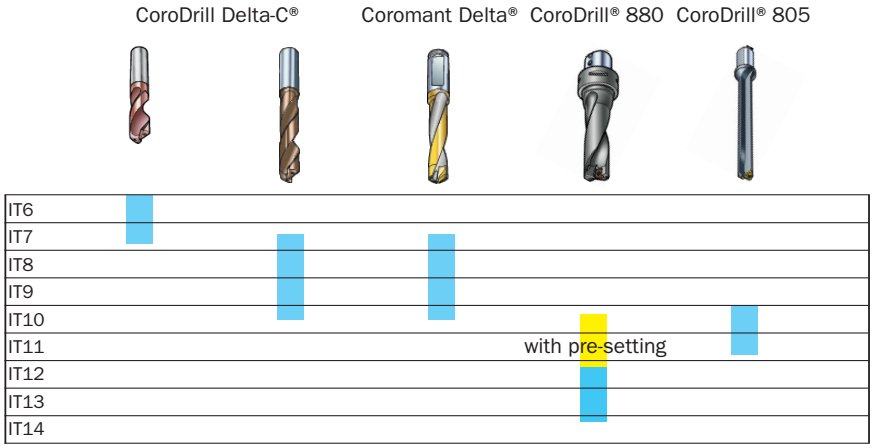
Sandvik Coromant has a large programme of drilling tools, covering diameters from 0.30 mm up to 110 mm; our specially engineered products can produce even larger diameter holes.

Making the right choice means achieving the required hole quality with a good capability at the lowest cost per drilled hole.



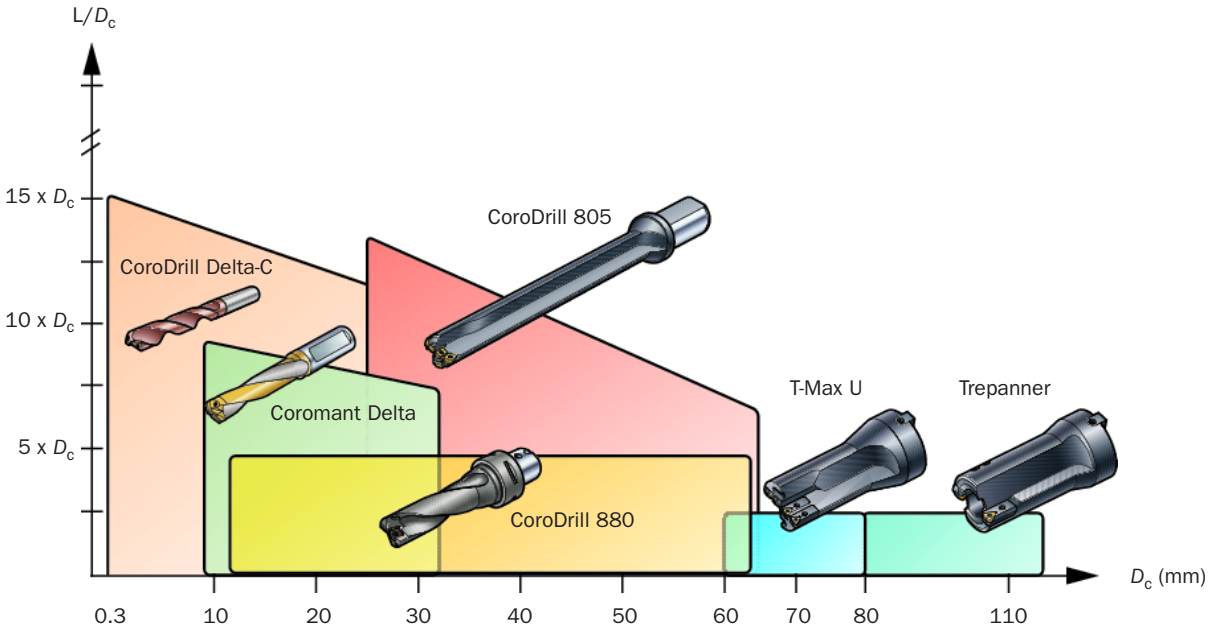
Choice of tools

Hole tolerance

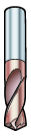


(See Information/Index, Chapter I, for IT-tolerances)

Hole depth and diameter



Small to medium diameter holes, ~0.3–30 mm

	CoroDrill Delta-C®					Coromant Delta®	CoroDrill® 880
	 R840	 R842	 R844	 R846	 R850	 R411.5	 880
Drill diameter D_c mm	0.30–20.00 (–25.00)*	3.00–16.00	8.00–18.00	3.00–12.00	5.00–14.00	9.50–30.40	12.00–63.50
Drill depth	2–7 x D_c (–15 x D_c)*	2–5 x D_c	1–1.5 x D_c	2–5 x D_c	2–7 x D_c	3.5–5 x D_c (–10 x D_c)*	2–5 x D_c
Material	P M K N S H	K	P H	S	N	P M K N H	P M K N S H
Hole tolerance	IT8–10	IT8–10	IT5–6	IT8–10	IT8–10	IT8–10	IT12–13
Surface finish R_a	1–2 µm	1–2 µm	0.5–1 µm	1–2 µm	1–2 µm	1–4 µm	1–5 µm

**Tailor Made/special*

All-round choices

CoroDrill 880 normally produces holes at the lowest cost and should always be the first choice considered.

For a closer hole tolerance, pre-setting with CoroDrill 880 can be an option. See page E 32.

CoroDrill Delta-C is the main choice for close tolerance holes and for smaller hole diameters.

Coromant Delta is a complementary choice to CoroDrill Delta-C, esp. for larger diameters (> 20 mm) or when stability is poor.

Deep holes

Note a short drill is recommended for drilling a pilot hole when hole depth is greater than 7 x D_c.

CoroDrill Delta-C
Standard program up to 7 x D_c and up to approx. 15 x D_c as an engineered product. For best chip evacuation, order drills with polished chipchannels.

Coromant Delta
Available up to approx. 10 x D_c as Tailor Made.

Dedicated choices

Cast iron – CoroDrill Delta-C R842 grade GC1210
Optimized choice for ISO-K applications
Wear resistant geometry/grade in Cast Iron

Hard steel – CoroDrill Delta-C R844, grade GC1220
First choice for ISO-P precision and hard steel applications.
IT6 tolerance possible

HRSA, Titanium – CoroDrill Delta-C R846, grade GC1220
First choice for ISO-S applications
Ni/Co-based stainless steel
Suitable for Ti and SS

Aluminium – CoroDrill Delta-C R850, grade N20D
First choice for ISO-N applications
Aluminium up to 12% Si
Suitable for copper/copper alloys

Medium to large diameter holes, ~25–110 mm

	CoroDrill® 880	T-Max® U	CoroDrill® 805	T-Max® U Trepanner	Boring	Milling
	 880	 R416.9*	 805	 R416.7*		
Drill diameter D_c mm	12.00–63.50	60.00–80.00	25.00–65.00	60.00–110.00	See Boring, Chapter F.	See Milling, Chapter D, for circular ramping with milling tools,
Drill depth	2–5 x D_c	2.5 x D_c	7–15 x D_c	2.5 x D_c		
Material	P M K N S H	P M K N S H	P M K N S H	P M K N S H		
Hole tolerance	IT12–13	IT13	IT10	IT13		
Surface finish R_a	1–5 µm	2–7 µm	2 µm	2–7 µm		

**See electronic catalogue for ordering information.*

All-round choice

CoroDrill 880 indexable insert drill
Available in larger diameters with a wide choice of insert geometries and grades for all types of materials. See page E 50.

Dedicated choices

There are three options for drilling large holes with limited machine power:

1. Use the T–Max U Trepanner. See page E 38
2. Enlarge the hole with a boring tool. See Boring, Chapter F.
3. Use helical interpolation with milling tools. See Milling, Chapter D.



Deep holes

CoroDrill 805
Available for drill depths up to approx. 15 x D_c. Also consider a tool for the pilot hole. See page E 19.

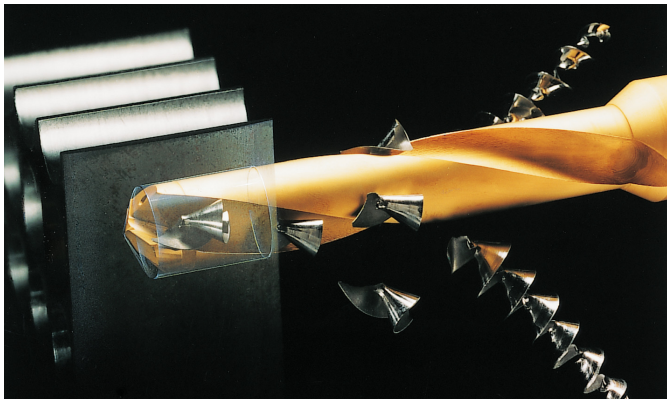


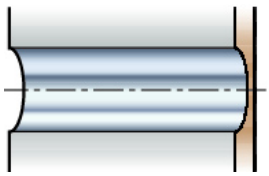
How to apply

Chip evacuation

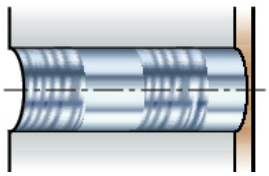
Chip formation and evacuation form the single factor in drilling that affects hole quality and the reliability of the whole process.

The chip formation is acceptable when chips can be evacuated from the drill without disturbance. The best way to identify this is to listen during drilling. A consistent sound means that chip evacuation is good, but an interrupted sound indicates chip jamming. Check feed force or power monitor. If there are irregularities, chip jamming can be the reason. Look at the chips. If they are long and bent, instead of curled, chip jamming has occured. Look at the hole. If chip jamming has occured , an uneven surface will be visible.





A hole with good chip evacuation.



A hole affected by chip jamming.

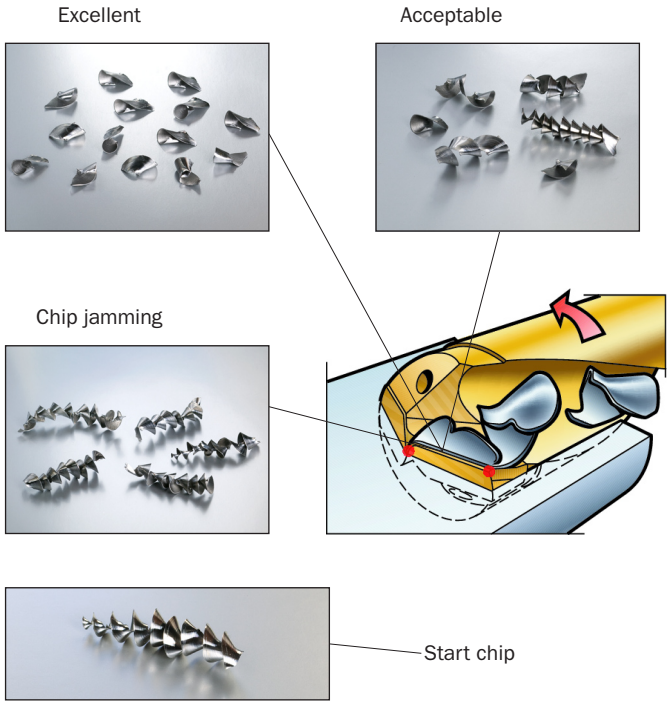
CoroDrill® 880

The central insert forms a conical chip that is easy to identify. The peripheral insert forms a chip similar to that formed by turning.



CoroDrill Delta-C® and Coromant Delta®

One chip is formed from the centre to the periphery of the edge.



Note: The start chip from entry intro the workpiece is always long and does not create any problems.

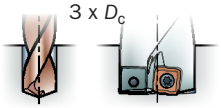
Working in different materials

P Low carbon steel

Issue: Chip formation can be a difficult issue with low carbon steels, which often are used for welded components. The lower the hardness, carbon and sulphur content of the steel, the longer the chips are that will be produced.

Recommendations
CoroDrill Delta-C: Standard geometry R840 in grade GC1220 is the first choice. If problems with chip formation occur, increase the speed, v_c, and decrease the feed, f_n (note in normal steels, the feed should be increased).
CoroDrill 880: First choice is geometry LM and grade GC4024/1044. If problems with chip formation occur, increase the speed, v_c, and decrease the feed, f_n.
Other: Use high pressure and internal coolant supply. Mixture preferably 4-7%.

Example – Low carbon steel

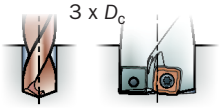
D_c	10 mm	20 mm
	3 x D_c	
		
v_c High	130	300 m/min
f_n Low	0.25	0.06 mm/r
Geometry	R840	-LM
Grade	GC1220	GC4024/1044

M Austenitic and duplex stainless steels

Issue: Austenitic, duplex, and superduplex materials can cause problems with chip evacuation.

Recommendations
CoroDrill Delta-C: Drill geometry R840 in grade GC1220 is the first choice. A complementary choice is R846, which has a larger back taper and a stronger peripheral corner due to the convex shaped edge.
CoroDrill 880: First choice for drill diameter 14.00-35.99 mm is geometry -MS and grade GC2044/1144. For other sizes use geometry -LM and grade GC4044 alt. GC4024/1044.
Geometry -GT is a complementary choice for tough conditions.
Other: Internal coolant, high pressure with a mixture of preferably 9-12% or neat oil.

Example – Austenitic stainless steel

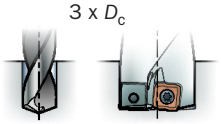
D_c	10 mm	20 mm
	3 x D_c	
		
v_c	70	180 m/min
f_n	0.20	0.10 mm/r
Geometry	R840	-LM -MS
Grade	GC1220	GC4044/1044 GC2044/1144

K CGI (Compact Graphite Iron)

Issue: CGI does not normally require extra attention. It produces larger chips than grey cast iron, but they are well broken. Cutting forces are higher, which affects tool life. Extra wear resistant grades are necessary. Corner wear is typical as in all cast irons.

Recommendations
CoroDrill Delta-C: R842 in grade GC1210 is the first choice. R840 in GC1220 is complementary.
CoroDrill 880: First choice is GR geometry in grade GC4024/1044. If problems with chip formation occur, increase the speed, v_c, and decrease the feed, f_n.
Other: Internal coolant, mixture of preferably 5-7%.

Example – CGI (Compact Graphite Iron)

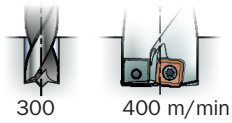
D_c	10 mm	20 mm
	3 x D_c	
		
v_c	100	150 m/min
f_n	0.25	0.18 mm/r
Geometry	R842	-GR
Grade	GC1210	GC4024/1044

N Aluminium alloys

Issue: Burr formation and chip evacuation can be a problem.

Recommendations
CoroDrill Delta-C: Use drill geometry R850 in grade N20D. The optimized geometry minimizes burr formation at hole exit and can be used with very high feeds. An economical and more productive alternative to PCD-drills for aluminium alloys with Si contents up to 12%.
CoroDrill 880: First choice is geometry LM. Grade H13A is uncoated and forms a sharper edge which minimizes burr formation. For best chip formation, use low feed and high speed.
Other: Use emulsion or mist coolant at high pressure.

Example – Aluminium

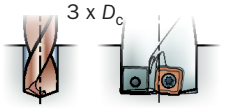
D_c	10 mm	20 mm
	3 x D_c	
v_c	300	400 m/min
f_n	0.40	0.10 mm/r
Geometry	R850	-LM
Grade	N20D	H13A

S Titanium and Heat Resistant Alloys

Issue: Thin walled components deflect due to feed force. Work hardening of hole surface affects subsequent operations. Chip evacuation can be critical.

Recommendations
CoroDrill Delta-C: Geometry R846 is specially developed for this material group. Large backtaper and a small circular land minimizes work hardening. The convex edge maximizes peripheral corner strength against notch wear, reduces the feed force and improves chip formation.
CoroDrill 880: Geometry LM and grade H13A for titanium and GC4044/1044 for other HRSA.
Other: High pressure (up to 70 bar) coolant improves performance.

Example – Waspalloy

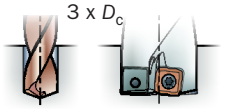
D_c	10 mm	20 mm
	3 x D_c	
v_c	25	30 m/min
f_n	0.10	0.05 mm/r
Geometry	R846	-LM
Grade	GC1220	GC4044

H Hard steels

Issue: Notch wear on the peripheral corner.

Recommendations
CoroDrill Delta-C: Standard geometry R840 in grade GC1220 can be used in steels up to 60HRc hardness. To further improve the wear resistance drill geometry R844 with a corner radius can be ordered as Tailor Made.
CoroDrill 880: Geometry GM and grade GC4024/1044 is the first choice.
Other: Emulsion with a high mixture or neat oil improves tool life. Use shortest possible drill to maximize the torque strength.

Example – Hardened steel 55 HRc

D_c	10 mm	20 mm
	3 x D_c	
v_c	20	60 m/min
f_n	0.10	0.10 mm/r
Geometry	R840	-GM
Grade	GC1220	GC4024

Cutting data
The examples shows typical values for a CoroDrill Delta-C drill in diameter 10 mm and a CoroDrill 880 in diameter 20 mm with length 3 x D_c.

Close tolerance holes

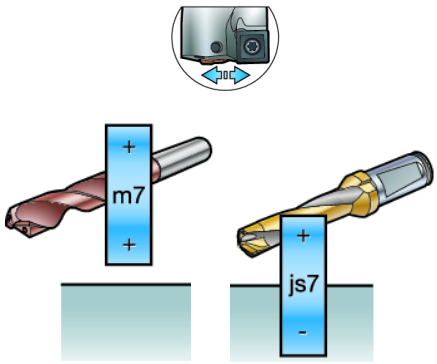
CoroDrill 880

By pre-setting the CoroDrill 880, the manufacturing tolerances can be eliminated and a closer hole tolerance can be achieved. See page E 33.

CoroDrill Delta-C and Coromant Delta

Note that the CoroDrill Delta-C is diameter ground to m-tolerance (plus-plus) acc. to DIN 6537, while the Coromant Delta is ground to a ±tolerance (js). This means that the CoroDrill Delta-C produces a slightly larger hole than the Coromant Delta drill. For precision holes (IT6) CoroDrill Delta-C type R844 can be ordered as Tailor Made. See Information/Index, Chapter I, for IT-tolerance classes.

See Boring, Chapter F, for precision holes.



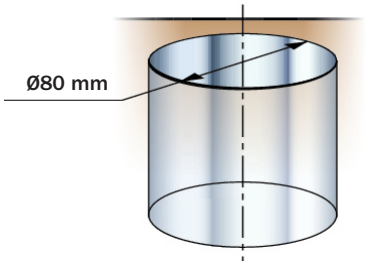
CoroDrill Delta-C and Coromant Delta

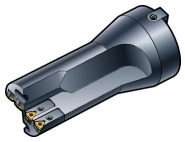
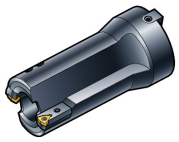
Large diameter holes

Stability of both the component and the machine is important when drilling large holes. Also, machine power and torque can be limiting factors.

The example below shows three different methods for producing a diameter 80 mm hole. A fourth method is to drill a smaller hole and then enlarge it with a boring tool.

From a productivity point of view, the drilling tools are superior: 5 times faster than milling the hole using helical interpolation. The T-Max U Trepanner can, however, only be used in through hole applications. The milling cutter has the lowest power and torque requirements by far.



Hole diameter (mm): 80 Hole tolerance (mm): Open Hole depth: 1.25 x D_c	**T-Max® U drill**	**T-Max® U Trepanner**	**CoroMill® 300**
			
Material:	CMC 02.2 Low alloyed steel	CMC 02.2 Low alloyed steel	CMC 02.2 Low alloyed steel
Tool:	R416.9-0800-25-01	R416.7-0800-25-01	R300-050Q22-12M (z=4)
Diameter, D_c (mm)	80	80	50
Cutting data			
n (rpm)	600	600	955
v_c (m/min)	150	150	150
f_n (mm/rev)	0.18	0.18	1.2 (f_z=0.30)
v_f (mm/min)	110	110	430 (v_{fm}=1150)
a_p (mm)	–	–	4.94
D_{vf} (mm)	–	–	30
Result:			
P (kW)	**30**	**14**	**6**
M_v (Nm)	**480**	**330**	**60**
Time per hole (min)	**0.93**	**0.93**	**4.66**

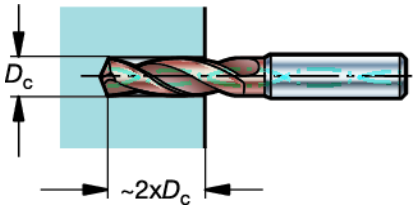
Deep holes ~8-15 x D_c

CoroDrill Delta-C®

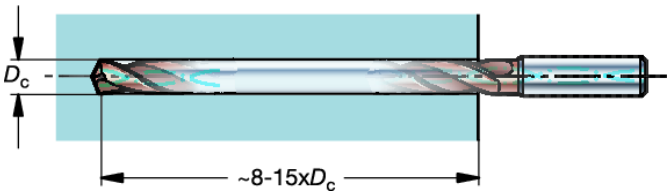
Internal coolant at a pressure of at least 20 bar. The procedure is valid for CoroDrill Delta-C and Coromant Delta:

1. Pre-drilling/pilot hole is recommended to a depth of ~2 x D_c. The pilot drill should have the same bottom profile and nominal diameter (0 to +0.02 mm) as the main hole.
2. Enter pilot hole at reduced speed (V_c) to avoid tool bending.
3. When entered, use recommended cutting data.

