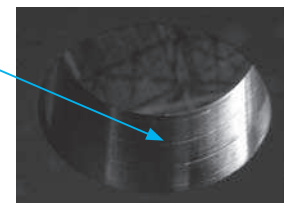
Q&A

・[High rate + Small feed]
This setting makes the chips thinner and uses centrifugal force to cut them off.
E.g. Vc=200m/min、f=0.07 ～ 0.09mm/rev
・[Step machining at entrance]
E.g. Entrance to 10 mm deep: 1 mm step machining
E.g. 10 mm deep or more: Vc = 150 m/min, f = 0.15 mm/rev (Continuous machining)

Q-4 Tool markings are made on the finished surface. Is there any countermeasure?

A-4 During processing, force of deflection is applied to the center of the drill.
If the drill is just pulled out from the position where processing is finished, tool markings will be made.
To prevent tool markings, perform offset before pulling out the drill.

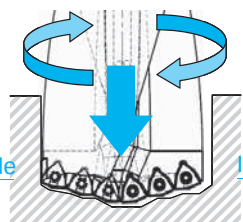
Tool marking



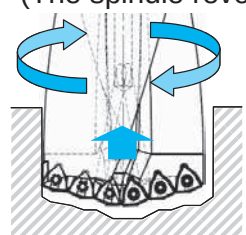
Example of tool marking

・How to prevent tool markings

① Drill the hole. (The spindle revolves.)



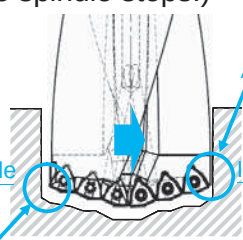
② Turn back approximately 0.5 mm. (The spindle revolves.)



Chips are adhering to the bottom when drilling stops.

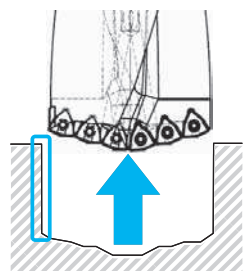
·Without turning back, chips remains adhering to the bottom.
·If offset ③ is performed without turning back, the bottom of drill contacts with the inner surface of hole.
·Turning back is necessary for blind holes but not for through holes.

③ Stop revolution and perform offset. (The spindle stops.)



Make a clearance to prevent the tool from contacting when pulling out.
(Approximately 0.1 to 0.2 mm)

④ Pull out the drill.



Tool markings are not made (or are only slight even if made).

Example of drilling program

```
G90G54G0G43X0Y0Z100.0H10
S477M03
Z2.5M8
G01Z-80.0F48
Z-79.5M19   ← The spindle stops at the specified position.
X0.2Y0.2
Z100.0M9
```

* The M code and X and Y moving directions are unique to the equipmen

Q-5 Chattering occurs. Is there any countermeasure?

A-5 Chattering usually occurs during chamfering and when the feed rate per revolution is not high enough.
Try changing the drilling conditions as follows.

・Increase the feed rate if it is small.
If the feed rate is f = 0.06 mm/rev, for example, increase it to f = 0.08 to 0.12 mm/rev.
Increasing the feed rate will improve chamfering and thus prevent chattering.

・If the cutting speed is too high, lower it to Vc = 100 to 150 m/min.

・If the chamfering point and pass-through point are not plain, or if the workpiece clamping rigidity is low, lower the feed rate to f = 0.07 ~ 0.08 mm/rev.

・If chattering occurs on the full contact surface (e.g. during step machining), make adjustments by increasing the feed rate during chamfering or lowering the cutting speed.
Once chattering occurs during chamfering, it will continue throughout the drilling.