



HEIDENHAIN



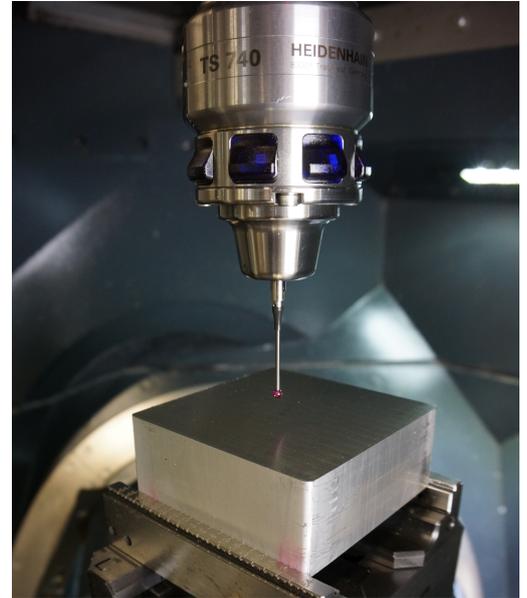
NC solutions

Description of NC program 8235

English (en)
6/2018

1 Description of NC program 8235_en.h

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



Scenario:

Five sides of a cube are to be machined at various positions with a cycle. The positions in the X/Y plane are defined separately for every side in a point table. The depth defined in the machining cycle is referenced to the exact surface of the respective position. The workpiece surfaces have 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:

Measuring:

In NC program 8230_en.h, the application is programmed based on an example involving five sides of a cube.

For each side of the cube, a point table is defined with the X coordinates and the Y coordinates of the machining positions.

In the NC program, the control first calls the touch probe. Then the control rescinds the tilt of the working plane. After this, the control shifts the datum to the first side to be measured and tilts the working plane to this side.

With the function **FN26: TABOPEN**, the control then opens the point table of the side to be machined. Next, you use parameter Q52 to define how many positions the table should contain.

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.

Then the control reads the X coordinate and the Y coordinate from line zero of the table. The control performs measuring 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 in the "Z" column of the point table. After this, the control increments the number of 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.