If chip evacuation is a problem, try peck-drilling.



1. Pilot hole



2-3. Deep hole

CoroDrill® 805

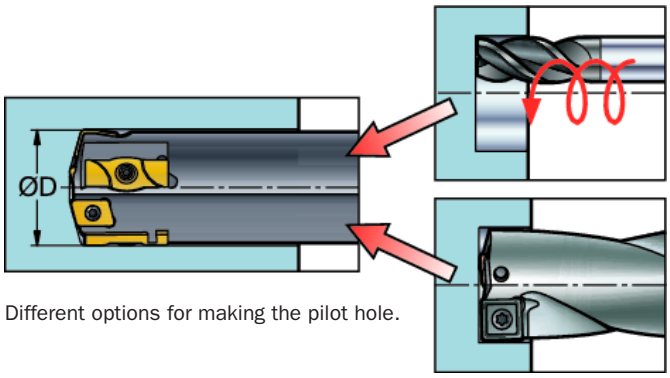
The tool set-up should be horizontal for best chip evacuation. Use neat oil or an emulsion with EP additives, preferably with a mixture of more than 8%. Pressure and volume as for a CoroDrill 880, but in a vertical set-up, the coolant volume and pressure both need to be higher. Rotation of both drill and workpiece improves holes straightness.

How to make a pilot hole

- The pilot hole diameter should be within H8-tolerance. Use helical interpolation with a CoroMill Plura endmill. If hole tolerance is not critical, use a CoroDrill 880.
- Pilot holes should be as deep as the front end of the support pads (between 12 mm and 20 mm depth).
- The bottom of the pilot hole should be as flat as possible, over 140 degrees, to avoid the intermediate insert entering before the central.

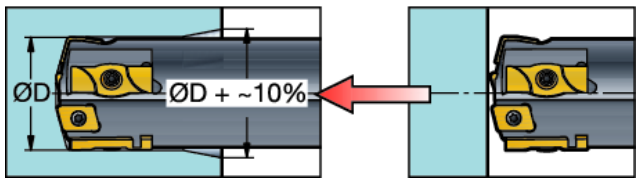
To drill with a pilot hole

Feed the CoroDrill 805 into the pilot hole at a slow rotation and with the coolant turned on. The rotation should be stopped before pulling the drill out of the hole.



Different options for making the pilot hole.

To drill without a pilot hole



The hole entrance will be approx. 10% larger (bell mouth).

Note: Drilling without a pilot hole will form a ”bell mouth” in the beginning of the hole and is only recommended in easy to machine materials, i.e. medium carbon steels and cast iron.

Start at a low speed and very low feed when entering the workpiece, or the drill will push out from the centre-line. For normal steel, use f_n = 0.02 mm/r and v_c = 45m/min until the front end of the pads are in the hole. Then the feed can be gradually increased to the normal recommendation (see page E 55), until the pads are in full support.

Irregular surface and cross hole drilling

Drilling irregular surfaces can lead to excessive, uneven forces on drill cutting edges, which leads to premature wear.

It is important to follow the guidelines and reduce feed when necessary.



Choice of tools

	CoroDrill Delta-C®	Coromant Delta®	CoroDrill® 880	CoroDrill® 805
	 R840 R842 R846 R850	 R411.5	 880	 805
Drill diameter D_c mm	3.00–20.00	9.50–30.40	12.00-63.00	25.00–65.00
Drill depth	2–7 x D_c	3.5–5 x D_c	2–5 x D_c	7–15 x D_c
Material	P M K N S H	P M K N H	P M K N S H	P M K N S H
Comment				*Cross holes only*

CoroDrill 880
Can penetrate convex, concave, inclined and irregular surfaces. In most cases, the feed rate needs to be adjusted.

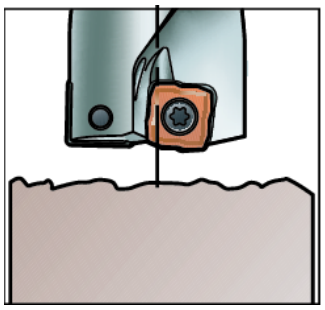
CoroDrill Delta-C
Drilling of non-flat surfaces inclined to a maximum of 10° is acceptable. However, a reduction of feed upon entry is essential to prevent drill sliding; and also upon exit to prevent wear on the circular land and to prevent drill breakage.

CoroDrill 805
Needs a pilot hole when entering into the workpiece material, i.e. irregular surfaces is not acceptable.

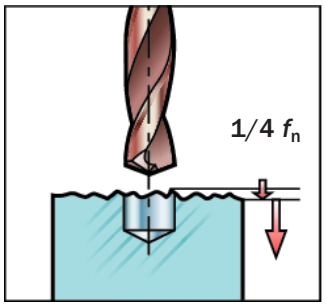
Cross hole drilling, however, can be performed, if additional support pads are applied on the drill body.

How to apply

Irregular surface

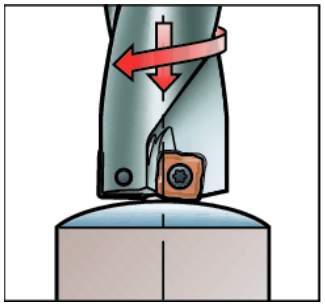


Irregular, rough surfaces can damage inserts, when penetrating or exiting the surface. Feed rate should be reduced.

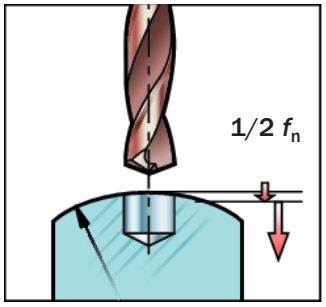


When entering, feed must be reduced to 1/4 normal rate to avoid chipping.

Convex surface

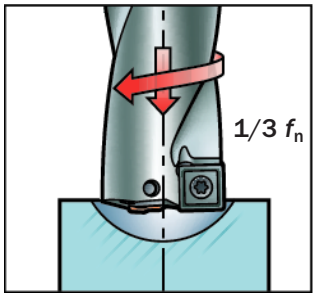


Convex surfaces are not as difficult to drill, because the centre of the drill contacts the workpiece first, giving normal torque.



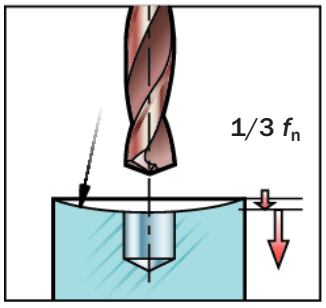
Convex surfaces are possible to drill, if the radius is > 4 times drill diameter and the hole is perpendicular to the radius. The feed should be reduced to 1/2 of normal rate when entering.

Concave surface



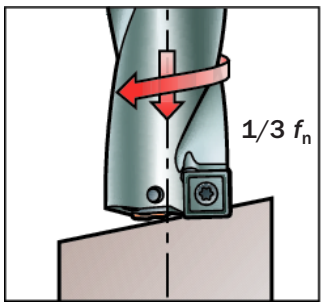
Concave surfaces cause drill engagement to vary, depending on the radius of the concave surface and hole diameter in relation to the height of the drill point.

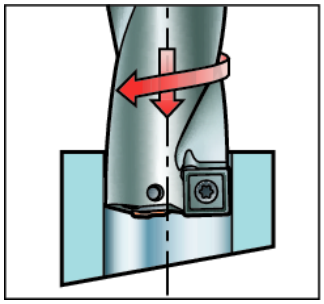
If the radius of the concave surface is small in relation to hole diameter, the periphery of the drill will be engaged first. To reduce the tendency for the drill to deflect, feed rate should be reduced to 1/3 of that recommended.



Concave surfaces are possible to drill if the radius is > 15 times drill diameter. Feed should be reduced to 1/3 of normal rate when entering.

Angled or inclined surface

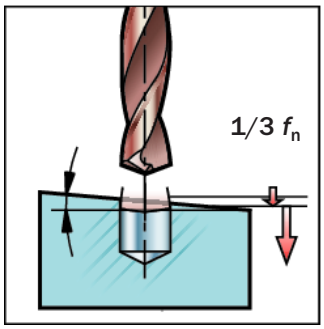


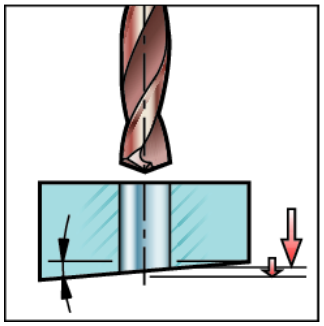


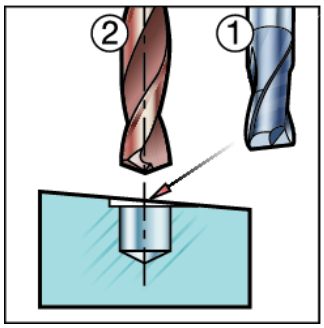
Angled or inclined surfaces lead to excessive, uneven forces on drill cutting edges, which wear the drill, cause vibrations and distort the drilling profile.

These uneven loads require a stable drill – a short length in relation to diameter to remain within tolerances. If the inclined surface angle is > 2 degrees, feed should be reduced to 1/3 of that recommended.

The same recommendations should be followed when exiting an inclined surface.





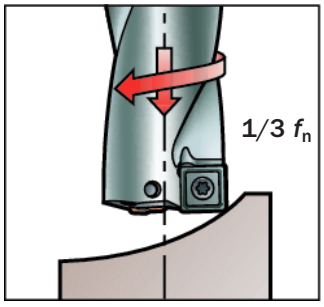


Entering inclined workpiece surfaces at an angle smaller than 5 degrees involves intermittent cutting action. Consequently, the feed should be reduced to around 1/3 of the normal feed rate until the full diameter of the drill is engaged. The same recommendations should be followed when exiting an inclined surface.

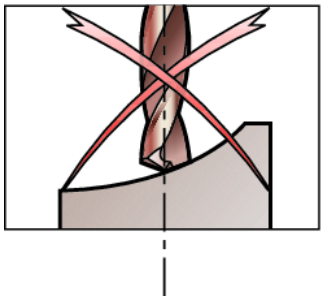
Workpiece surfaces with an inclination of 5-10° should be started by performing a centreing operation using a short drill that has the same point angle as the surface. Workpiece surfaces with inclinations larger than 10° cannot be drilled.

One alternative is to mill a small flat surface before drilling starts.

Asymmetrically curved surfaces

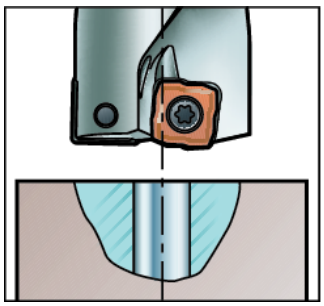


Asymmetrically curved surfaces cause the drill to bend outwards from the centre, similar to penetrating an inclined surface. Feed should be reduced to 1/3 of that for initial penetration of concave surfaces.

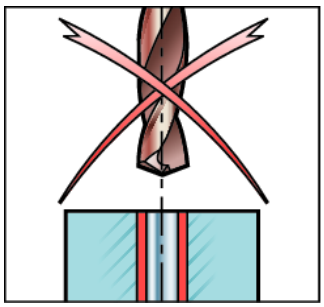


Asymmetrically curved surfaces cannot be drilled with CoroDrill Delta-C drills.

Pre-drilled holes

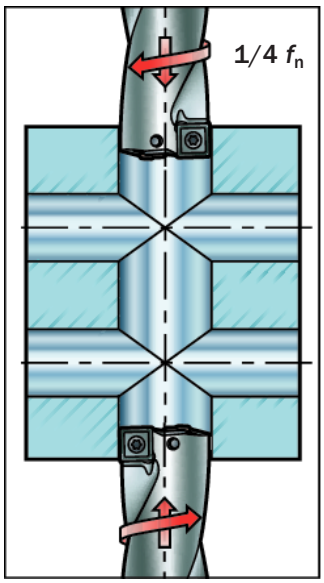


To keep the cutting force balance between the central and peripheral insert on an acceptable level, the pre-drilled hole should not be larger than $D_c/4$.



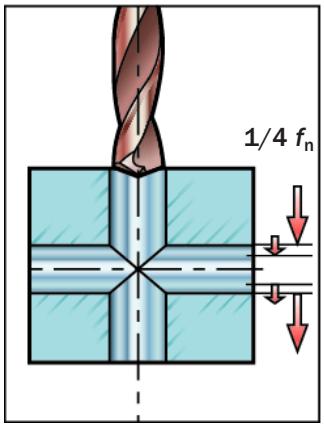
It is not possible to enlarge existing holes by counter-boring with a solid carbide or brazed drill, because no chip breaking will take place.

Crossing holes

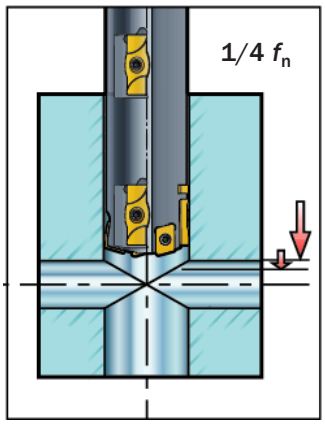


A hole crosses the axis of another hole; the drill exits from a concave surface and re-enters a concave surface, increasing the of chip evacuation problems. Great emphasis should be placed on tool stability.

When crossing a hole larger than $D_c/4$, the feed rate should be reduced to 1/4 of recommended.



Cross-hole drilling can be done if feed is reduced to 1/4 of normal rate when entering and exiting the cross hole.



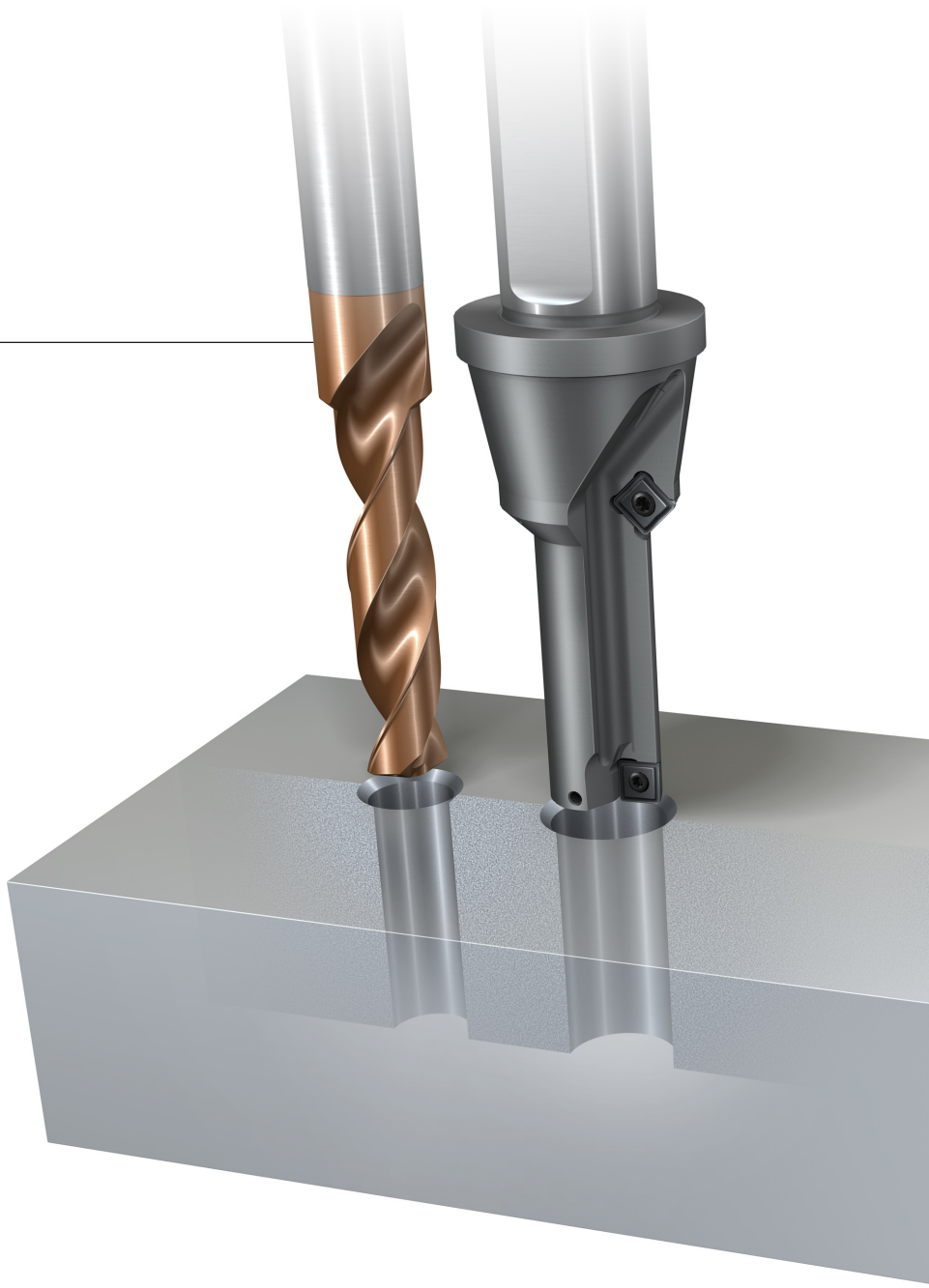
Cross hole drilling with CoroDrill 805 can be done, if the drill is equipped with an extra pair of support pads (see picture) on the drill body. These pads will support the drill when crossing a hole. Reduce feed to 1/4.

