



HEIDENHAIN



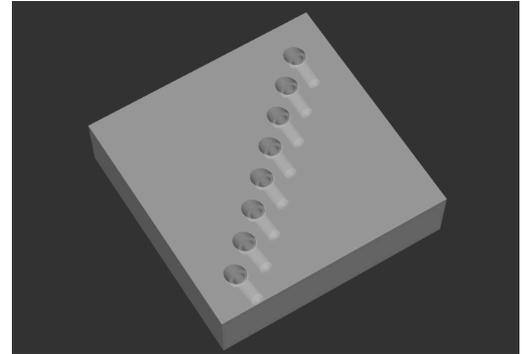
NC Solutions

Description of NC Program 4225

English (en)
3/2020

1 Description of NC program 4225_en.h

NC program for machining a workpiece at a defined number of positions. The machining processes are arranged in a row in the X/Y plane. The machining processes are executed in a tilted working plane.



Requirement

Drill holes in a row. The row is defined in the workpiece coordinate system. The holes must be drilled in a working plane that is tilted relative to the workpiece coordinate system. In order to ensure accuracy, center at the positions before drilling the holes. Center holes prevent the drill from running incorrectly.



The NC program was written for a machine that is adjusted for tilting with spatial angles.

NC program 4225_en.h

At that beginning of the program, define the parameters for the machining operations. In order to execute the defined number of machining processes, the control subsequently calculates the number of required program section repeats.

The next step in the NC program is the machining of the center holes. For this purpose, the control first calls the tool. Then it calls the **LBL 10** subprogram. In this subprogram, the control resets all coordinate transformations. After the control jumps back from the subprogram, Cycle 252 CIRCULAR POCKET is defined. The control machines the center holes with this cycle. Some cycle parameters are defined with the values from the parameter input. If you need to adapt further parameters, you can define the values directly in the cycle.

After the cycle definition, the control calls the **LBL 3** subprogram in which the control defines the machining position, positions the tool, and executes the machining operations. When the subprogram is executed completely and the jump back to the main program was successful, the control retracts the tool.

Subsequently, the control calls the drilling tool. Then Cycle 200 DRILLING is defined. Some of these values are also transferred from the parameter input. You can make changes directly in the cycle. In order to execute machining at the positions, the control calls the **LBL 3** subprogram also after this cycle definition. When the control has executed all machining processes, it retracts the tool and ends the NC program.

The subprograms are defined after the end of the program.

The first subprogram is **LBL 3**. In this subprogram, the control first shifts the datum to the coordinates of the first machining operation. Then it tilts the working plane with Cycle 19 WORKING PLANE. Then it retracts the tool in the tool axis. Depending on the machine kinematics, it might necessary for you to adjust the safe tilting position!

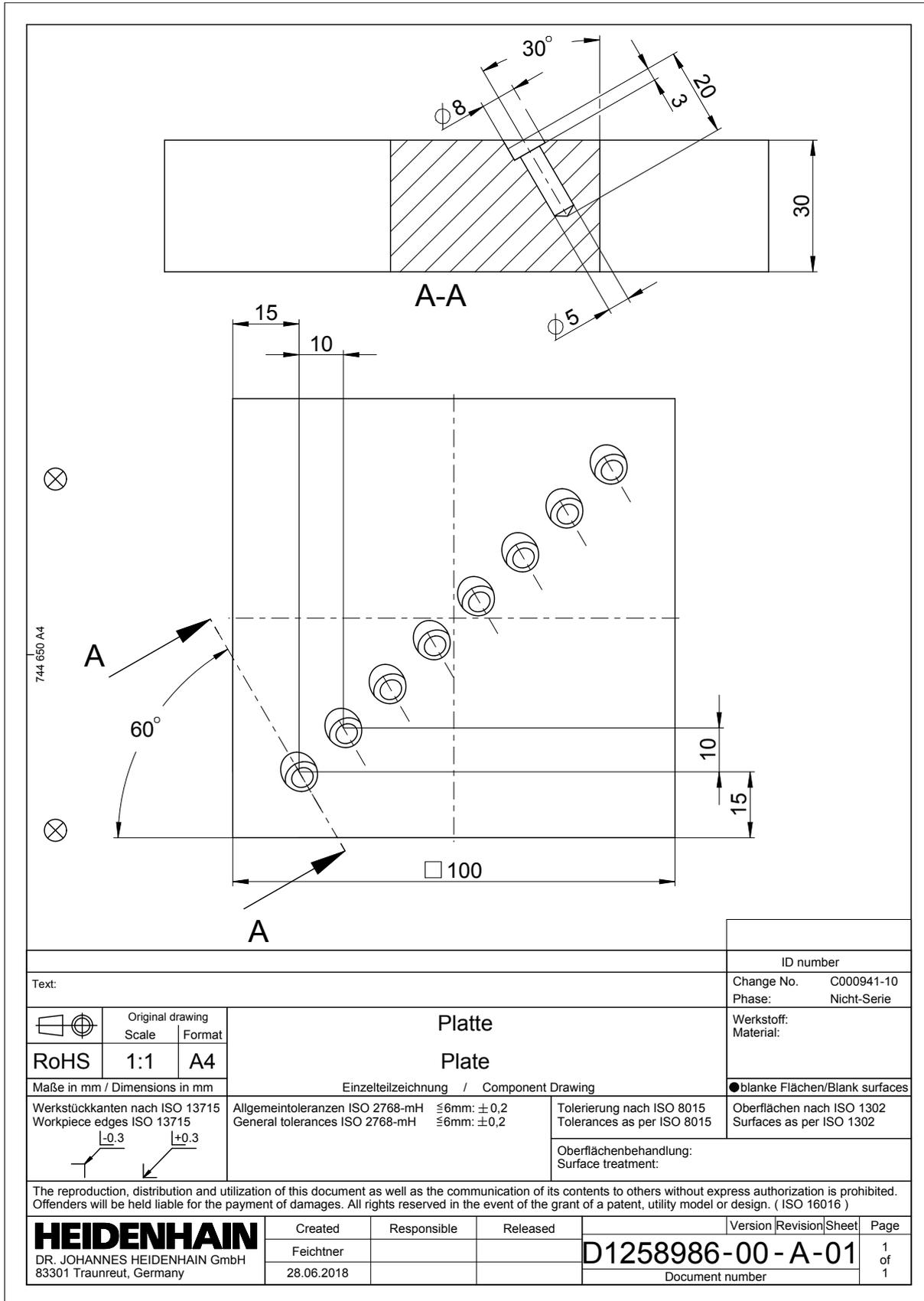
Subsequently, the control positions the rotary axes to the axis angles calculated in Cycle 19. Then the control positions the tool to the first machining position and calls the machining cycle.

