HEIDENHAIN
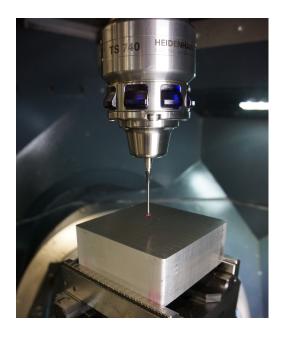
NC Solutions

Description of NC program 8230

1 Description of NC program 8230_en.h

In this NC program, the control measures the Z coordinate at various positions and takes this coordinate into account during a later machining operation.

Scenario:

A workpiece is to be machine at various positions by means of a cycle. The positions in the X/Y plane are defined in a point table. The depth defined in the machining cycle is referenced to the exact surface of the respective position. The workpiece surface has not yet been machined; this is why the exact Z coordinate of the surface must be determined at each machining position. This coordinate must then be taken into account during execution of the cycle.

NC program 8230_en.h:

In NC program 8230_en.h, the application is programmed for the side of a cube—as an example.

The X coordinates and Y coordinates of the machining positions are defined in point table 82301_en.pnt.

In the NC program, the control first calls the touch probe. The control then opens the point table by means of the **FN26: TABOPEN** function. Next, you define how many positions the table should contain using parameter Q52.

The control then calls the "MEASURE" subprogram. The beginning of the subprogram specifies that the first line of the table to be read is line 0. The control also calculates the number of required repetitions for the measurement routine.

The control then reads the X coordinate and Y coordinate from line zero of the table. The control executes measurement cycle 427 at the position that has been read and measures the coordinate of the Z axis. By default, the measurement result is saved by the control in parameter Q160. The control then writes the result to the "Z" column. After this, the control increments the number of the table rows to be read out and written to by one and repeats the section of the subprogram. This loop is repeated by the control until all of the positions have been measured. The control then ends the subprogram and jumps back to the main program.

In the main program, the control then calls the next tool for the machining operation—in this example, a drill. After this, the machining cycle is defined, followed by the point table by means of the **SEL PATTERN** function. Next, the control prepositions the tool and then calls the machining cycle with **CYCL CALL PAT** at all positions of the table. The Z coordinates entered in the table act as a coordinate-defined surface, so that the machining depth is referenced to the respective Z coordinate.

In the final step, the control moves the tool to a set-up position. Since the location of this position depends on the machine, you will need to adjust the coordinates in the "SAFE" subprogram accordingly. The control then ends the NC program.

□50

28.284

⌀9

A A

A A

1x45°

744 650 A4

50

2

A-A

1:2

ID number

Text:

	Change No.	C000941-10	
	Phase:	Nicht-Serie	

	Original drawing		Klotz	Werkstoff: Material:
	Scale	Format		
RoHS	1:1	A4	Block	

Maße in mm / Dimensions in mm	Einzelteilzeichnung / Component Drawing		●blanke Flächen/Blank surfaces

Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715	Allgemeintoleranzen ISO 2768-mH ≦6mm: ±0,2 General tolerances ISO 2768-mH ≦6mm: ±0,2	Tolerierung nach ISO 8015 Tolerances as per ISO 8015	Oberflächen nach ISO 1302 Surfaces as per ISO 1302
-0.3 +0.3		Oberflächenbehandlung: Surface treatment:	

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
83301 Traunreut, Germany

	Created	Responsible	Released		Version	Revision	Sheet	Page
	Feichtner			D1245445-00-A-01				1 of 1
	16.02.2018			Document number				