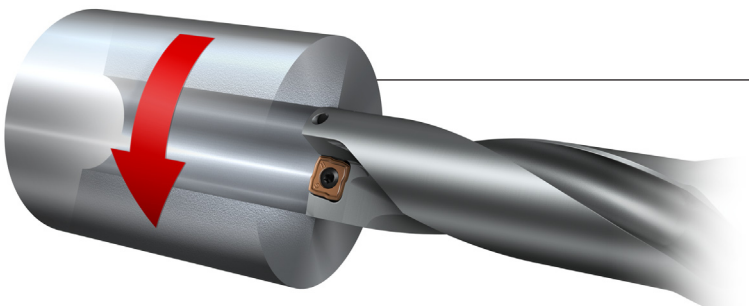
Step and chamfer drilling

Application overview

Chamfer drilling

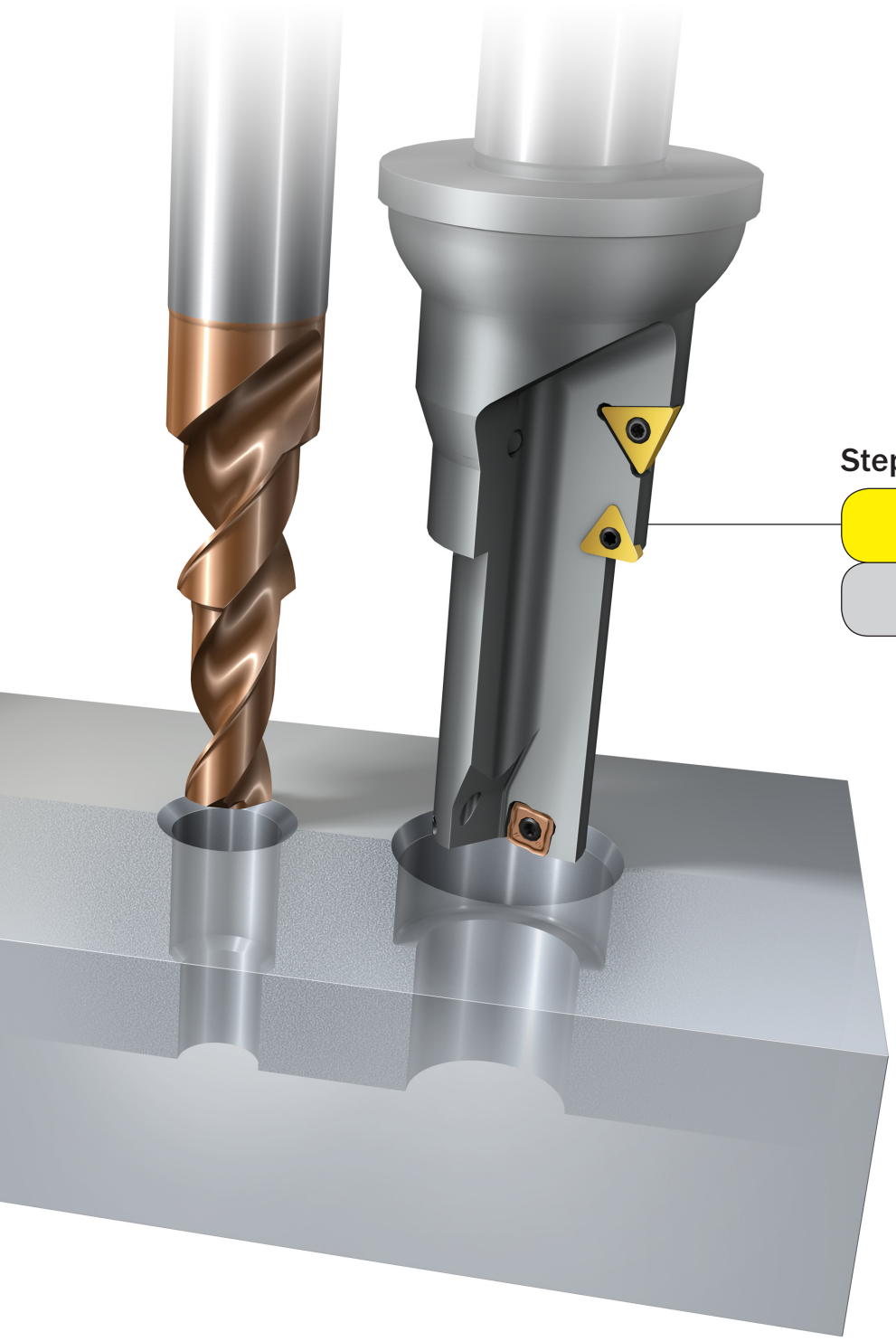
Choice of tools **E 26**
How to apply **E 27**





Non-rotating drilling

Choice of tools **E 42**
How to apply **E 42**



Step or step and chamfer drilling

Choice of tools **E 28**

How to apply **E 29**

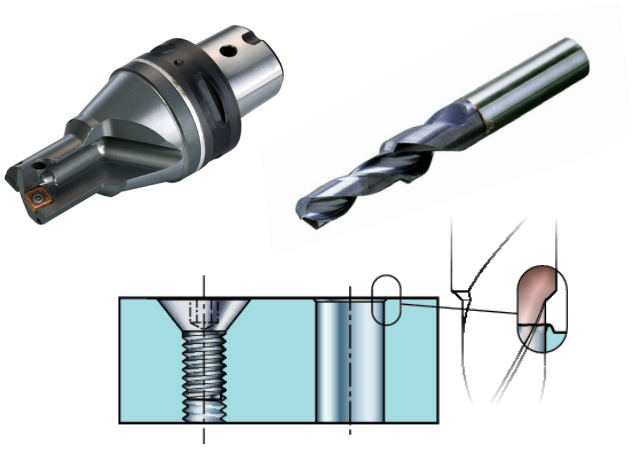
Drilling

Trouble shooting **E 44**

Chamfer drilling

A large number of all holes produced need a chamfer or at least some deburring. Some typical examples are screw and rivet holes.

There are a number of drills to choose from and, by using circular interpolation, even a turning or a milling tool can be used, see Milling, page D 126.



Choice of tools

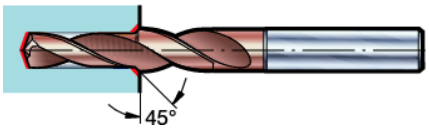
CoroDrill Delta-C®					Coromant Delta®	CoroDrill® 880		CoroTurn XS
 R841	 R840	 R842	 R846	 R850	 R411.5			
Standard	Tailor Made and Specials				Tailor Made	Tailor Made and Specials		Standard
P M K N S H	P M K N S H	K	S	N	P M K N H	P M K N S H		P M K N S H

Hole and chamfer made in one pass → Programming a tool path

CoroDrill Delta-C®

Thread holes – CoroDrill Delta-C R841

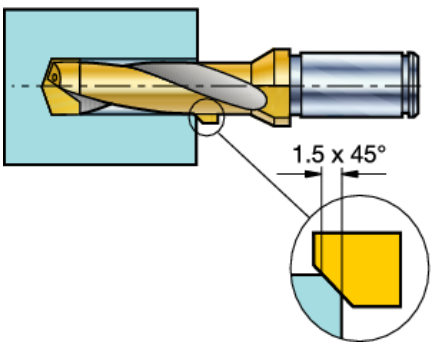
A standard programme of drills for 45 ° chamfers adapted for different thread/tap holes up to hole depths 2-3 x D_c. See Main catalogue. Can be used in all types of materials.



Coromant Delta®

45° chamfer as Tailor Made option

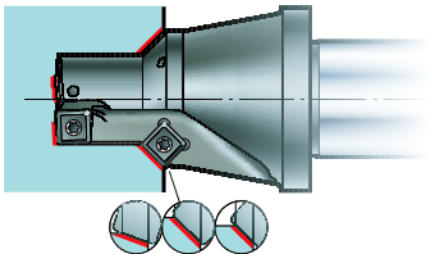
The chamfer size is 1.5 x 45° ±0.3 mm and will be located at the requested drill depth (l_4) as shown in the figure.



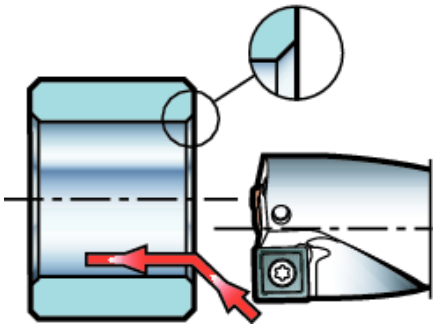
CoroDrill® 880

Dedicated choice
Tailor Made and specials

Chamfer width and angle optimized on your request. The wide choice of insert geometries and grades makes it suitable for all types of material.







Non-rotating drill

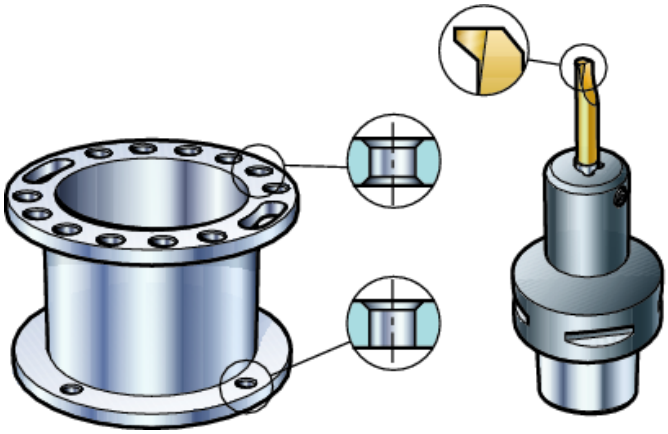
Programming the tool path

A standard CoroDrill 880 can be used by programming the tool path in a non-rotating drill application that has a rotating workpiece. See page E 42.

CoroTurn® XS and CoroMill® 327

Chamfer / back chamfering, deburring
Programming the tool path

See Milling, page D 126

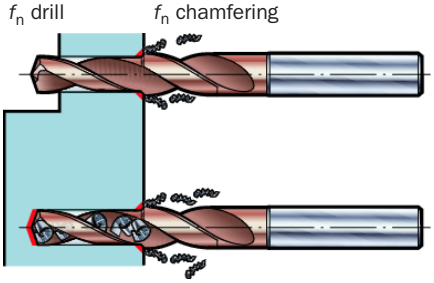


How to apply

Drilling

Normally, the same feed, f_n (mm/rev) should be used when the chamfer insert is cutting. This is important, especially when drilling a blind hole, because a reduction in feed can result in long chips from the drilling operation.

However, sometimes in long-chipping materials, a feed adjustment may be necessary during the chamfering operation to prevent long chips from becoming wrapped around the drill.

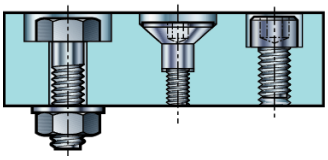


The feed, f_n, should be kept constant during the chamfering operation i.e. f_n drill = f_n chamfering. Especially important when drilling blind holes.

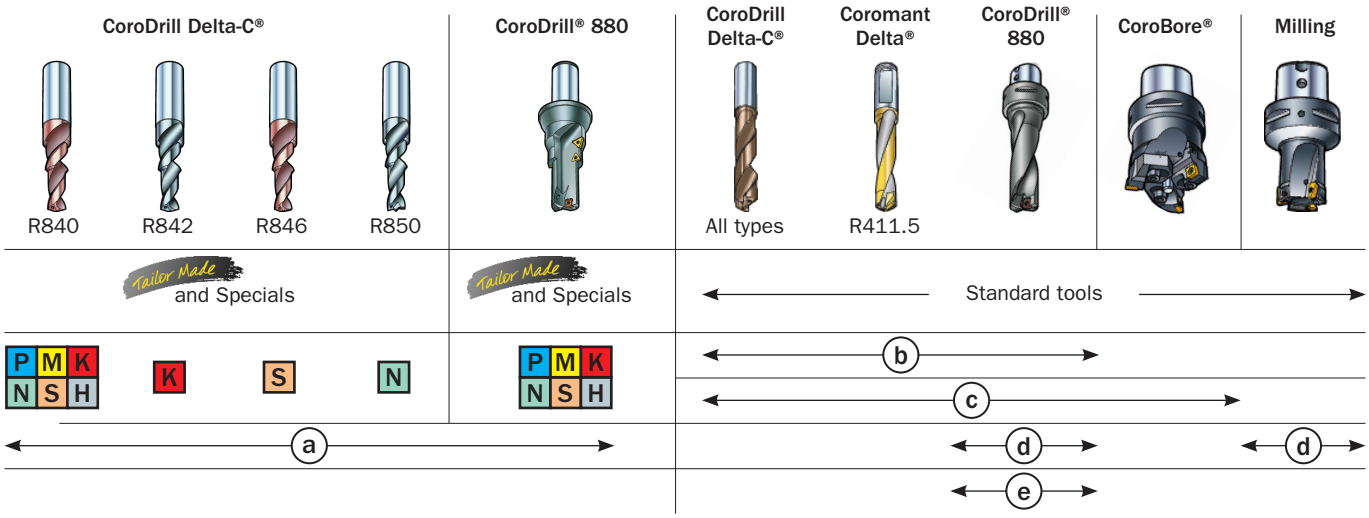
Step or step and chamfer drilling

Another common type of hole is a stepped or stepped and chamfered hole. Typical applications are components with screws or bolts where the head needs to be ”hidden”.





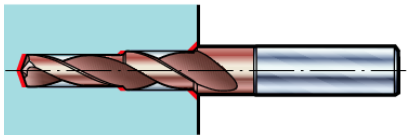
Choice of tools

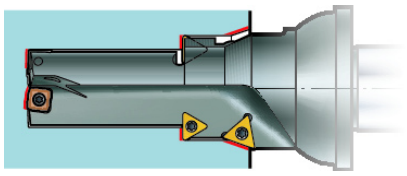


(a) Step/chamfer holes in one pass

Tailor Made and Specials to your request

CoroDrill Delta-C – All types are available with steps and chamfers.
CoroDrill 880 – Steps and chamfers to your request.





(b) Stepped holes by drilling two holes

Standard drills

CoroDrill Delta-C, Coromant Delta or CoroDrill 880 – Choose drills by hole sizes and depths.

(c) Stepped holes by drilling and boring

CoroDrill Delta-C, Coromant Delta or CoroDrill 880 plus CoroBore 820 or DuoBore – Choose drill and boring tool by hole size and depth.

(d) Stepped holes by helical interpolation

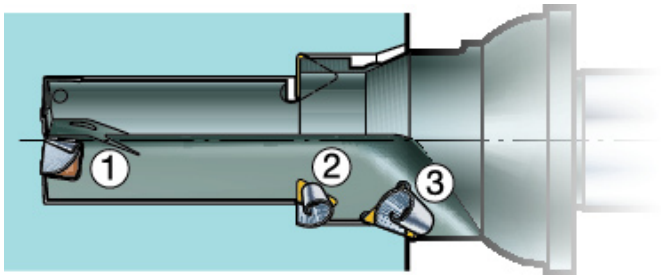
CoroDrill 880 or CoroMill cutters – The step diameter hole can be 2 x D_c. Helical interpolation with CoroDrill 880 is a slow operation and should not be used for deep steps. Choose CoroMill cutter by hole size. See Milling, Chapter D.

(e) Non-rotating drill

Choose a standard CoroDrill 880 drill and programme the tool path. See page E 42.

How to apply

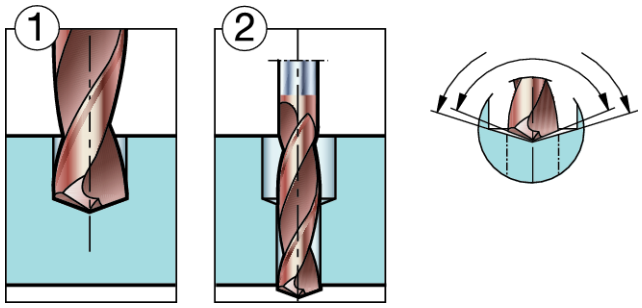
(a) Step/chamfer holes in one pass



The challenge, when using a multidiameter tool, is to achieve good chip breaking from all edges in the cut. In long-chipping materials, like low carbon steels and stainless steels, the initial tests should be made in steps:

1. First, check chip formation from the drill
2. Check chip formation from the step insert
3. Check chip formation from chamfer insert

(b) Stepped holes by drilling two holes



First larger diameter, then smaller.

Always start drilling the large diameter to assure centering of drill point and avoid chipping of cutting edges.

When drilling the small diameter with a CoroDrill Delta-C or a Coromant Delta drill, make sure that the drill point angle is equal or smaller than the large drill to ensure that the point hits the workpiece first.

(c) Stepped holes by drilling and boring

Start with drilling then boring.
See Boring, Chapter F.

(d) Stepped holes by helical interpolation

See page E 35.
See Milling, Chapter D, to learn more about hole making with milling cutters.

Dedicated methods

Application overview

Boring

Choice of tools **E 34**

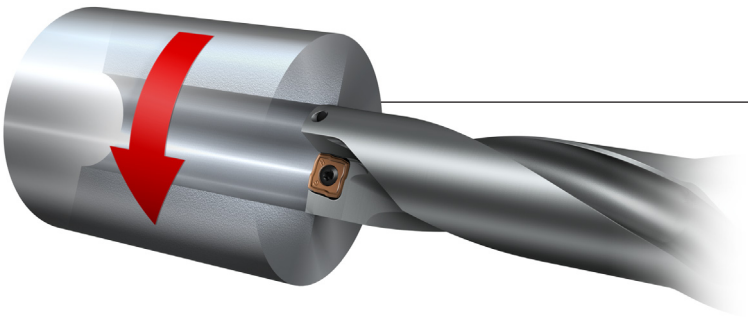
How to apply **E 34**

Radial adjusted drilling

Choice of tools **E 32**

How to apply **E 33**





Non-rotating drilling

Choice of tools **E 42**

How to apply **E 42**

Circular ramping

Choice of tools **E 35**

How to apply **E 35**

Plunge drilling

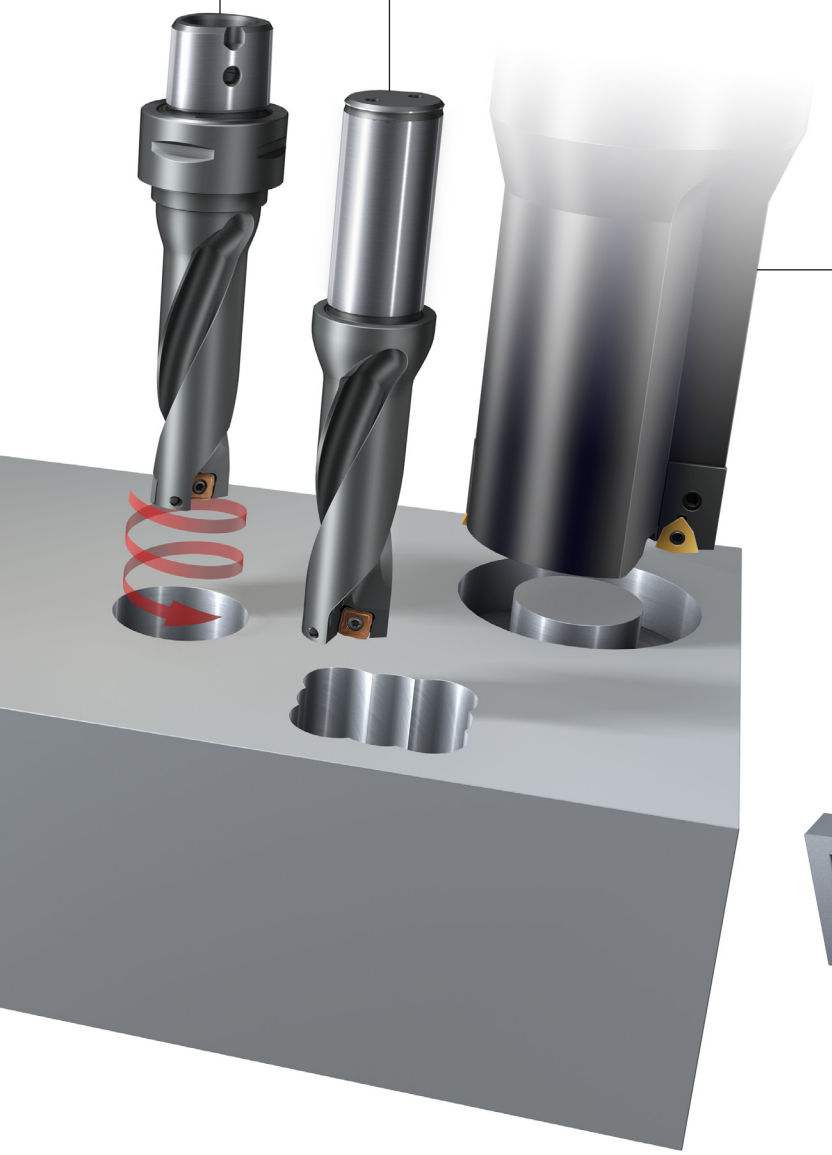
Choice of tools **E 36**

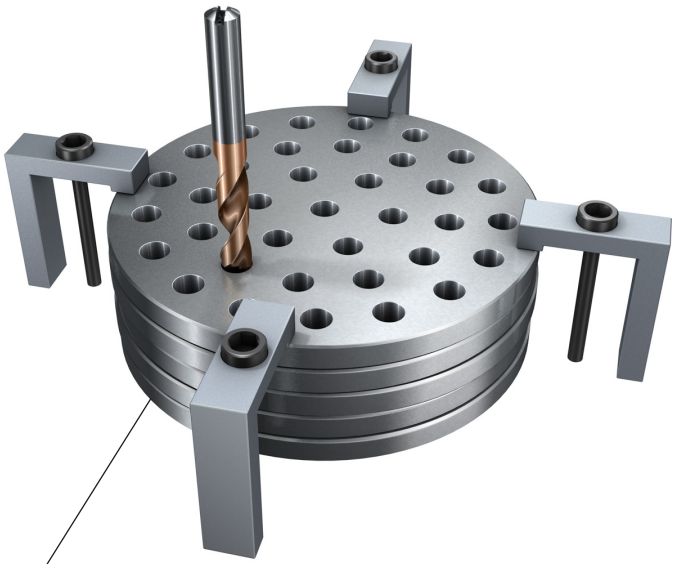
How to apply **E 37**

Trepanning

Choice of tools **E 38**

How to apply **E 38**





Stack drilling

Choice of tools **E 40**

How to apply **E 40**

Drilling

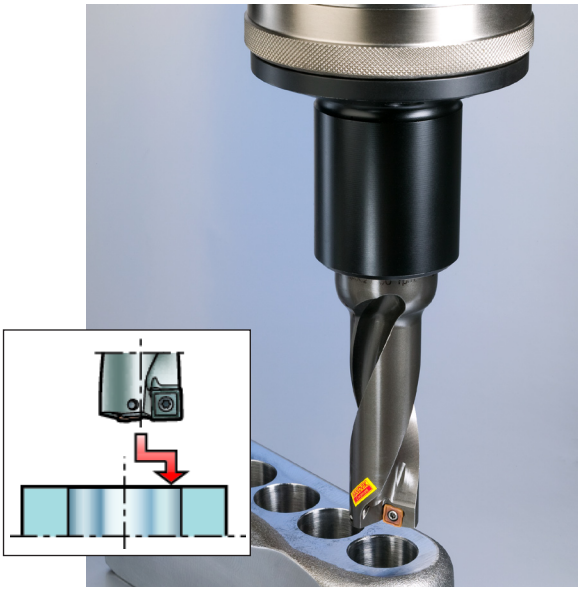
Trouble shooting **E 44**

Radial adjusted drilling

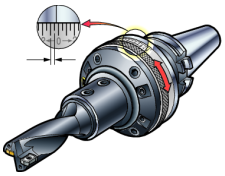
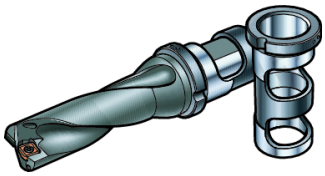
Radial adjustment of the diameter of an indexable insert drill, such as CoroDrill 880, extends the working area and makes it possible to:

- achieve a closer hole tolerance by pre-setting the exact drill diameter, which eliminates manufacturing tolerances of drill and insert.
- make a hole larger than the drill diameter and avoid the need to stock other sizes
- produce stepped and chamfer holes with a standard drill, which is possible in a non-rotating drill application.

