



HEIDENHAIN



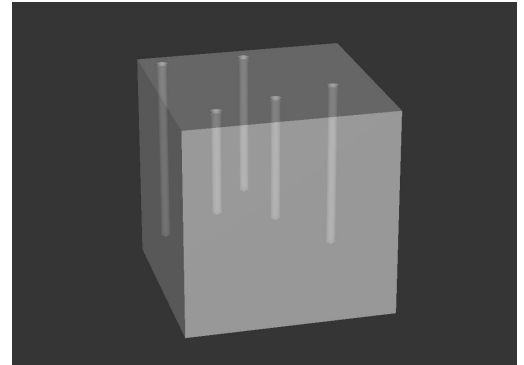
NC Solutions

Description of NC program 1065

English (en)
4/2017

1 Description of NC programs 1065_en.h, 10651_en.tab and 10652_en.h

NC program for defining a nonsymmetrical point pattern with various hole depths. At the machining positions the control executes one machining step in each case for centering, generating a pilot hole and generating a hole.



Description

The control creates a point pattern with this NC program. You define the machining positions, depths, safety clearances and feed rates in a table. At the positions the control calls a drilling cycle and the NC program 10652_en.h. The control machines holes with this NC program using a sequence for deep-hole drilling.

1065_en.h NC program

At the start of the program you define the path of the table from which the control reads the positions, depths and other parameters. You then define the centering tool, the machining cycle for generating the center holes and two parameters. These parameters specify which lines in the table are read by the control. The control then jumps to a subprogram. In this subprogram the control reads the required values out of the table, approaches the positions and executes the machining. When the subprograms are executed the control returns to the main program.

Here you define the tool for machining the pilot holes and the machining cycle required for this. In the example program, only the depth parameter has been reset. The control takes all other parameters from the previous cycle. You can also define a complete cycle here. After the definition, the control then jumps again to the subprogram and generates the pilot holes.

For generating the deep holes you define the tool in the main program and several parameters. The sequence for deep-hole drilling is programmed in the NC program 10652_en.h. You then define the program path from this NC program in Cycle 12, enabling this NC program to be started with a cycle call. The control also jumps into the subprogram for this third machining, approaches the positions and executes the machining steps. As the last step in the program the control retracts the tool and terminates the program.

Parameters for NC program 1065_en.h

Parameter	Name	Meaning
Q51	START LINE	Line number of the first table line from which the parameters are read
Q52	FURTHER LINES	Number of additional table lines from which the parameters are read
Q1	SURFACE	Z coordinate of the upper surface of the hole, referencing the workpiece zero point
Q2	PILOT DRILLING DEPTH	Depth of pilot drilling incremental to the surface
Q3	CHIP REMOVAL HEIGHT	Retraction height for chip removal incremental to the surface
Q4	FEED RATE FOR PLUNGING	Feed rate for entering the pilot hole
Q5	FEED RATE FOR DRILLING	Feed rate during drilling
Q6	SPINDLE SPEED	Speed during drilling
Q7	INFEED	Incremental infeed per drilling step
Q8	DEPTH	Depth of hole incremental to the surface
Q9	DWELL TIME	Time in seconds during which the tool dwells at chip removal height
Q10	CHIP BREAKING VALUE	Incremental value at which the control retracts the tool for chip breaking if the plunging depth was reached
Q11	SAFETY VALUE 1	Coordinate at which the control pre-positions the tool. Incremental to the surface
Q12	SAFETY VALUE 2	Coordinate at which the control positions the tool after machining. Incremental to the surface
Q13	NUMBER OF INFEEDS	Number of infeeds until chip removal

Table 10651_en.tab

The table 10651_en.tab used is a freely definable table. You can add further lines to this table if required. You define the values for a machining step in each line. When a line is read, the control may overwrite parameters that were defined in the NC program.

Define the following values in the table:

- X coordinate of machining
- Y coordinate of machining
- Z coordinate of the workpiece plane
- Drilling depth
- Safety clearance for pre-positioning
- Feed rate for drilling
- Clearance height after machining

