



HEIDENHAIN



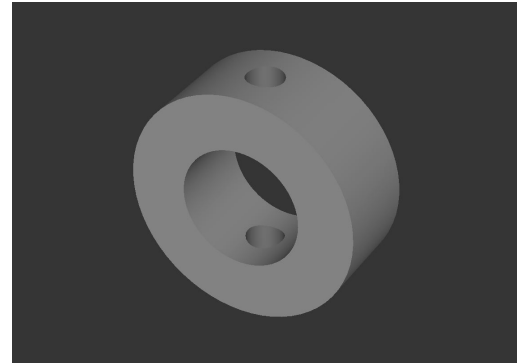
NC Solutions

Description of NC program 1060

English (en)
4/2017

1 Description of NC programs 1060_en.h and 10601_en.h

NC programs for cross-drilling of a tube with the possibility of defining different spindle speeds and feed rates.



Description

You define the tool and parameters required for the sequence in a main program (1060_en.h). With Cycle 12 the control then calls a further NC program (10601_en.h) at the hole position. The actual drilling sequence is contained in this NC program. Because this process is executed with a cycle call, the drilling sequence can be performed at various positions. The parameters must not be re-entered for this purpose.

Drilling sequence

- 1 The control positions the tool in rapid traverse at the position defined in the main program at safety clearance above the surface coordinate, and then activates the defined speed for spot drilling.
- 2 The tool moves to the surface. The feed rate is 25 % of the programmed feed rate in the air.
- 3 Spot drilling starts with 10 % of the drilling feed rate. The control increases the feed rate to 100 % along the length of the defined acceleration path.
- 4 The control modifies the speed to the value defined for drilling and machines through the upper wall at the defined feed rate. The depth moved to by the control is defined by the wall thickness of the tube and the defined overrun. The purpose of the overrun is to ensure that the tool drills through the wall with the entire diameter and therefore depends on the tool diameter, its point angle and the inside diameter of the tube.
- 5 At the programmed feed rate in the air, the control positions to safety clearance above the second wall. From there it continues to machine with the drilling feed rate.
- 6 Before the tool tip reaches the outside diameter of the tube, the control reduces the feed rate to 10 % of the programmed value along a deceleration path. The control machines deeper about the overrun with the second wall as well to ensure a complete through-hole.
- 7 The control then retracts the tool again to safety distance above the plane.

1060_en.h NC program

In the main program 1060_en.h and after the tool call, Cycle 12 PGM CALL is programmed in which the NC program 10601_en.h is defined. This enables a simple call via CYCL CALL or M99. You then define all parameters required for the drilling sequence.

The control then approaches the first hole position and calls Cycle 12 in which the NC program for the drilling sequence is defined. This could be followed by further positions with a cycle call. In the example the control retracts the tool after a machining step and terminates the program.

Parameter	Name	Meaning
Q1	SAFETY CLEARANCE	Z clearance between the tool and workpiece surface approached by the control in rapid traverse before machining is executed
Q2	COORDINATE SURFACE	Z coordinate of the workpiece plane
Q3	LENGTH OF ACCELERATION/ DECELERATION PATH	Path length along which the control slowly increases or reduces the feed rate
Q4	WALL THICKNESS OF TUBE	Wall thickness of the tube to be drilled
Q5	DIAMETER OF TUBE	Outside diameter of the tube to be drilled
Q6	FEED RATE FOR DRILLING	Traverse speed of the tool during drilling
Q7	FEED RATE IN THE AIR	Traverse speed of the tool between the tube walls
Q8	ROTATION SPEED FOR SPOT DRILLING	Spindle speed during the spot drilling sequence
Q9	ROTATION SPEED FOR DRILLING	Spindle speed during drilling
Q10	OVERRUN	Path length along which the control extends the tool movement during machining to ensure a complete through-hole

10601_en.h NC program

In the 10601_en.h NC program the control executes all calculations and path contours required for the drilling sequence.

Do not change this NC program if you do not wish to modify the drilling sequence. All required parameters are defined in the main program.

- 1 The control defines the spot drilling speed with a tool call.
- 2 The control computes the individual depths at which the feed rate changes and the increments for these feed rate changes during the acceleration and deceleration paths.
- 3 Following pre-positioning, the control traverses the acceleration path in LBL 2. This means that the control slowly increases the depth and feed rate until the end of the path with the full feed rate (100 %) has been reached.
- 4 The control switches to drilling speed and approaches the individual depths with the appropriate feed rates. The control traverses the deceleration paths inversely to the previous acceleration in LBL 3.
- 5 The control retracts the tool after the deepest position has been reached. The NC program terminates and the control returns to the main program.