Note: Adjusting the drill to a smaller diameter is not recommended, because it can cause the drill body to scratch against the hole wall.



Choice of tools

	CoroDrill® 880	Adjustable holder	Eccentric sleeve
			
Drill diameter D_c mm	12.00–63.50	12.00–63.50	12.00–63.50
Material	P M K N S H	P M K N S H	P M K N S H
	Non-rotating drill	Rotating drill	Rotating drill

Non-rotating drill

Choose CoroDrill 880. No specific holder is needed.
See page E 42.

Rotating drill – adjustable holder

This is most accurate and stable solution for radial adjustment in a rotating drill application. Max. radial offset is drill diameter +1.4 mm, adjustable in increments of 0.05 mm. Use the CoroDrill 880 with the ISO 9766 shank.

Rotating drill – eccentric sleeve

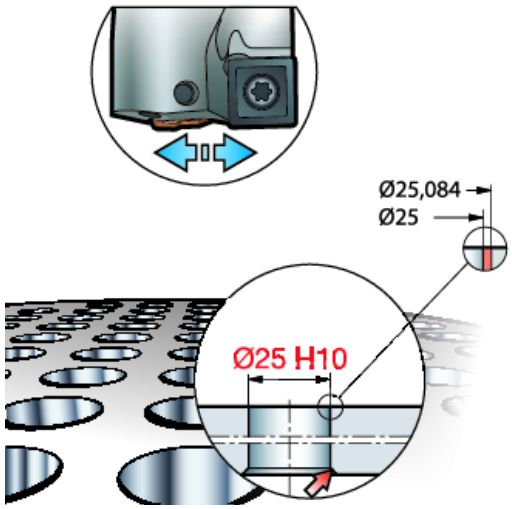
An alternative solution is the eccentric sleeve for CoroDrill 880 drills, with metric ISO 0766 shanks. It should only be used for pre-setting to reach a closer hole tolerance. The diameter offset range is ±0.3 mm. When using the sleeve, you will need to use a holder that is one size larger, i.e. drill shank diameter 25 mm + sleeve means that the holder for 32 mm diameter should be used.

How to apply

Improve hole tolerance

CoroDrill 880 produces holes with a tolerance of IT12-13, depending on drill length. By pre-setting the drill, and thereby eleminating the manufactu-ring tolerances of the drill body/tip seat and the insert, a tolerance within ±0.05 mm (IT10-11) can be maintained under stable conditions.

Note: Indexable insert drills produce a slightly smaller hole diameter when breaking through a hole, due to loss of the force balance between the two inserts. However, performing a small chamfer/deburring operation can eliminate this deviation, see picture.



Producing a hole larger than the drill

Max. radial adjustment for a CoroDrill 880 depends on the central and peripheral insert overlap. In the Main catalogue, max. radial adjustment for different drill diameters are listed; see example to the right. Radial adjustment influences the cutting force, balance therefore choose the lower feed in the recommended range. See Main catalogue.

Drill dia.		Radial adjustment	
D_c mm	**Ordering code**		D_c max
14	**880-D1400L20-02**	0.50	15.0
14.5	**880-D1450L20-02**	0.45	15.4
15	**880-D1500L20-02**	0.40	15.8
15.5	**880-D1550L20-02**	0.30	16.1
16	**880-D1600L20-02**	0.30	16.6

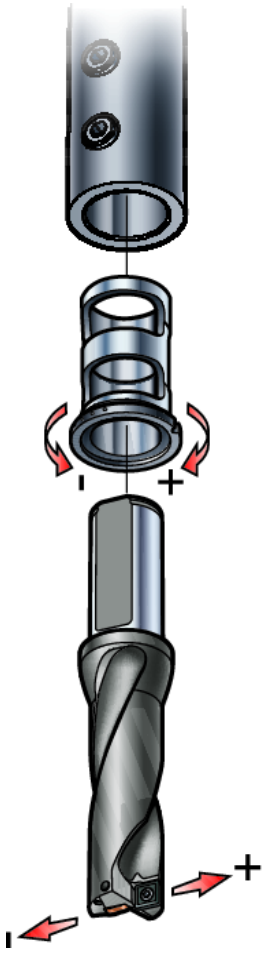
Setting the drill

Non-rotating drill, see page E 42.

Rotating drill – adjustable holder
See Tool holding/Machines, Chapter G, for setting instructions.

Rotating drill – excentric sleeve
Drill diameter can be adjusted for closer hole tolerance. The adjustment range is approx. ±0.3 mm, but adjustment in the negative direction should be made only if the drill produces an oversized hole (not in order to achieve undersized holes).

- One dot increases/decreases the diameter by 0.10 mm.
- Increase the diameter by turning the sleeve clockwise.
- Decrease the diameter by turning the sleeve counterclock-wise.
- Use both screws to clamp the drill in the fixture and make sure the bolts in the holder are long enough.



Boring

Use the versatile indexable insert drill CoroDrill 880 in boring operations can be made in order to save tool changing time.

CoroDrill Delta-C in not recommended, except with drill type R850.



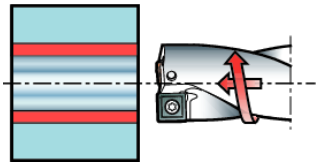
Choice of tools

	CoroDrill® 880	CoroDrill Delta-C®
		 R850
Drill diameter D_c mm	12.00–63.00	5.00–14.00
Material	P M K N S H	P M K N S H

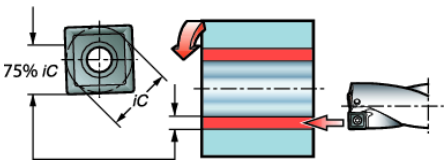
How to apply

CoroDrill®880

Enlarging a hole can be done using a larger drill. Boring with a non-rotating drill can be performed the same way, or by off-setting the tool radially. A shorter drill will cope with this operation better, because it is less prone to vibration and able to operate at higher feeds. The maximum depth for boring operations should be limited to 75% of the insert to avoid drill deflection.



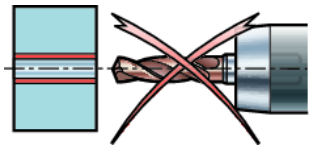
Boring with a larger drill.



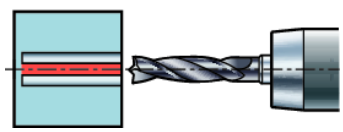
A non-rotating drill can also be off-set radially.

CoroDrill Delta-C® 850

Normally, boring with CoroDrill Delta-C drills is not recommended:, chip breaking becomes an issue because only the periphery of the cutting edges will be included in the cut. However, type R850 drill can be used in core drilling of aluminium.



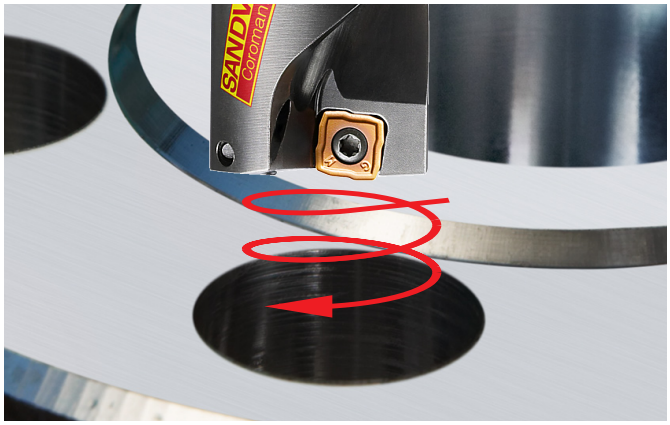
CoroDrill Delta-C 840, 842, 844, 846 should not be used for boring operations.



CoroDrill Delta-C 850 works well in core drilling of aluminium.

Circular ramping

A rotating tool makes a hole following a circular toolpath while simultaneously being fed axially. The method can be used to machine existing holes. This is a circular ramping operation where a tool, normally about half the diameter of the hole, is used at a ramping angle recommended for the tool.



Choice of tools

	CoroDrill® 880
	
Drill diameter D_c mm	12.00–63.50
Drill depth	2 x D_c
Material	P M K N

Suitable tools are, in addition to CoroDrill 880, the CoroMill 390 endmill, the CoroMill 300 round insert cutter and the CoroMill Plura solid carbide end mill, and any cutter with ramping ability.

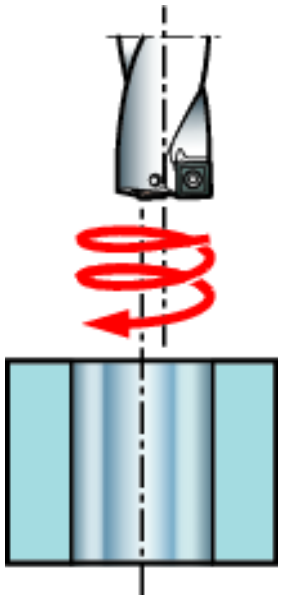
Note: Circular ramping with longer drills is not recommended

How to apply

- Circular ramping can be performed with the CoroDrill 880
- This should be considered for certain one-off operations, or when productivity is of lesser importance

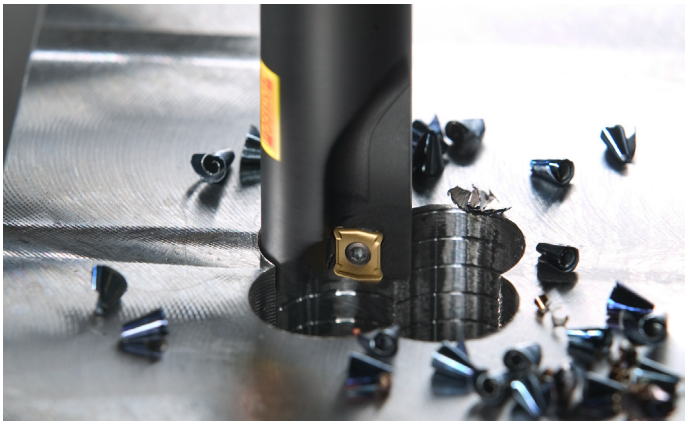
Cutting data:

- Cutting speed and feed rate can be the same as that recommended for conventional drilling
- Max. hole depth is 2 x D_c
- Max. pitch is the insert radius +0.03 mm



Plunge drilling

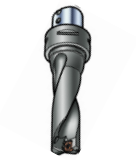
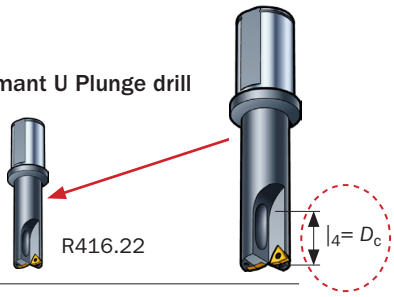
Plunge drilling is an effective method for roughing-out cavities; it makes a deeper hole or cavity through repeated axial cuts using a drill or milling cutter. It is especially suitable for rough machining – it is power-efficient, productive and reduces demands on the machine tool spindle, because the main cutting forces are directed axially along the spindle. Internal plunge milling is started by drilling a hole for the plunging tool.



Choice of tools

Indexable drills are used centre cutting with two inserts, and are very productive. 70% of the drill diameter can be applied to the next overlapping cut, if you are using the plunge drill at full indexable insert drill cutting data; metal removal rate is high.

Plunge milling is similar to plunge drilling using the positive indexable insert cutter CoroMill 390, the round insert cutter CoroMill 300 or the solid carbide CoroMill Plura. The high-feed milling CoroMill 210 is suitable for many applications. However, a milling cutter has a limited step-over – only half of the axial insert cutting edge length. It can be an interesting alternative when diameters are large enough to have a number of tooth engaged to provide high feeds.

	CoroDrill® 880	Coromant U Plunge drill
		
Drill diameter D_c mm	12.00–63.00	12.7–35.00
Material	P M K N S H	P M K N S H
	Standard CoroDrill 880 can be used for a hole depth up to 3 x D_c, without risk of drill deflection or vibrations.	For depths up to 6 x D_c, the dedicated plunge drill Coromant U (416.22) is ideal. Max drill depth is 1 x D_c, but plunge drilling (overlapping holes) can be machined up to 6 x D_c in through- and blind-holes. Blind holes need to be monitored for chip accumulation and evacuation.

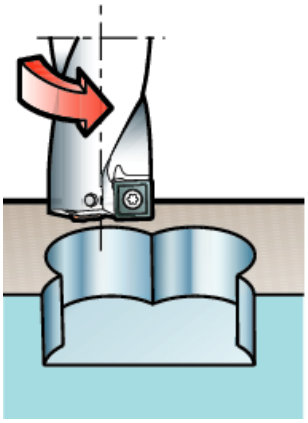
Grade and geometry

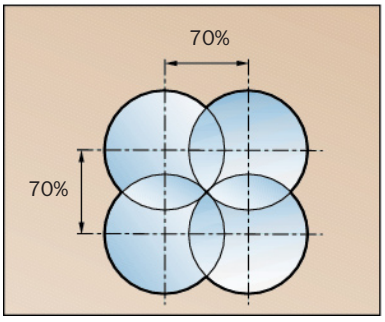
Due to the interrupted cut, a tough grade and a strong geometry should be used.
CoroDrill 880: Geometry GR or GT and grade GC4044/1044.
Coromant U Plunge drill: Geometry 53 and 1020.

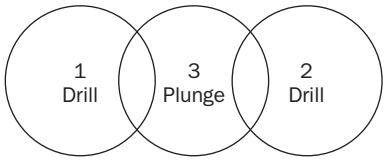
How to apply

- For best stability, always use the shortest possible drill.
- Only use an internal coolant-supply to ensure sufficient chip evacuation.
- Apply same cutting data as in ordinary indexable insert drilling.
- A step-over rate of 70% of the drill diameter is the maximum and also provides the optimum cavity-making efficiency without leaving a material-core in the cavity.
- Cavities to be plunge drilled should be started using an ordinary indexable drill to make an initial hole. This drill should be as large as possible.
- If two holes are drilled, followed by plunging in between, plenty of space is left for chips to be evacuated.

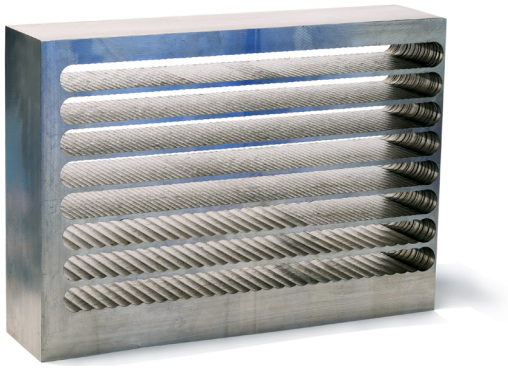
If conditions are unstable, feed should be reduced down to 1/3 of the normally recommended rate.

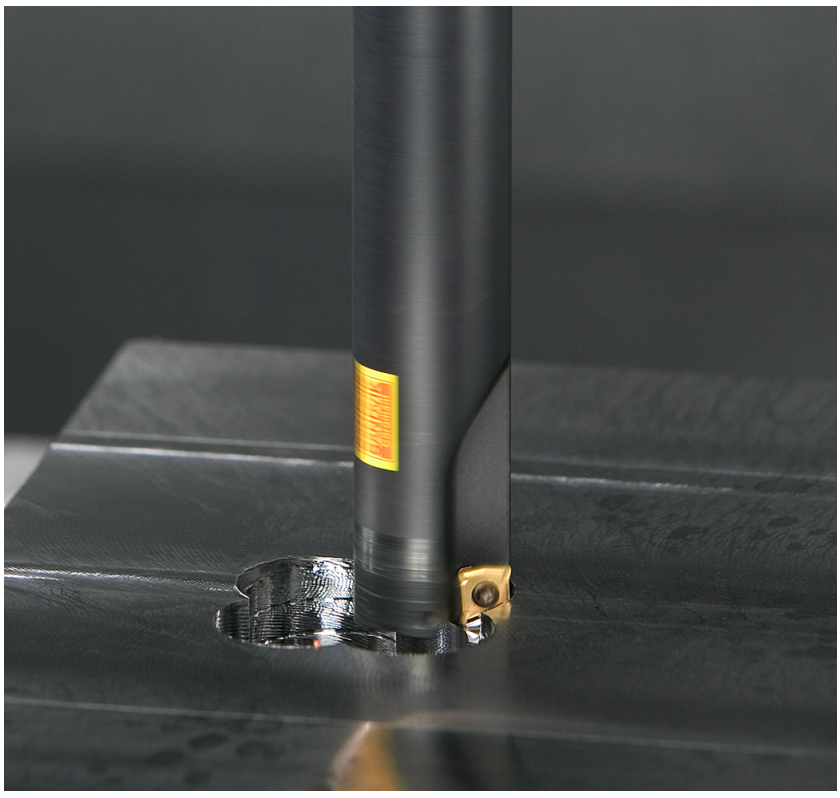






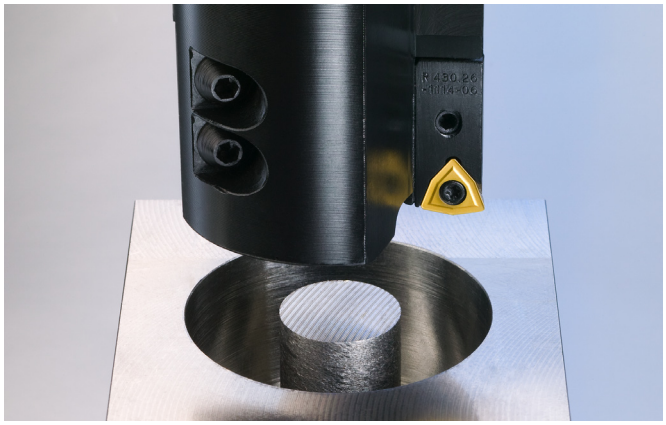






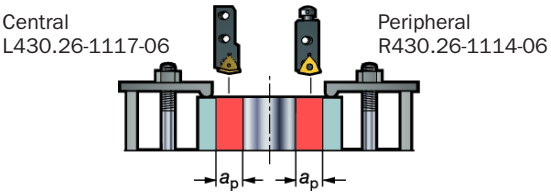
Trepanning

Trepanning is used for larger hole diameters and where machine power is limited, because it is not as power-consuming as solid drilling. The trepanning tool does not machine the whole diameter, only a ring at the periphery. Instead of removing all the material in the form of chips, a core is left in the centre of the hole – consequently, this method is for through-hole applications.