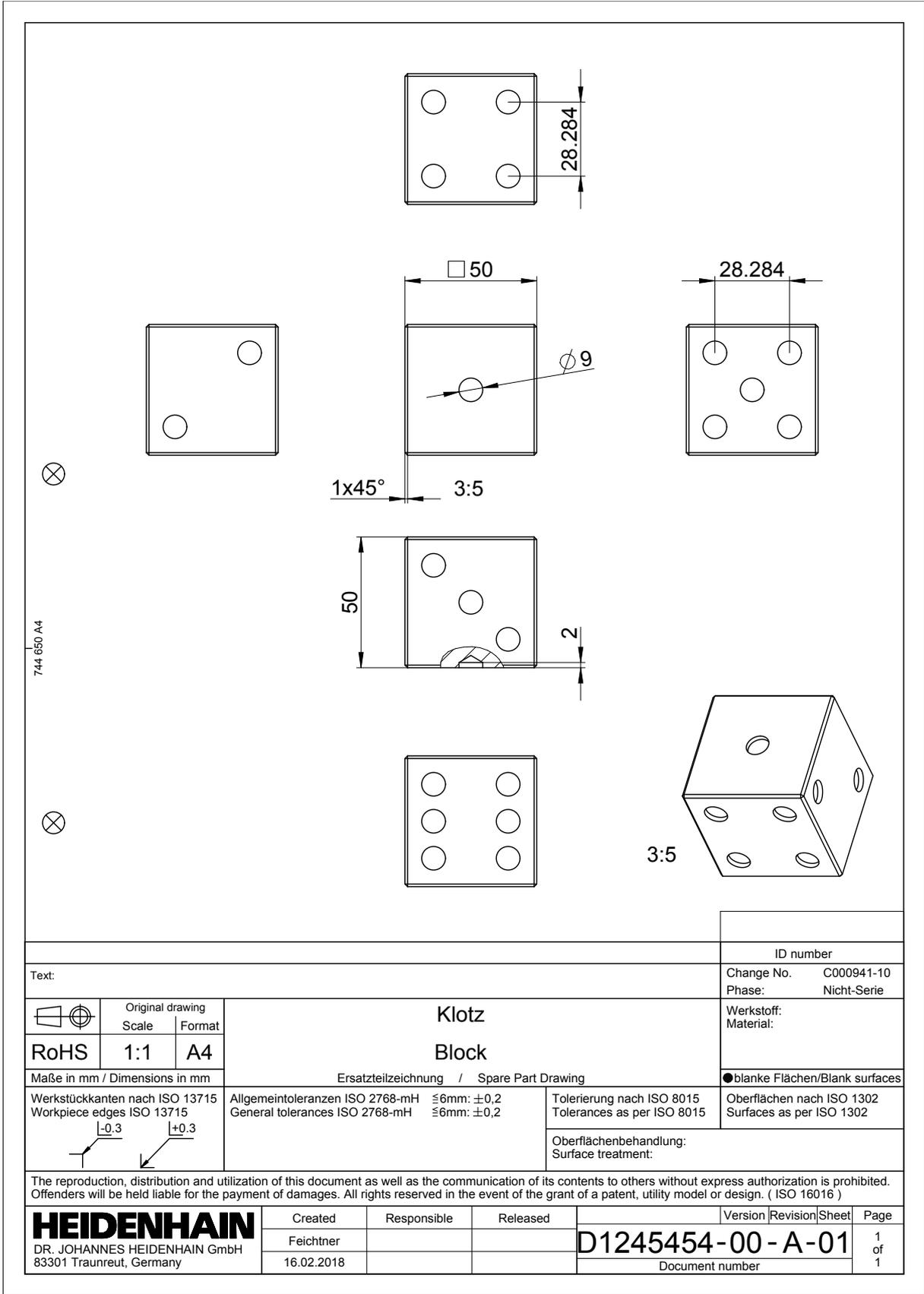
After jumping back to the main program, the control resets the tilt of the working plane and shifts the datum to the second side. Then the control tilts the working plane to this side. You then once again define the table to be used for this side and, in parameter Q52, the number of positions contained therein. After this, the control once again calls the "MEASURE" subprogram. This sequence is then programmed in likewise manner for the other three sides.

Machining:

When the measuring program has been completed for all five sides, the control then calls the tool for machining in the main program. In this example program, a drill is defined. After this, the machining cycle is first defined, and then the control resets the tilt. The control then shifts the datum to the first side and tilts the working plane. In the next step, the point table for the first side is defined with the function **SEL PATTERN**. 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 serve as coordinate surfaces, so that the machining depth is referenced to the respective Z coordinate.

When the first side has been machined, the same routine is programmed for the other four sides.

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.



Text:		ID number
Change No. C000941-10		Phase: Nicht-Serie
Werkstoff: Material:		
Maße in mm / Dimensions in mm		●blanke Flächen/Blank surfaces
Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715	Allgemeintoleranzen ISO 2768-mH General tolerances ISO 2768-mH	Tolerierung nach ISO 8015 Tolerances as per ISO 8015
$\begin{matrix} \swarrow -0.3 \\ \searrow +0.3 \end{matrix}$	$\leq 6\text{mm}: \pm 0,2$ $\leq 6\text{mm}: \pm 0,2$	Oberflächen nach ISO 1302 Surfaces as per ISO 1302
Oberflächenbehandlung: Surface treatment:		
The reproduction, distribution and utilization of this document as well as the communication of its contents to others without express authorization is prohibited. Offenders will be held liable for the payment of damages. All rights reserved in the event of the grant of a patent, utility model or design. (ISO 16016)		
HEIDENHAIN DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany	Created	Responsible
	Feichtner	
16.02.2018	Released	
Version		Revision
D1245454-00 - A-01		Sheet
Document number		Page
		1 of 1