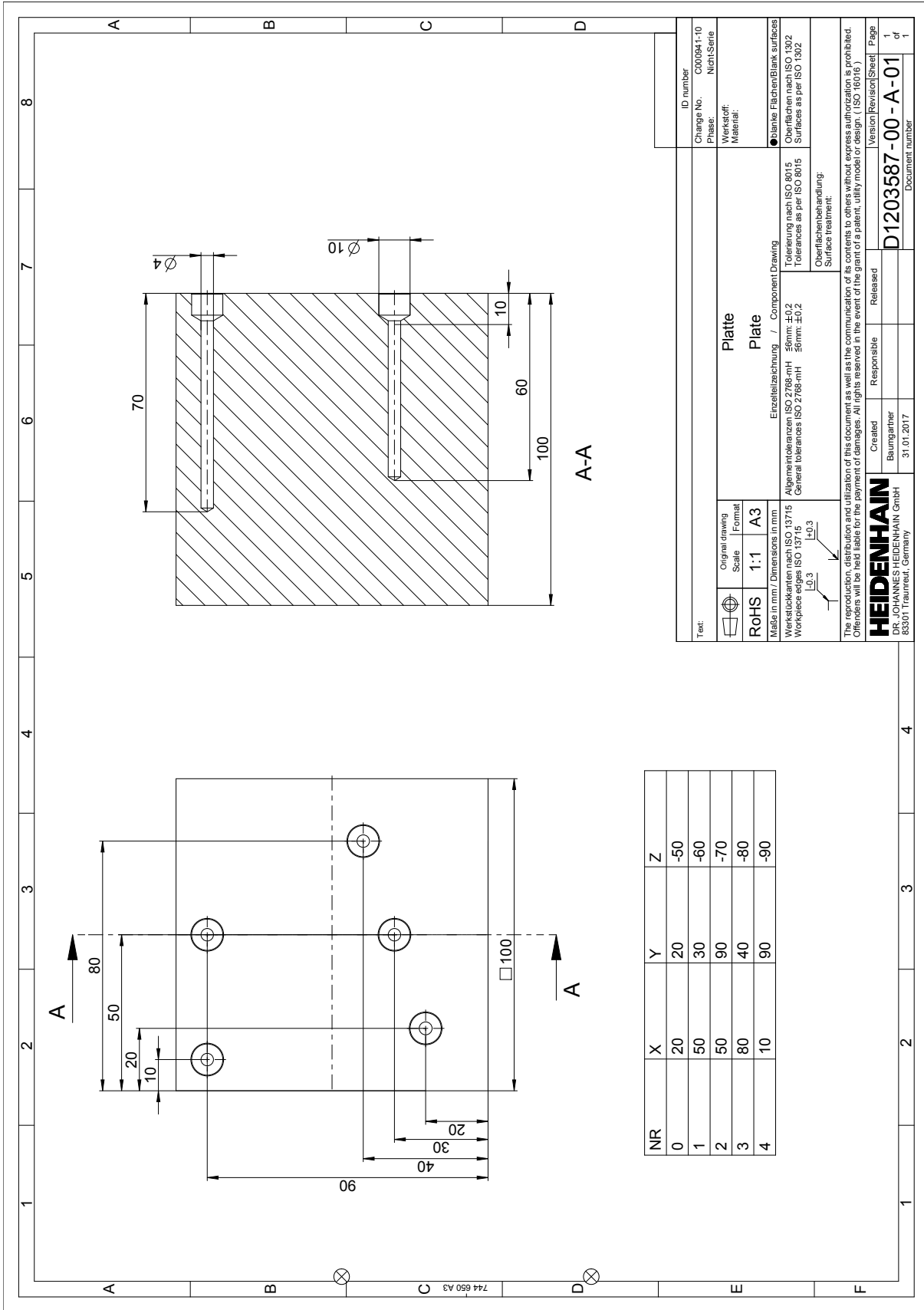
NR.	X	Y	Z	Depth	Distance	Feed_rate	Safe_height
1	50	30	0	-60	2	100	10
2	50	30	0	-70	2	100	10
3	60	40	0	-80	2	100	10
4	10	90	0	-90	2	100	10



If you create the freely definable table yourself, make sure that all column names start with a letter.

10652_en.h NC program

In the 10652_en.h NC program the control executes all calculations and path contours for the drilling sequence. You do not need to change anything in this program because all required parameters are defined in the main program and the control reads the parameters from the table.



NR	X	Y	Z
0	20	20	-50
1	50	30	-60
2	50	90	-70
3	80	40	-80
4	10	90	-90

Text:

Original drawing	Scale	Format	ID number
RoHS	1:1	A3	Change No. C000941-10
Maße in mm / Dimensions in mm	Platte		
Werkstoffkennzeichen ISO 13715	Platte		
Workpiece edges ISO 13715	Einzelteilzeichnung / Component Drawing		
General tolerances ISO 2768-mS	Tolerierung nach ISO 8015		
General tolerances ISO 2768-mH	Tolerances as per ISO 8015		
Surface treatment:	Oberflächenbehandlung:		
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)			
Created		Responsible	Released
BAUNGARTNER			
31.01.2017			
Version		Revision/Sheet	Page
D1203587-00-A-01			1
Document number		of 1	

HEIDENHAIN
 DR. JOHANNES HEIDENHAIN GmbH
 83301 Traunreut, Germany