Choice of tools

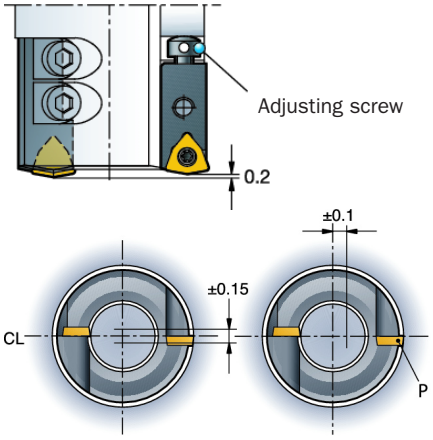
	T-Max® U Trepanner
	 R416.7
Drill diameter D_c mm	60.00–110.00
Drill depth	2.5 x D_c
Material	P M K N S
Hole tolerance	±0.2
Surface finish R_a	2–7 µm



The drill is designed for drilling solid workpieces as well as stacked components, with or without air gaps.

How to apply

Set-up recommendations



Rotating and non-rotating drills

The peripheral insert must be set 0.20 mm behind the inner cartridge.

Non-rotating drill

- The drill should be positioned with the inserts on the horizontal axis.
- The drill must not be offset from the workpiece centerline, with reference to the peripheral insert, by more than -0.15 mm.
- The peripheral cutting edge (P) should be set to within +0.1 mm in the horizontal plane from the centre-line of spindle (CL).

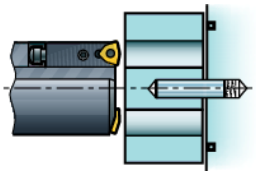
Core handling

Vertical set up

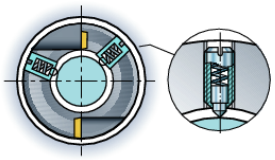
The core will fall down when drill breaks through, which does not normally cause a problem.

Horizontal set-up

Long and heavy cores may require a core support to prevent the core from falling down and chipping the inner insert.



Rotating drill - core fixing
Drill a hole into the core. Fix plug in the hole to prevent the core (C) from falling.



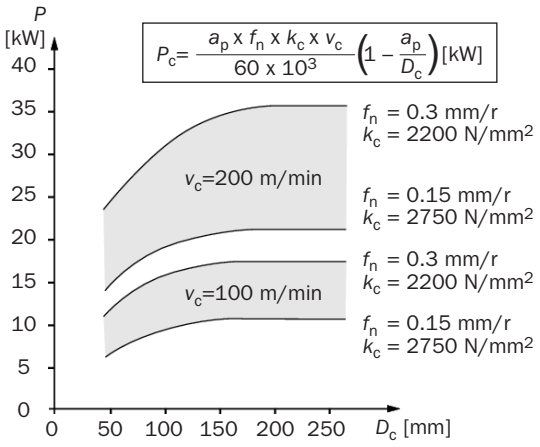
Non-rotating drill - core support
If the drill is fitted with a core support, the drill should be mounted with the cartridges located on the vertical axis and the peripheral insert downwards.

Cutting data

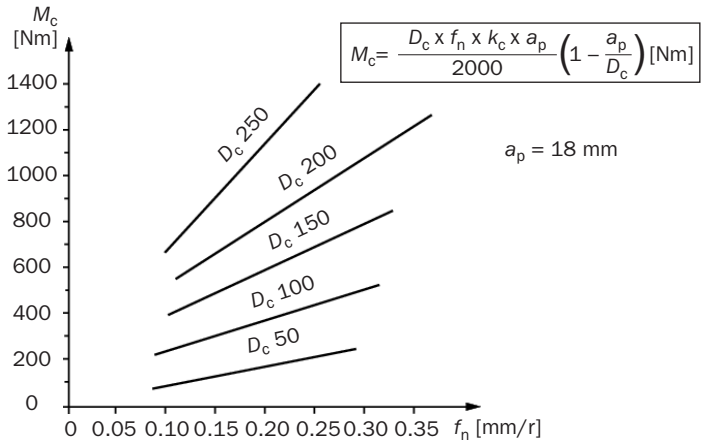
	ISO	Cutting speed v_c m/min	Feed f_n mm/r
P	Steel	100–250	0.07–0.20
M	Stainless steel	100–250	0.07–0.20
K	Cast iron	100–250	0.15–0.25
N	Aluminium	250–400	0.12–0.22
S	Titanium	40–100	0.08–0.16

Graphs for T-Max® U trepanning tools – R416.7

Net power

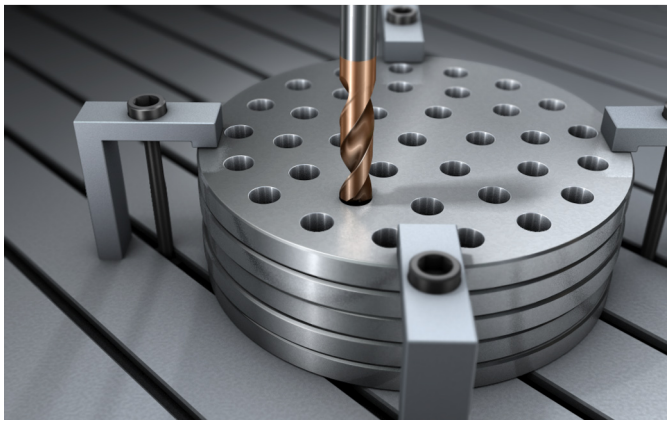


Trepanning torque



Stack drilling

This method is used for drilling a large number of holes through thin plates. Typical applications include the drilling of baffle plates for heat exchangers and components for bridges.



Choice of tools

	CoroDrill Delta-C®	Coromant Delta®	T-Max® U Stack drill *	T-Max® U Trepanner *
	 R840 R846	 R411.5	 R416.01*	 R416.7*
Drill diameter D_c mm	0.3–20.00	9.50–30.40	27.00–59.00	60.00–110.00
Drill depth	2–7 x D_c	3.5–5 x D_c	2.5 x D_c	2.5 x D_c
Material	P M K N S H	P M K N H	P M	P M K N S

**See electronic catalogue for ordering information.*

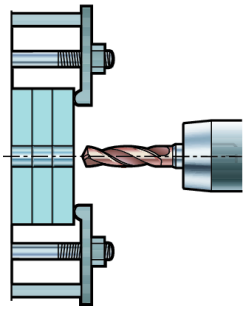
How to apply

General

Minimizing the gap between the stacked plates is crucial for successful stack and drilling. This can be done by clamping or welding the plates together. A good common practice is to insert industrial paper (thickness of approx. 0.5-1 mm) between the plates to level out any irregularities and damp vibrations.

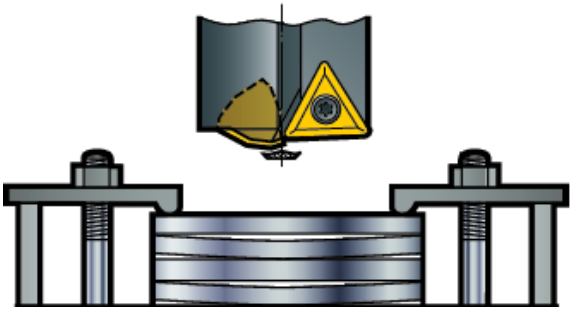
CoroDrill Delta-C® and Coromant Delta®

No feed reduction is normally needed. Note that type R850, which has a point angle configuration that is different from the other CoroDrill Delta-C drills, is not suitable for stack drilling.



T-Max® U stack drill R416.01

The drill design is optimized for stack drilling and creates a minimal disc that is evacuated along the chip channel when the drill breaks through a stacked plate.



Cutting data for:

P Low carbon steel:
v_c = 150–300 m/min and
f_n = 0.05–0.12 mm/r

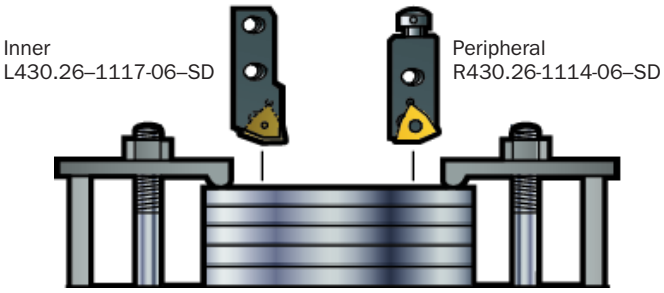
M Stainless steel:
v_c = 75–200 m/min and
f_n = 0.05–0.12 mm/r

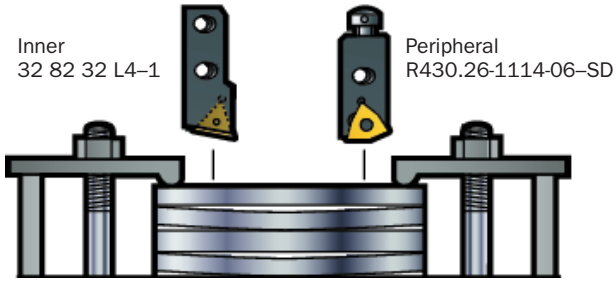
Note: CoroDrill 880 is not suitable for stack drilling because the break through disc is too large.

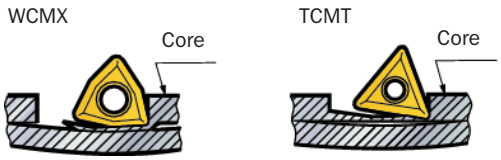
T-Max® U Trepanner R416.7

For workpieces without air gaps, use the SD cartridge and a WCMX insert size 06 for both peripheral and inner cartridges.

For workpieces with air gaps, use type 32 82 32 L4-1 as the inner cartridge together with a TCMT size 16 insert.





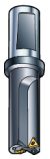


Non-rotating drilling

Turning centers and bar fed sliding machines often rotate the workpiece instead of the drill. When using this method, it is extremely important. To ensure that the drill is aligned with the centre line of the machine.



Choice of tools

	CoroDrill Delta-C®	Coromant Delta®	CoroDrill® 880	CoroDrill® 805	T-Max® U Trepanner	T-Max® U Stack drill
	 All types	 R411.5			 R416.7	 R416.01
Drill diameter D_c mm	0.3–20.00	9.50–30.40	12.00–63.50	25.00–65.00	60.00–110.00	27.00–59.00
Drill depth	2–7 x D_c	3.5–5 x D_c	2–5 x D_c	7–15 x D_c	2.5 x D_c	2.5 x D_c

All drilling tools can be used in a non-rotating drilling application.

(See page E 13 and E 14 for additional information)

How to apply

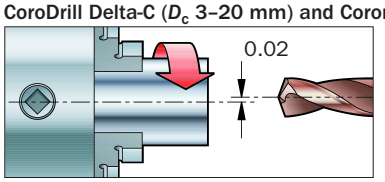
Tool run-out

Make sure that the run-out or TIR (Total Indicator Readout) is within the values in the figure at the right.

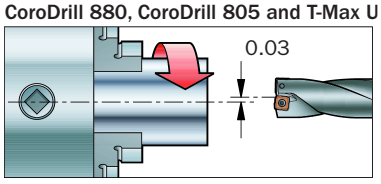
An unacceptable run-out can be temporarily reduced by turning the drill or the collet 90°, 180° or 270° to find lowest TIR.

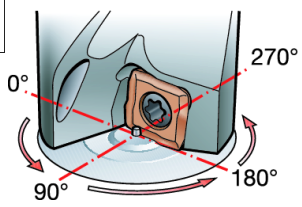
Note: An indexable insert drill, like CoroDrill 880, forms a small centre core which can be seen in the bottom of the hole or the disc. The size of the core should be within 0.05-0.15 mm; otherwise it can cause edge fracture, vibrations, oversized holes and wear on drill body. When turning the drill, the core dimension will vary due to the different positions.

CoroDrill Delta-C (D_c 3–20 mm) and Coromant Delta.



CoroDrill 880, CoroDrill 805 and T-Max U drills



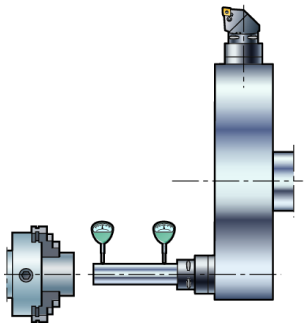


Aligning the drill

The drill alignment must be parallel with the spindle axis; otherwise the hole can be over- or under-sized, or funnel-shaped. Measurement can be carried out with a dial-indicator together with a test-bar.

Another possibility is to drill holes with the drill turned 90°, 180° or 270°, in between. Hole measurement will indicate the state of machine alignment.

For pre-setting the gauge, see Tool holding/Machines, Chapter G.

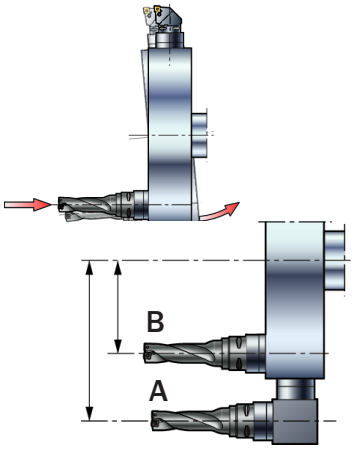


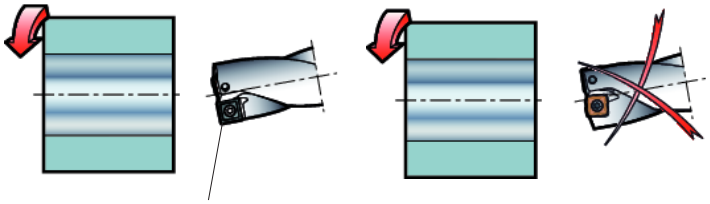
Misalignment due to turret deflection

Deflection of the turret on a CNC-lathe can be a problem, especially when using larger drills and high feeds, f_n, which can cause a high force.

First, check if the leverage can be minimized by mounting the tool differently. Position B is preferable to position A.

If this is not possible, a reduction of the feed/revolution (f_n), will decrease the feed force. To maintain the same productivity, the speed, v_c, can be increased, because it doesn't affect the feed force.





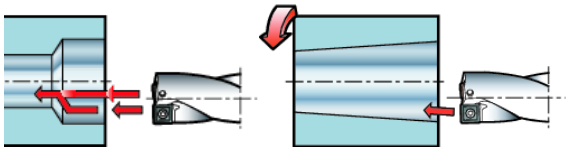
If turret deflection/misalignment can't be avoided, the drill should be mounted with the peripheral insert set as shown in the picture at left, to avoid wear on the drill body.

Radial adjustment of non-rotating drill

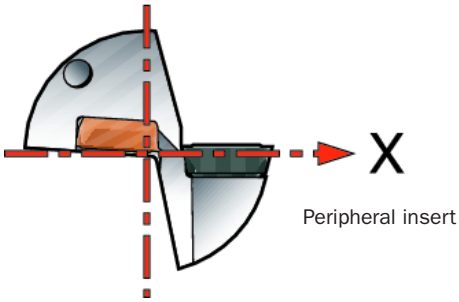
Non-rotating drilling opens up great possibilities in combination with CoroDrill 880 to:

- achieve closer tolerance holes by pre-setting. See page E 33.
- perform boring operations. See page E 34.
- programme a tool path in order to achive stepped and/or chamfered holes or conical holes using a standard CoroDrill 880 drill. The stepped/chamfered hole is made in two steps: drilling plus boring.
- A conical hole can be made in one pass, if the max. hole diameter does not exceed the max. radial adjustment of the drill.

Note: The peripheral insert should be parallel to the x-axis of the machine. The position of the drill in the turret will then determine how the offset influences the hole diameter.



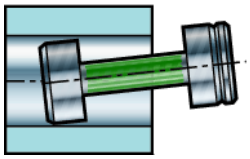
Stepped and/or chamfered and conical holes



Peripheral insert

Trouble shooting

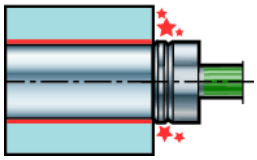
CoroDrill® 880



Oversized holes

Rotating drill
1. Increase coolant flow, clean filter, clear coolant holes in drill
2. Try a tougher geometry on peripheral side (keep centre insert)

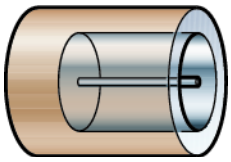
Non-rotating drill
1. Check alignment on lathe
2. Rotate drill 180 degrees
3. Try a tougher geometry on peripheral side (keep centre insert)



Undersized holes

Rotating drill
1. Increase coolant flow, clean filter, clear coolant holes in drill
2. Try a tougher geometry on centre side and a light cutting geometry on peripheral side

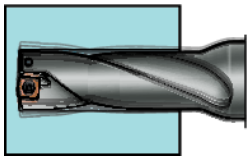
Non-rotating drill
1. Stationary: Check alignment on lathe
2. Stationary: Rotate drill 180 degrees
3. Try a tougher geometry on the centre and a light-cutting geometry on the periphery



Pin in hole

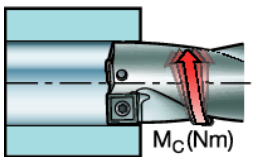
Rotating drill
1. Increase coolant flow, clean filter, clear coolant holes in drill
2. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data
3. Shorten drill overhang

Non-rotating drill
1. Check alignment on lathe
2. Increase coolant flow, clean filter, clear coolant holes in drill
3. Shorten drill overhang
4. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data



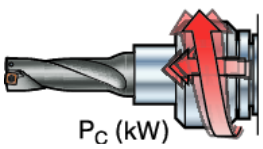
Vibrations

1. Shorten drill overhang, Improve the workpiece stability
2. Reduce cutting speed
3. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data



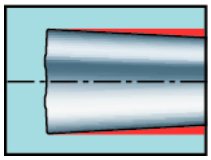
Insufficient machine torque

1. Reduce feed
2. Choose a light cutting geometry to lower the cutting force



Insufficient machine power

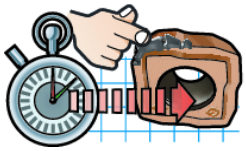
1. Reduce speed
2. Reduce feed
3. Choose a light cutting geometry to lower the cutting force



Hole not symmetrical

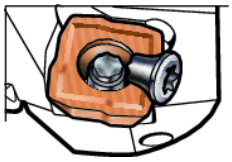
Hole widens at bottom (due to chip jam on centre insert)

1. Increase coolant flow, clean filter, clear coolant holes in drill
2. Try a different geometry on peripheral side and adjust feed rate within recommended cutting data
3. Shorten drill overhang



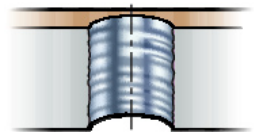
Poor tool life

1. Check cutting data recommendation
2. Increase coolant flow, clean filter, clear coolant holes in drill
3. Shorten drill overhang, Improve the workpiece stability
4. Choose a more wear-resistant grade if possible



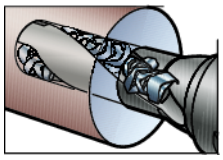
Broken insert screws

1. Use torque wrench to fasten the screw together, apply Molykote



Bad surface finish

1. Important to have good chip control
2. Reduce feed (if it is important to keep v_f, increase speed as well)
3. Increase coolant flow, clean filter, clear coolant holes in drill
4. Shorten drill overhang, improve the workpiece stability

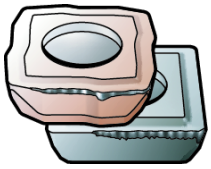


Chip jamming in the drill flutes

Caused by long chips

1. Check geometry and cutting data recommendations
2. Increase coolant flow, clean filter, clear coolant holes in drill
3. Reduce feed within recommended cutting data
4. Increase cutting speed within recommended cutting data

Tool wear – CoroDrill® 880

	Cause	Solution
 **Flank wear**	**a)** Cutting speed too high **b)** Insufficiently wear resistant grade	**a)** Reduce cutting speed **b)** Choose a more wear resistant grade
 **Crater wear**	**Peripheral insert:** Diffusion wear caused by temperature too high on rake face **Central insert:** Abrasive wear caused by built-up-edge and smearing.	**Peripheral insert:** • Select GC4024 or GC4014 with Al_2O_3-coating against oxidation • Reduce speed **Central insert:** • Choose GC1044, if H13A is used • Reduce feed **General:** Choose a more positive geometry
 **Plastic deformation (peripheral insert)**	**a)** Cutting temperature (cutting speed) too high, combined with a high pressure (feed, hardness of workpiece) **b)** As a final result of excessive flank wear and/or crater wear	**a–b)** Select a more wear resistant grade with better resistance to plastic deformation, i.e. GC4014 or GC4024. **a–b)** Reduce cutting speed **a)** Reduce feed
 **Chipping**	**a)** Insufficent toughness of grade **b)** Insert geometry too weak **c)** Built-up-edge (BUE) **d)** Irregular surface **e)** Bad stability **f)** Sand inclusions (cast iron)	**a)** Select a tougher grade, i.e. GC4044 **b)** Select a stronger geometry, i.e. -GT **c)** Increase cutting speed or select a more positive geometry **d)** Reduce feed at entrance. Choose geometry, -GT **e)** Improve stability **f)** Choose a stronger geometry i.e. -GR or -GT. Reduce feed
 **Built-up-edge (BUE)**	**a)** Low cutting speed (temperature too low at the cutting edge) **b)** Cutting geometry too negative **c)** Very sticky material, such as certain stainless steels and pure aluminium **d)** Percent of oil mixture in cutting fluid too low	**a)** Increase cutting speed or change to a coated grade **b)** Select a more positive geometry **c-d)** Increase oil mixture and volume/pressure in cutting fluid

Chip evacuation - general recommendations

Check points and remedies

1. Make sure the right cutting data and drill geometry are used.
   See pages E 16 and E 17 for recommendations.
2. Inspect chip form (compare with picture on page E 15).
3. Check if the cutting fluid flow and pressure can be increased.
4. Inspect the cutting edges. Chipping on the edge can cause long chips, because the chip is divided.
5. Check if the machinability has changed due to new batch of workpiece material. Cutting data may need to be adjusted.
6. Adjust feed and speed. See diagram on page E 7.

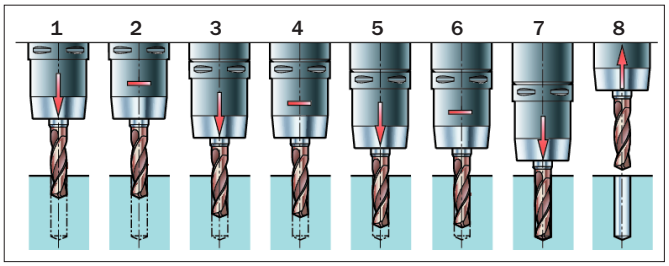


Peck drilling – CoroDrill Delta-C® and Coromant Delta®

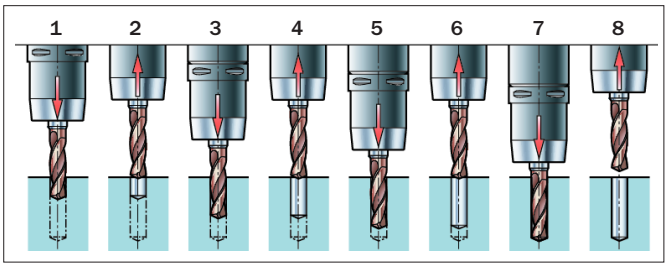
Peck drilling can be used if no other solution is to be found. There are two different ways to perform a peck drilling cycle:

Method 1 for best productivity:
Do not retract the drill more than approx. 0.3 mm from the hole bottom. Alternately, make a periodical stop, while the drill is still rotating, before continuing to drill.

Method 2 for best chip evacuation:
After each drilling cycle, retract the drill out from the hole to ensure that no chips are stuck onto the drill

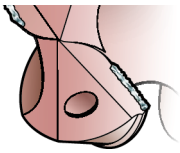
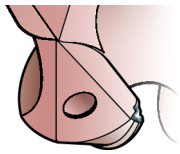
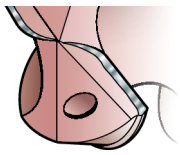
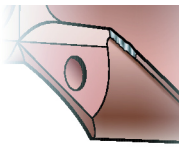
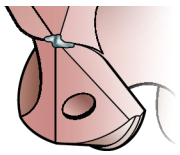
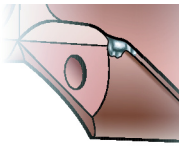
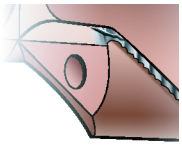


Method 1



Method 2

Tool wear – CoroDrill Delta-C® and Coromant Delta®

	Cause	Solution
 Built-up-edge	1. Cutting speed too low and edge temperature too high 2. Neg. land to large 3. No coating 4. Percentage of oil in the cutting fluid too low	1. Increase cutting speed or use external cutting fluid 2. Sharper cutting edge 3. Coating on the edge 4. Increase the percentage of oil in the cutting fluid
 Chipping on the edge corner	1. Unstable fixturing 2. TIR to large 3. Intermittent cutting 4. Insufficient cutting fluid (Thermal cracking) 5. Unstable tool holding	1. Check fixture 2. Check radial run-out 3. Decrease the feed 4. Check cutting fluid supply 5. Check the toolholder
 Large wear on the cutting edge	1. Cutting speed too high 2. Feed too low 3. Grade too soft 4. Lack of cutting fluid	1. Decrease the cutting speed 2. Increase the feed 3. Change to harder grade 4. Check for proper cutting fluid supply
 Chipping on the cutting edge	1. Unstable conditions 2. Maximum allowed wear exceeded 3. Grade too hard	1. Check the set-up 2. Replace drill sooner 3. Change to softer grade
 Wear on the circular lands	1. TIR too large 2. Cutting fluid too weak 3. Cutting speed too high 4. Abrasive material	1. Check the radial run-out 2. Use neat oil or stronger emulsion 3. Decrease cutting speed 4. Change to harder grade
 Wear on the chisel edge	1. Cutting speed to low 2. Feed too high 3. Chisel edge too small	1. Increase cutting speed 2. Decrease feed 3. Check dimensions
 Wear due to plastic deformation	1. Cutting speed or/and feed too high 2. Not enough cutting fluid supply 3. Unsuitable drill/grade	1. Decrease the cutting speed or/and feed 2. Increase cutting fluid pressure 3. Use a harder grade
 Thermal cracks (notches)	1. Inconsistent cutting fluid	1. Check cutting fluid supply 2. Fill cutting fluid tank

Products – drilling



CoroDrill® 880

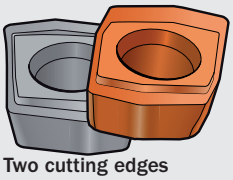


Up to 100% productivity increase
Step-by-step entrance into the workpiece and perfectly balanced cutting forces mean that higher feeds (f_n) can be used.

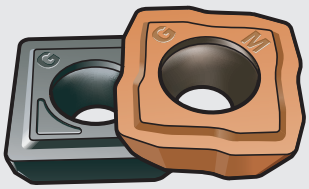


Closer hole tolerances
Balanced cutting forces produce closer hole tolerances and wiper geometry on the peripheral insert offers better surface finish.

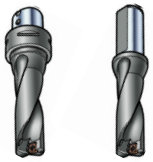




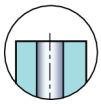
Two cutting edges
12.00–13.99 mm

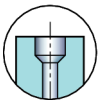


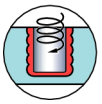
Four cutting edges
14.00–63.50 mm

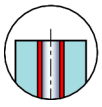
				Engineered drills 
Drill diameter D_c mm	12.00–13.99	• 12.00–43.00 •• 43.01–58.00 ••• 58.01–63.50	12.00–63.50	12.00–
Drill depth	2–5 x D_c	• 2–5 x D_c •• 2–4 x D_c ••• 2–3 x D_c	2–5 x D_c	
Material	P M K N S H	P M K N S H	P M K N S H	P M K N S H
Hole tolerance	IT12–13	IT12–13	IT12–13	IT12–13
Surface finish R_a	1–5 µm	1–5 µm	1–5 µm	1–5 µm

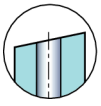
Applications

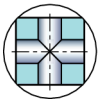

E 12
Conventional drilling

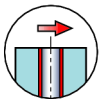

E 28
Step and chamfer


E 35
Helical Interpolation


E 34
Boring


E 20
Angled surfaces


E 20
Cross hole

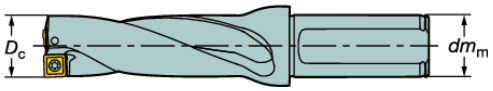

E 32
Radial adjustment


E 36
Plunge drilling


Suitable drills sizes for thread holes. See Information/Index, Chapter I.

Drill and hole tolerance





Drill depth 2–3 x D_c

Drill diameter, mm	12.00–43.99	44.00–52.99	53.00–63.50
Hole tolerance, mm	0/+0.25	0/+0.28	0/+0.30
Tolerance D_c mm	0/+0.20	0/+0.25	0/+0.28

Drill depth 4–5 x D_c

Drill diameter, mm	12.00–43.99	12.00–43.99	12.00–43.99
Hole tolerance, mm	0/+0.40	0/+0.43	0/+0.45
Tolerance D_c mm	+0.04/+0.24	+0.04/+0.29	+0.04/+0.32

Drill tolerance

The diameter tolerance of an indexable insert drill is a combination of the tipseat tolerance in the drill body and the insert tolerance. The drill + insert manufacturing tolerance are included in the hole tolerance. By using pre-setting, a closer hole tolerance can be achieved.

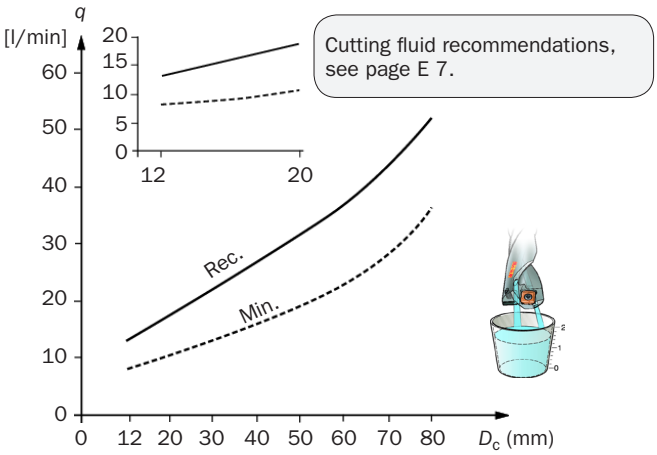
Hole tolerance

Drill length, alignment, run-out, machine stability and workpiece material influence hole tolerance and position.

For indexable insert drills, the manufacturing tolerances and the cutting force balance between peripheral and central inserts influence the hole tolerance. The CoroDrill 880 with Step Technology offers an optimal cutting force balance and a plus tolerance hole (oversize) because most holes are designed to H-tolerance. Pre-setting the drill in a lathe, or using an adjustable holder/sleeve eliminates the manufacturing tolerances of the drill body and inserts; a tolerance ±0.05 mm (IT10-11) is possible.

Graphs for CoroDrill® 880

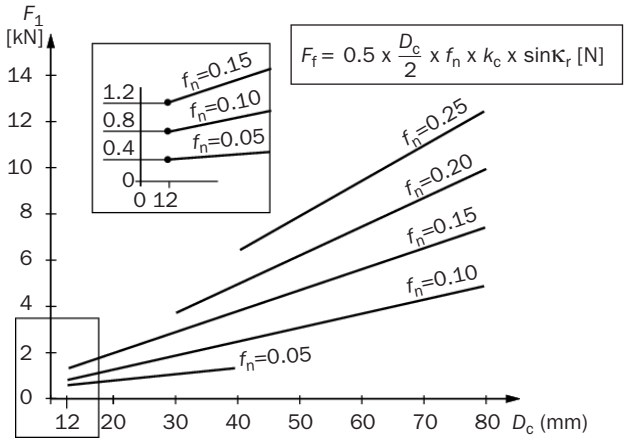
Cutting fluid flow



Cutting fluid recommendations, see page E 7.

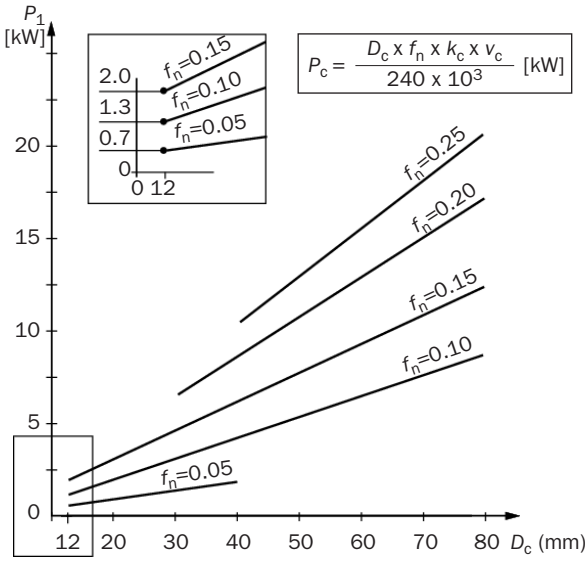
Feed force

k_c = 2500 N/mm²



Net power

k_c = 2500 N/mm² v_c = 100 m/min



Insert geometry recommendations

		Tough	First choice	Light
P	Low carbon High carbon	GT*	LM GR	GM GM
M		GT*	LM/MS***	GM
K		GT*	GR	GM
N		GT*	LM	GM
S		GT*	LM	GM
H		GT*	GM**	GM

* ***Note:*** GT geometry is available in insert sizes 02–05. Drills outside these sizes (01, 06, 07, 08 and 09) should use geometry –GR

** ***Note:*** For insert size 01, the first choice is –GR geometry

*** MS geometry available in peripheral insert sizes 02-06

Definitions

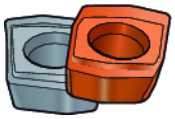
Light

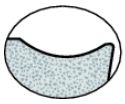
- Low feed rates
- Applications requiring low cutting forces, e.g. with long drill, 4-5 x D_c

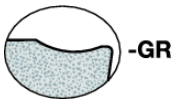
Tough

- Unstable conditions and interrupted cuts

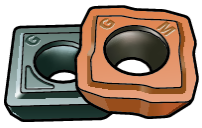
D_c mm
12.00 – 13.99

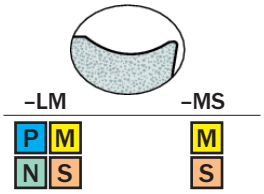


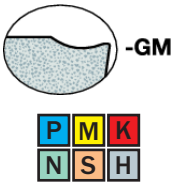


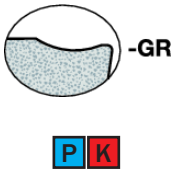


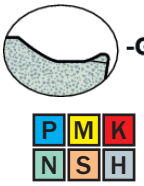
D_c mm
14.00 – 63.50



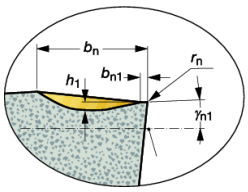








Characteristics		–LM	–MS	-GM	-GR	-GT
		Long-chipping materials Medium		**General materials Medium**	**General materials Roughing**	**General materials Tough**
		• Low to medium feed • Excellent chip control in long-chipping materials • Light cutting • Large corner radius • LM – first choice for long chipping materials • MS – sharp edge geometry optimized for stainless steels		• Low to medium feed • Light cutting • Short and shallow chip breaker provides excellent chip control in feed area • Small corner radius ensures low deflection of drill	• Low to high feed • Strong, reinforced edges • Large corner radius • Good chip control in high feed area	• Low to high feed • Very strong, reinforced edges • Good chip control in most materials • First choice for unstable conditions and interrupted cuts
		–LM	–MS			
Corner radius (peripheral insert)	r_e	Large	Large	Medium	Large	Large
Chipbreaker length	b_n	Long	Long	Short	Medium	Medium
Chipbreaker depth	h_1	Deep	Deep	Medium	Medium	Medium
Primary land width	b_{n1}	Long	Long	Short	Medium	Medium/Long
Primary land angle	γ_{n1}	Positive	Positive	Neutral	Neutral	Neutral
Edge radius (ER)	r_n	Medium	Small	Small	Large	Medium



Insert grade recommendations

Application		Toughness	First choice	Wear resistance
Peripheral insert	P	GC4044	GC4044*, GC4024	GC4014
	M	GC4044*	GC2044, GC4024	
	K	GC4044	GC4044*, GC4024	GC4014, GC4024*
	N	GC4044	H13A, GC4044*	H13A
	S	GC4044	H13A, GC4044*	H13A
	H	GC4044	GC4044	GC4024
Central insert	P		GC1044	
	M		GC1044	GC1144
	K		GC1044	
	N	GC1044	H13A, GC1044*	
	S	GC1044	H13A, GC1044*	
	H		GC1044	

P ISO P = Steel

M ISO M = Stainless steel

K ISO K = Cast iron

N ISO N = Non-ferrous material

S ISO S = Heat resistant super alloys

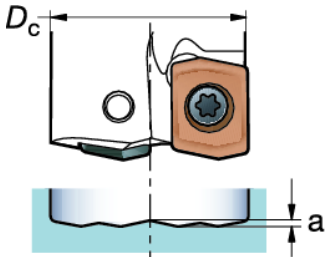
H ISO H = Hardened materials

* Recommended for smaller diameters. See Main catalogue for details.

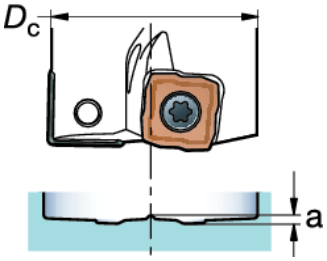
See page E 66 for more information about grades for CoroDrill 880.

Bottom profile

The bottom profile is not completely flat and varies depending on drill and insert size.



D_c 12.00–13.99 mm



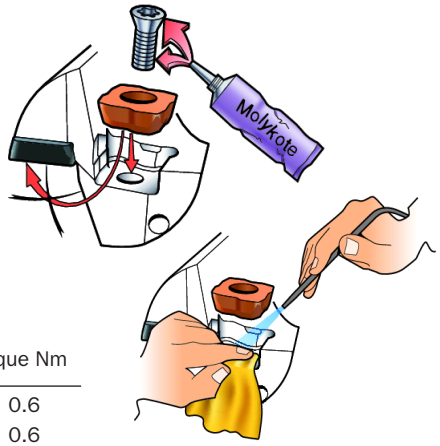
D_c 14.00–63.50 mm

Max. values per insert size:

Insert size	01	02	03	04	05	06	07	08	09
a max (mm)	0.4	0.6	0.7	0.9	1.1	1.1	1.4	1.6	2.0
D_c (mm)	12.00-13.99	14.00-16.49	16.50-19.99	20.00-23.99	24.00-29.99	30.00-35.99	36.00-43.99	44.00-52.99	53.00-63.50

Tool maintenance – indexable insert drills

Use a torque screw driver (see torque values in the table) and Molycote to ensure that the insert is securely seated. Change the insert screw regularly and ensure that the tip seat is clean and free from obstructions which could offset the insert.

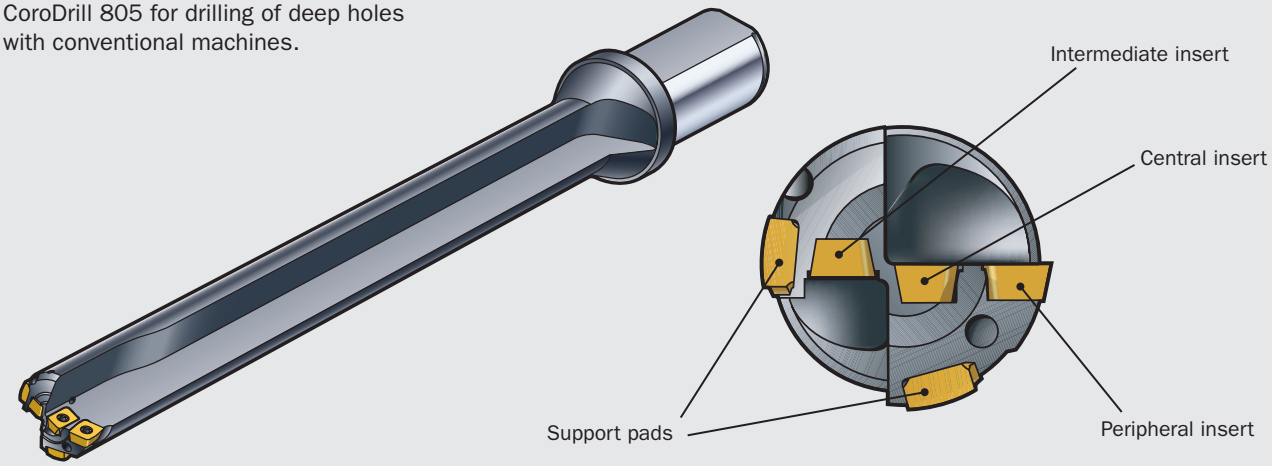


TorxPlus® torque wrench

Insert size	Insert	Screwdriver (Torx Plus)	Torque wrench	Torque Nm
880–01	5513 020–28	5680 046–08 (6IP)	5680 100–01	0.6
880–02	5513 020–28	5680 046–08 (6IP)	5680 100–01	0.6
880–03	5513 020–33	5680 046–03 (7IP)	5680 100–02	0.9
880–04	5513 020–58	5680 046–03 (7IP)	5680 100–02	0.9
880–05	5513 020–57	5680 046–04 (9IP)	5680 100–04	1.4
880–06	416.1–833	5680 046–05 (10IP)	5680 100–05	2.0
880–07	416.1–833	5680 046–05 (10IP)	5680 100–05	2.0
880–08	416.1–834	5680 046–02 (15IP)	5680 100–06	3.0
880–09	416.1–834	5680 046–02 (15IP)	5680 100–06	3.0

CoroDrill® 805

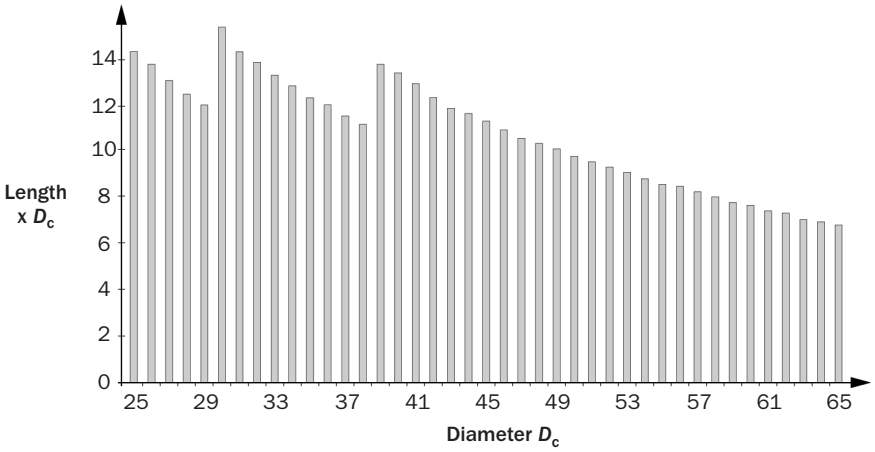
CoroDrill 805 for drilling of deep holes with conventional machines.





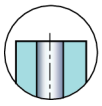
Drill diameter D_c mm	25.00–65.00
Drill depth	7–15 x D_c
Material	P M K N S H
Hole tolerance	IT10
Surface finish R_a	2 µm

CoroDrill® 805 diameter/length



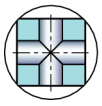
The diameter/length ratio across the range of CoroDrill 805 deep hole drills.

Applications



E 12

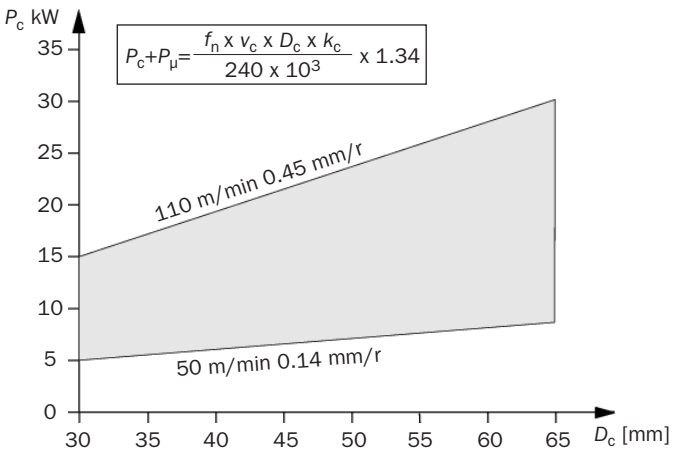
Conventional drilling



E 20

Cross hole

Feed force

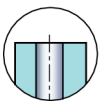
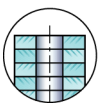
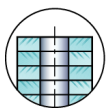


Cutting data

	ISO	Material	Cutting speed v_c m/min	Feed f_n mm/r drill diameter 25.00– 43.00 mm	Feed f_n mm/r drill diameter 43.01–65.00 mm
P	Steel	Unalloyed high alloy, hard	70–130 55–110	0.11–0.31 0.20–0.29	0.14–0.34 0.20–0.30
M	Stainless steel	Rolled/forged	40–110	0.11–0.30	0.20–0.33
K	Cast iron	Malleable nodular	80–120 50–110	0.11–0.29 0.11–0.29	0.24–0.31 0.24–0.31
N	Non-ferrous	Wrought aluminium	65–150	0.09–0.25	0.24–0.30
S	Heat resistant super alloys	Titanium	20–40	0.09–0.22	0.20–0.25

Coolant flow recommendations are minimally the same as for CoroDrill 880. See page E 51.

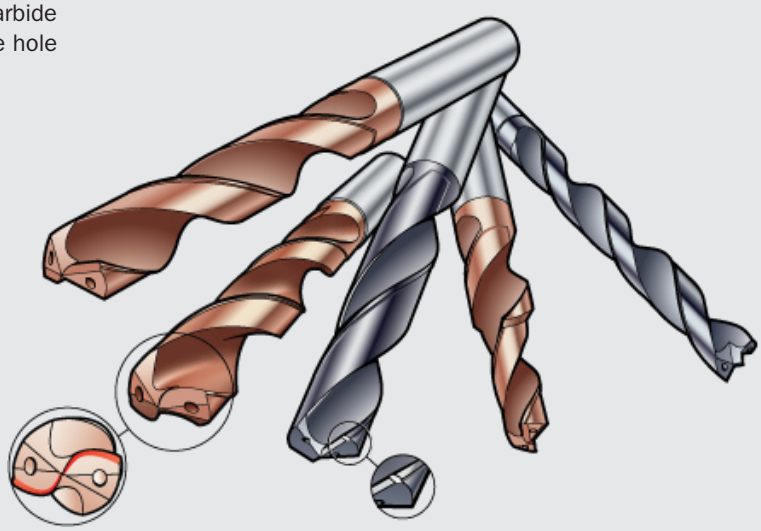
Other drills

	T-Max® U *	T-Max® U Trepanner *	Coromant U Plunge drill	T-Max® U Stack drill *
				
Drill diameter D_c mm	60.00–80.00	60.00–110.00	12.70–35.00	27.00–59.00
Drill depth	2.5 x D_c	2.5 x D_c	6 x D_c	2.5 x D_c
Material	P M K N S H	P M K N S	P M	P M K N S H
Hole tolerance	±0.2	±0.2	±0.2	±0.2
Surface finish R_a	2–7 µm	2–7 µm	1–5 µm	2–7 µm
Applications *See application section for more information.*	 E 12	  E 38 E 40	 E 36	 E 40

** See electronic catalogue for ordering information.*

CoroDrill Delta-C®

CoroDrill Delta-C, solid carbide drills for highly productive hole making.



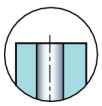

R854

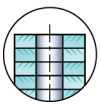

R856

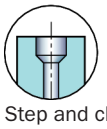
	 R840	 R841	 R842	 R844	 R846	 R850	Tailor Made and Engineered drills
Drill diameter D_c mm	0.30–20.00	0.30–20.00	3.00–16.00	8.00–18.00	3.00–16.00	5.00–14.00	up to 25.00
Nominal drill tolerance	m7	m8	m7	by request	m7	m7	by request
Drill depth	2–7 x D_c	2–7 x D_c	2–5 x D_c	1–1.5 x D_c	2–5 x D_c	2–7 x D_c	up to 15 x D_c
Material	P M K N S H	P M K N S H	K	P H	S	N	P M K N S H
Hole tolerance	IT8–10	IT8–9	IT8–10	IT5–6	IT8–10	IT8–10	IT5–10
Surface finish R_a	1–2 µm	1–2 µm	1–2 µm	0.5–1 µm	1–2 µm	1–2 µm	0.5–2 µm

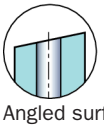
Reconditioning, see page E 64.

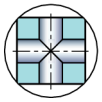
Applications


E 12
Conventional drilling

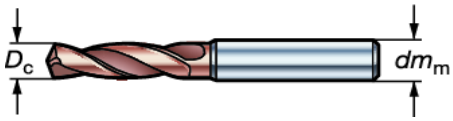

E 40
Stack drilling


E 26
Step and chamfer


E 20
Angled surfaces


E 20
Cross hole


Suitable drill sizes for thread holes. See Information/Index, Chapter I.



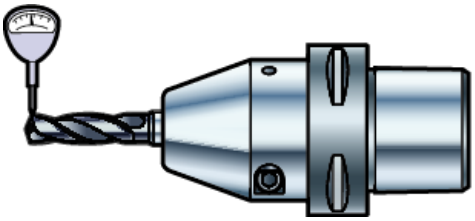
Drill and hole tolerance

Drill tolerance
Most CoroDrill Delta-C drills are manufactured acc. to DIN 6537 with the drill diameter (D_c) ground to m7 tolerance and the shank diameter (dm_m) ground to h6 tolerance. Other tolerances are available as Tailor Made options.

Hole tolerance
A drill that is ground to m7 tolerance (plus/plus) will normally produce a slightly oversized hole (max +0.04 mm). The hole tolerance, however , depends on drill length, alignment, run-out, machine stability and workpiece material.

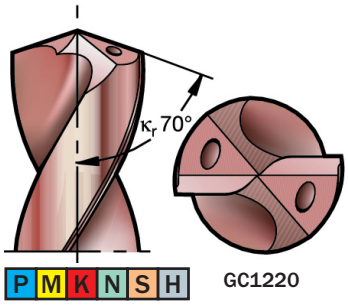
Tool holding

Tool holding and minimized run-out are essential for good drilling results with a solid carbide drill. For more information, see Tool holding/Machines, Chapter G.



Drill type recommendations

R840

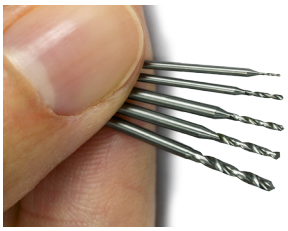


The basic choice

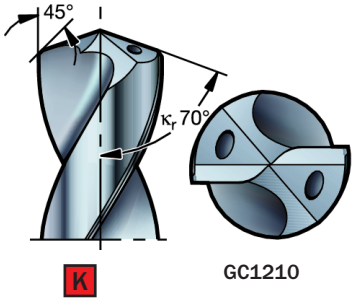
Type R840 is the all-round drill for most materials, including hard steels up to 60 HRC. Diameter range: 3.00-20.00 mm. Hole depths: 2-3 x D_c, 4-5 x D_c, and 6-7 x D_c. Easy to regrind.

Small part machining

Type R840 is also available in smaller diameters accordingly:
Diameter 1.50–2.90 mm, 4-5 x D_c, grade GC1220
Diameter 0.30–1.4 mm, 6-7 x D_c, grade H10F

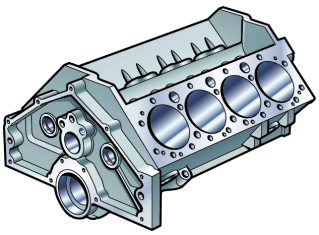


R842

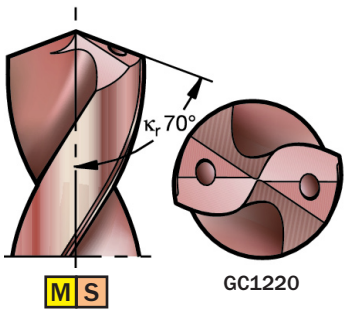


Material optimized drills

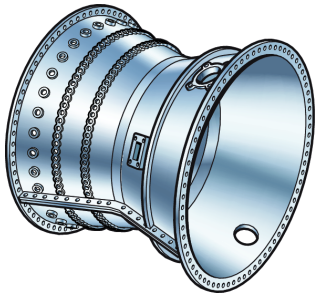
Type R842 is the first choice for machining all cast-iron materials,including CGI and ADI. Specially developed point-geometry, with corner-chamfers for edge reinforcement and for best hole quality.



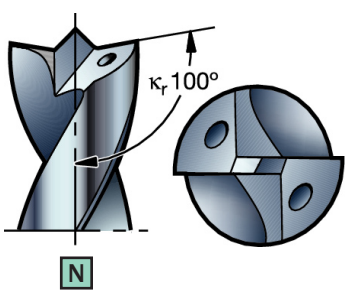
R846



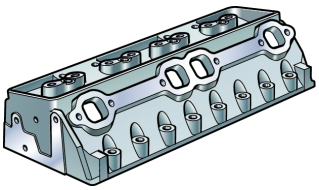
Type R846 is the first choice for ISO-S applications. First choice for Ni/Co-based superalloys; also suitable for titanium and stainless steels. Optimized convex edge geometry and a larger back taper for best chip formation and hole quality.



R850

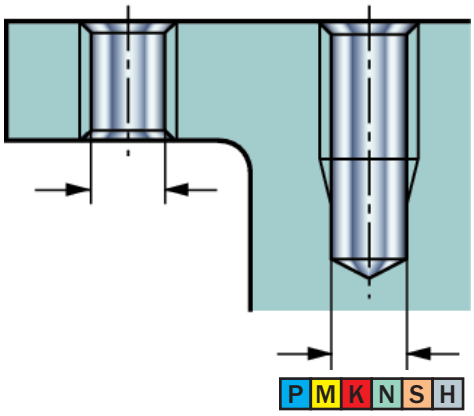


Type R850 is a drill optimized for aluminium and aluminium alloys (with a silicon content of up to 12%). Specially developed point geometry for high feeds and minimized burr formation. A productive alternative to polycrystalline diamond drills. Can also be used in copper/copper alloys.



Thread hole chamfer drill

Type R841 is a dedicated drill for making screw-thread holes in one operation. Available as standard for common thread types and dimensions.



R841



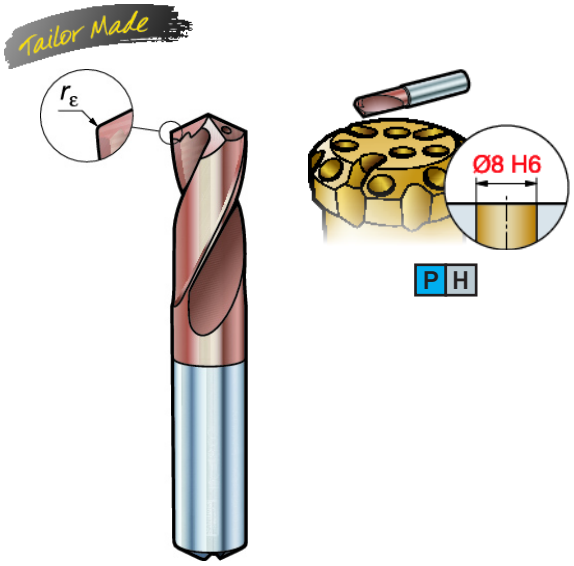
GC1220

High precision drill for hard steels

The R844 drill was originally developed for drilling rock-drill button holes and dowel holes requiring high precision holes with very close tolerances IT6.

The outer corner has a radius as an edge and hole reinforcement.

The R844 is manufactured to your request and ordered as a Tailor Made product.

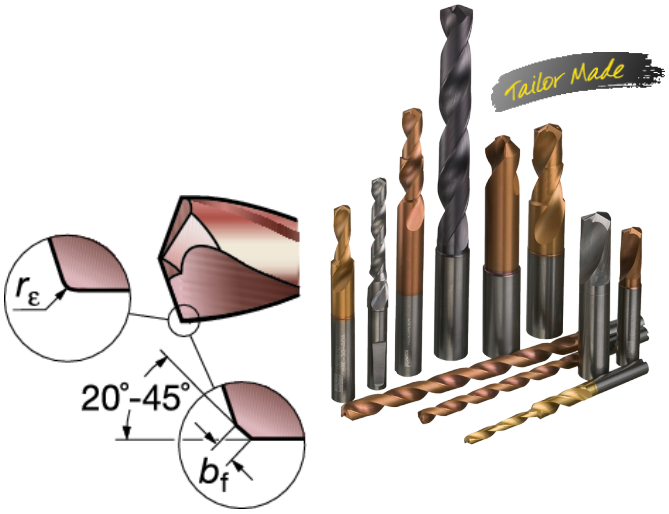


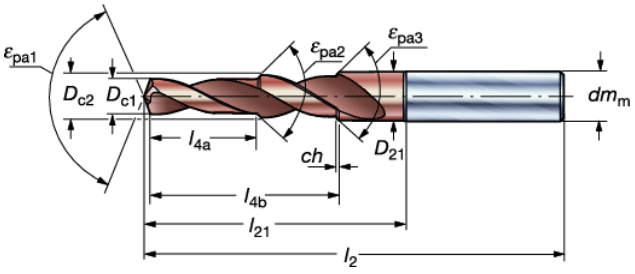
Extended offer

Tailor Made

Tailor Made offers you the opportunity to easily optimize a drill according to your specifications – within certain limits.

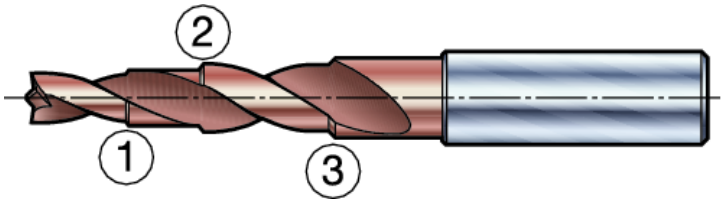
- Drill diameter
- Tolerance
- Chamfer
- Step
- Drill depth
- Corner chamfer/radius
- Back taper
- Carbide grade
- Mounting type and size
- Coolant supply





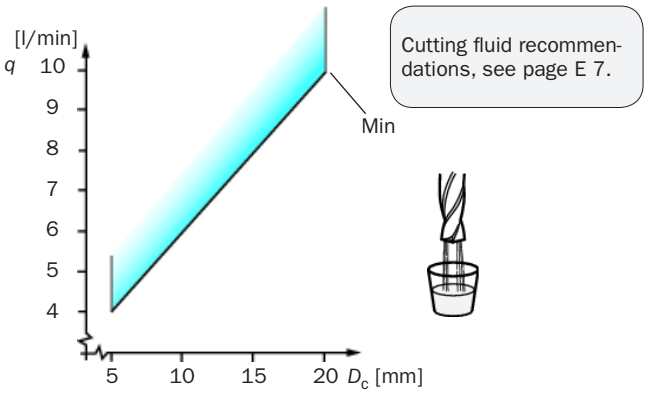
Engineered products

If your specifications fall outside the Tailor Made range, the drill can be engineered to your special request. Contact your nearest Sandvik Coromant representative.



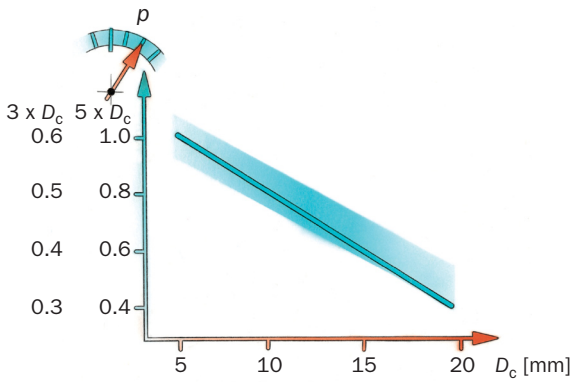
Graphs for CoroDrill Delta-C®

Cutting fluid flow

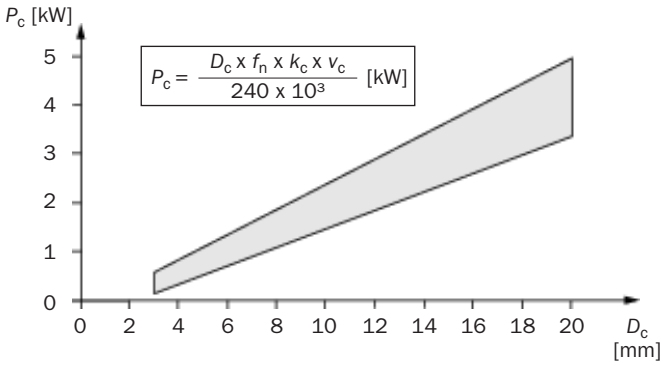


Cutting fluid recommendations, see page E 7.

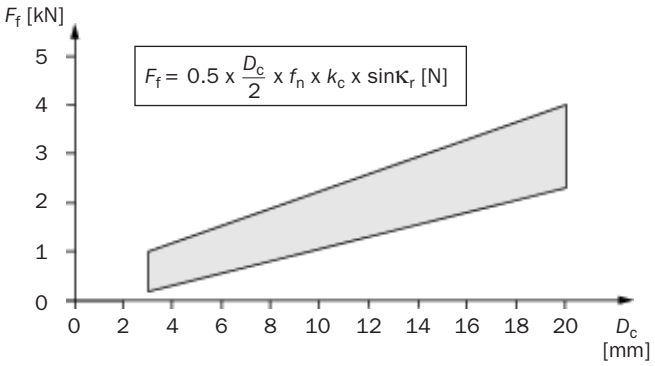
Cutting fluid pressure



Net power



Feed force



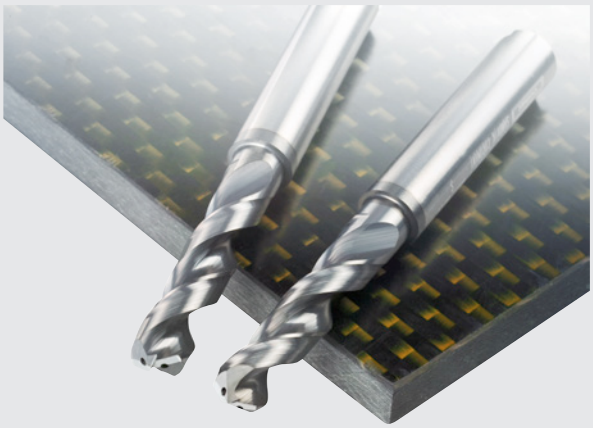
CoroDrill® 854 and CoroDrill® 856

Diamond-coated solid carbide drills
for composite materials

Geometry designed for aerospace hole requirements

Available as standard stocked items and engineered solutions

Generate the required hole quality when cutting carbon fibre



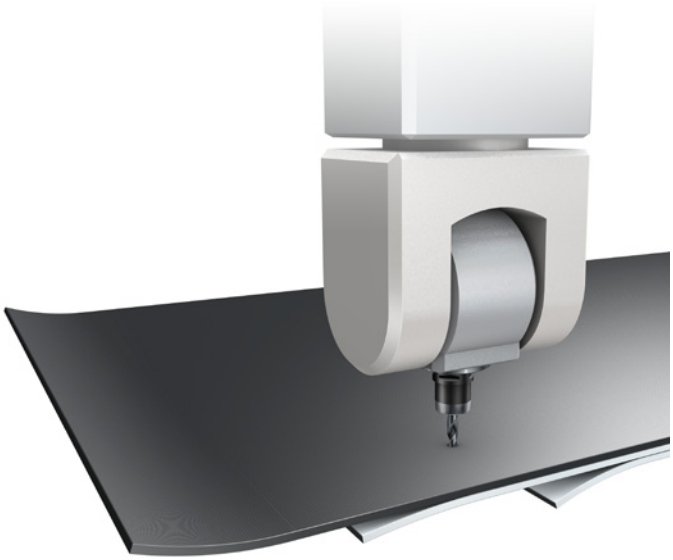
Applications

Materials

- CFRP fibre-rich materials
- CFRP/aluminium-stacked materials
- CFRP resin-rich materials
- Composite materials with, for example, glass layers: CoroDrill 856
- BMI: high-temperature resistant materials

Machines

- CNC-controlled machines
- Robots
- Portable electrical or pnuematic machines



Assortment

The standard assortment covers

- Diameters: 4 mm – ½ inch
- Lengths: up to 5xD (drilled length)
- Geometries: CoroDrill 854/856
- Grade: N20C

Drill selection guidelines

CFRP material type	Carbon fiber rich	Resin rich
Machining issue	Splintering / Fraying	Delamination
		
	CoroDrill® 854	CoroDrill® 856

CoroDrill 854 and CoroDrill 856 are available in grade N20C, a diamond-coated, solid carbide grade that provides long tool life in abrasive composite materials.



Grade selection guidelines

N20C, the first-choice standard grade

Solid carbide with standard coating

- low friction coefficient
- stable hole dimensions
- cost efficient
- not regrindable

N30C

Solid carbide with diamond-like carbon coating

- thin, wear resistant coating
- limited tool life
- low cost per tool
- regrindable

C10 (engineered)

Solid carbide with veined, PCD tip

- high hardness
- long and stable tool life
- cost efficient
- regrindable

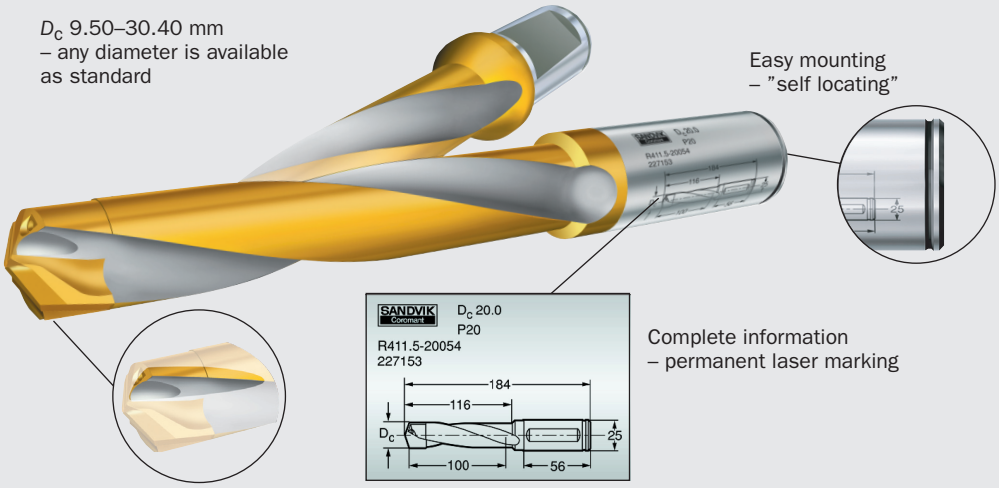
Coromant Delta®

D_c 9.50–30.40 mm
– any diameter is available as standard

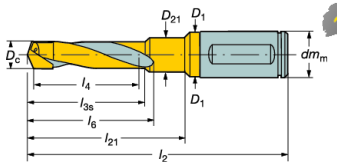
Easy mounting
– "self locating"

Regrinding service available

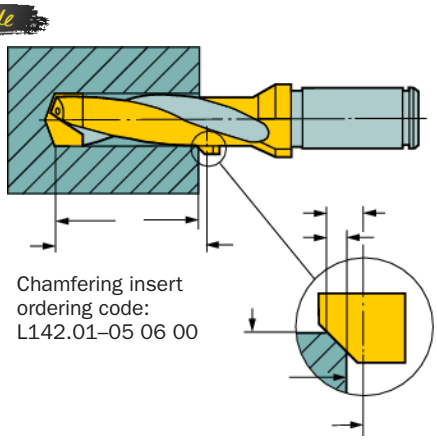
Longer total tool life
- more regrinds
- modified carbide tip
- chip flutes reinforced by laser hardening



Complete information
– permanent laser marking

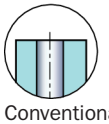
	R411.5		
Drill diameter D_c mm	9.50–30.40		9.50–30.40
Nominal drill tolerance	js7		js7
Drill depth	3.5–5 x D_c	3.5–5 x D_c	up to 10 x D_c
Grade	P20	K20	Optimized grades for different materials
Material	P	M K N H	
Hole tolerance	IT10		
Surface finish R_a	2 µm		

Reconditioning, see page E 64.



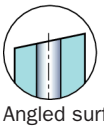
Chamfering insert ordering code:
L142.01–05 06 00

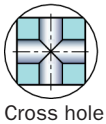
Applications

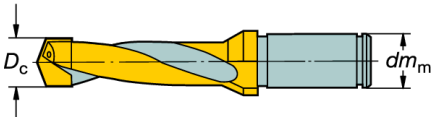

E 12
Conventional drilling


E 40
Stack drilling


E 26
Chamfer drilling


E 20
Angled surfaces


E 20
Cross hole



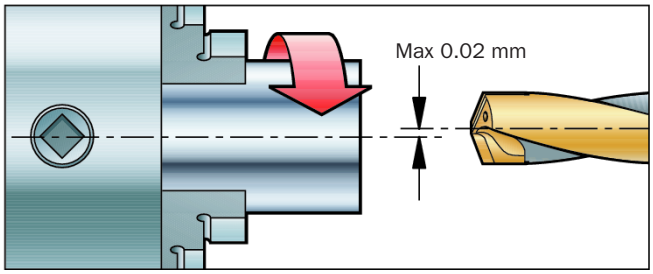
Drill and hole tolerance

Drill tolerance
Drill diameter (D_c) of Coromant Delta drills are ground to js7 tolerance, shank diameters are ground to h6 tolerance. Other tolerances are available as Tailor Made options.

Hole tolerance
A drill ground to js7 tolerance (plus/minus) will normally produce a hole very close to the nominal diameter (inside ±0.02 mm). The hole tolerance, however, depends on drill length, alignment, run-out, machine stability and workpiece material.

Drill recommendations

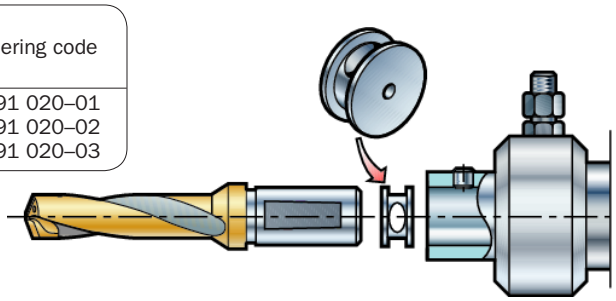
- Choose grade P20 for drilling steel and grade K20 for drilling stainless, cast iron, aluminium or very hard steels.
- Non-stocked standard drills can be ordered to a specific diameter.
- Optional grades and dimensions available as Tailor Made.



Cutting fluid volume compensator

When using a drill holder with a housing for cutting fluid supply together with a Coromant Delta drill, a coolant volume compensator should be used.

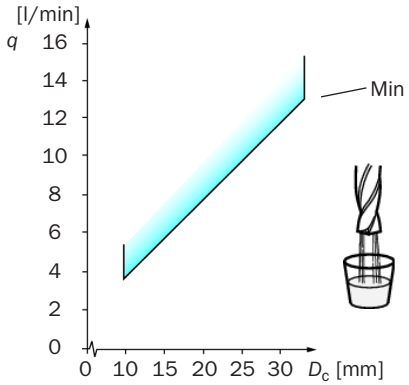
Drill diameter D_c mm	Ordering code
9.50 – 14.00	5691 020–01
14.01 – 17.00	5691 020–02
17.01 – 30.40	5691 020–03



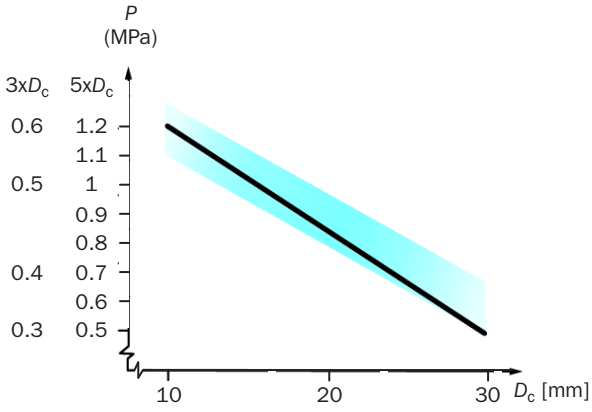
Only for Coromant Delta drills with Coromant Whistle Notch shank.

Graphs for Coromant Delta®

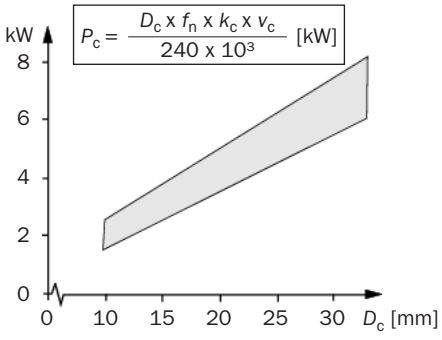
Cutting fluid flow



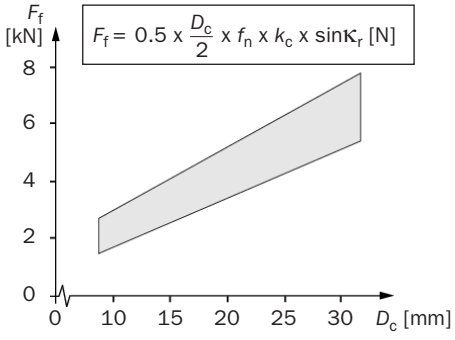
Cutting fluid pressure



Net power



Feed force



Reconditioning



Guidelines

Solid and brazed carbide drills

CoroDrill Delta-C and Coromant Delta drills can be re-ground and re-coated to extend tool life. The number of possible re-grinds depends on hole tolerance demands, workpiece, material, drill size, length and wear on the drill. Normally, a drill can be re-ground 3 to 5 times.

Original geometries must be retained to ensure best performance.

Wear before grinding should not exceed the maximum indicated. If wear exceeds these recommendations, the tool may have to be cut, causing a reduction in length at re-grinding. If the wear is too severe, reconditioning may not be possible.

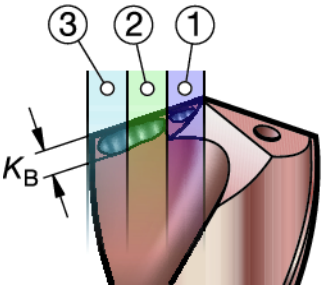
The coating disappears with re-grinding; this reduces the flank resistance. Re-coating is recommended.

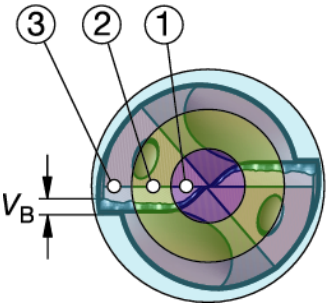
To achieve original performance of reconditioned tools drills should be sent to Sandvik Coromant for reconditioning.



Contact your Sandvik Coromant representative for more information about the reconditioning service available.

Maximum wear before re-grinding







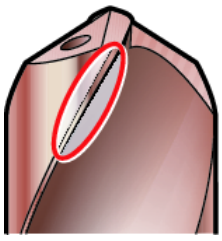
CoroDrill Delta-C®

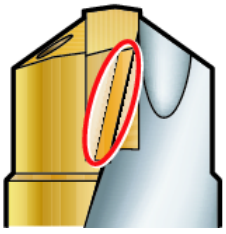
Drill dia. mm	Flank wear, V_b (mm) Zone 1	2	3	Crater wear, K_b (mm) Zone 1	2	3
3.00-6.00	0.20	0.20	0.20	0.20	0.20	0.20
6.01-10.00	0.20	0.20	0.25	0.25	0.25	0.25
10.01-14.00	0.25	0.25	0.25	0.30	0.30	0.30
14.01-17.00	0.25	0.25	0.30	0.30	0.30	0.30
17.01-20.00	0.30	0.30	0.35	0.35	0.35	0.35



Coromant Delta®

Drill dia. mm	Flank wear, V_b (mm) Zone 1	2	3	Crater wear, K_b (mm) Zone 1	2	3
9.50-14.00	0.25	0.25	0.25	0.30	0.30	0.30
14.0-17.00	0.25	0.25	0.30	0.30	0.30	0.30
17.01-20.00	0.30	0.30	0.30	0.35	0.35	0.35
20.01-24.00	0.30	0.30	0.40	0.35	0.35	0.35
24.01-30.40	0.35	0.35	0.45	0.40	0.40	0.40



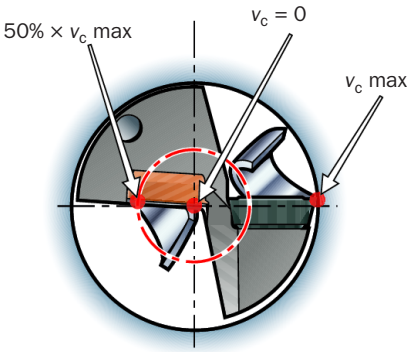


Make sure the margin portion of the drill is not damaged, chipped or cracked.

Grade information



CoroDrill® 880



Central insert grades

Grade GC1044 is the first choice for the central insert in all materials, except aluminium where H13A is preferred.

Peripheral insert grades

For the peripheral insert, there is a wider choice of grades because the cutting speed and conditions vary more. GC4044 is the toughest choice and GC4014 the most wear resistant.

GC1044 P M K N S H



The basic choice for central drilling inserts in all materials.
Fine-grained cemented carbide with an excellent combination of both hardness and toughness. PVD-coated with a 3-micron bronze colored TiAlN multi-layer provides excellent edge toughness and resistance to built-up edge.

GC1144 M S

Central insert grade for stainless steel
A fine-grained cemented carbide with good balance between toughness and wear resistance. A new PVD-oxide coating provides excellent wear resistance and resistance against built up edge in all types of stainless steels and as complementary in ISO S materials.

H13A N S



For heat resistant alloys, titanium and aluminium. Suitable both as the central and peripheral insert.
H13A is an uncoated and fairly fine-grained grade with good edge sharpness. Very good balance between wear resistance and toughness.



GC2044 M S

Peripheral insert grade for stainless steel
Based on fine-grained cemented carbide coated with a new PVD-oxide coating for excellent wear resistance and resistance against built up edge in all types of stainless steels and as complementary in ISO S materials.



GC4044 P M K N S H

The tough all-round choice for peripheral drilling inserts in all materials.
Fine-grained cemented carbide with an excellent toughness. The carbide is PVD-coated with a 3-micron black colored TiAlN layer for improved wear resistance and resistance against built-up edge.



GC4034 P M K

A more wear resistant peripheral insert alternative to GC4044 for steel, stainless and cast iron.
A grade with very good balance between toughness and wear resistance, MT-CVD coated with special treatment, improving edge line toughness and minimizing built-up-edge formation.



GC4024 P M K H

The productive peripheral insert choice for stable conditions.
A carbide substrate with a good balance between hardness and toughness, coated with a MT-CVD layer of TiCN followed by a layer of Al_2O_3 for high temperature resistance.



GC4014 P K

The peripheral insert choice for high cutting speeds at low to moderate feeds. Good plastic deformation resistance.
A hard carbide substrate with a thin cobalt-enriched gradient zone for edge toughness. On top of this is a MT-CVD layer of TiCN and Al_2O_3, providing very good high temperature protection, permitting high speeds.

Cutting data recommendations, see Main catalogue.

For recommendation overview, see page E 53.

SANDVIK Coromant

CoroDrill Delta-C® and Coromant Delta®

CoroDrill Delta-C – solid carbide drill

The basic choice is the universal GC1220. Optimized grades are available for cast iron (GC1210) and aluminium (GCN20D). GC1020 and H10F are dedicated for small diameters. For further optimization, different PVD-coatings are available as Tailor Made.

Coromant Delta – brazed carbide drills

The basic choice is grade P20 for steel applications and grade K20 for other materials. Grade H10F and different PVD coatings are available as Tailor Made.

GC1220 P M K N S H



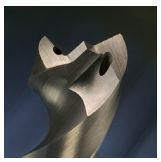
First choice grade for steel, stainless steels, HRSA and titanium, but can be used in all materials.
Fine-grained cemented carbide with an excellent combination of both hardness and toughness. The carbide is PVD-coated with 3-microns of nano-multilayers TiAlN coating, giving very good edge line security.

GC1210 P K



First choice in cast iron and a wear resistant alternative for steel.
Hard and very wear resistant carbide substrate. The carbide is coated with AlCrN, providing excellent wear resistance and resistance against plastic deformation.

GC N20D N



First choice for aluminium alloys with a silicon content up to 12%.
A fine-grained cemented carbide substrate. The fine grains maintain the sharp cutting edge throughout the entire tool life. PVD coated with smooth TiAlN adding wear resistance and decreasing built-up-edge formation.

P20 P



First choice for steel alloys.
A wear resistant and tough substrate PVD-coated with a low-friction TiN-layer.

K20 M K N S H



First choice for stainless steel, cast iron, aluminium and heat resistant materials.
PVD-coated with a low-friction TiN-layer on a tough WC-Co substrate.

Application		Toughness	First choice	Wear resistance
CoroDrill Delta-C	P	GC1220	GC1220	GC1210*
	M	GC1220	GC1220	
	K	GC1220	GC1210	
	N	GC1220	GC N20D	
	S	GC1220	GC1220	
	H	GC1220	GC1220	
Coromant Delta	P		P20	K20
	M	P20	K20	
	K		K20	
	N		K20	
	S	P20	K20	
	H	P20	K20	

* Tailor Made

Cutting data recommendations, see Main catalogue.