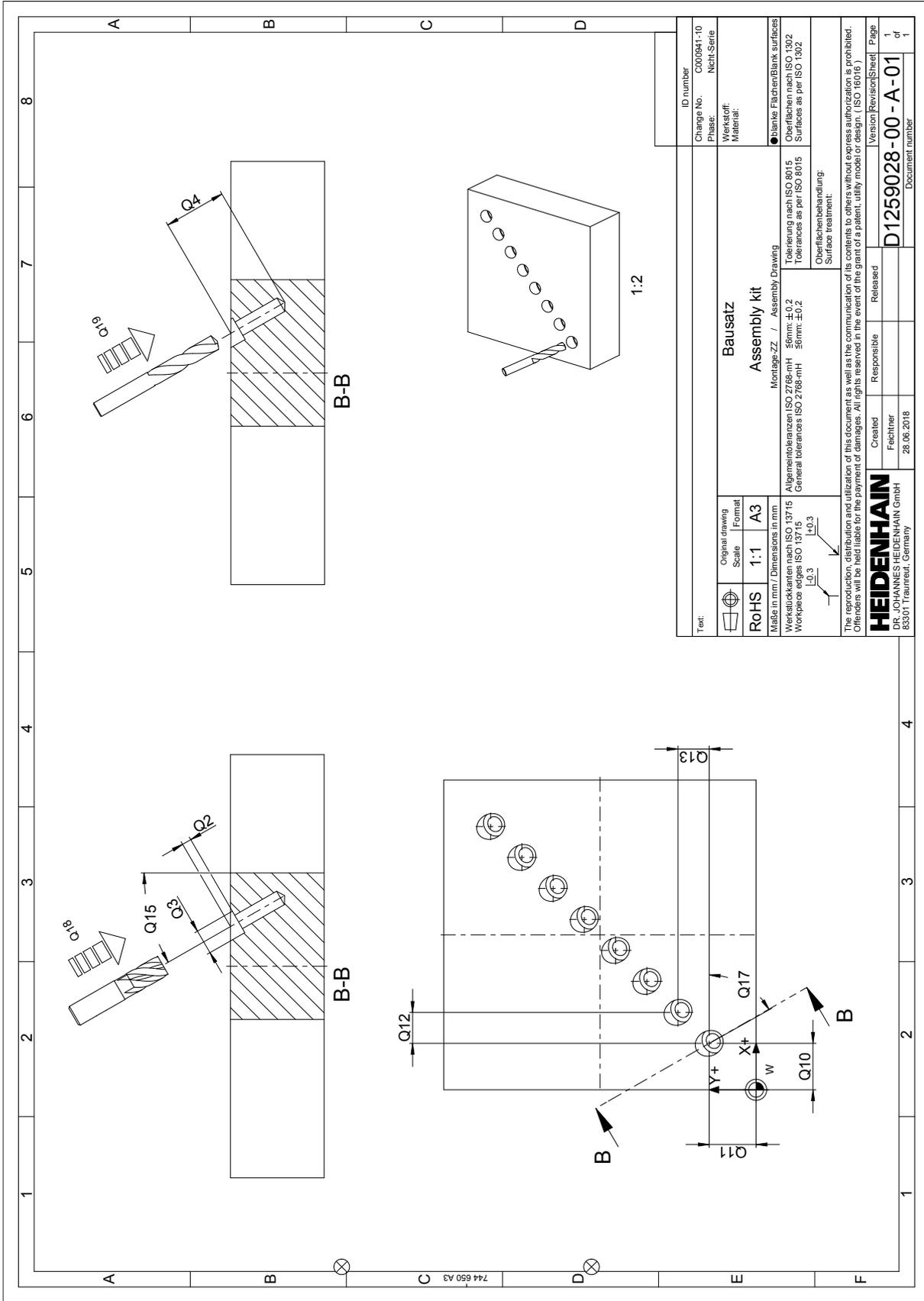
After the first machining operation, a program section repeat is defined for the subsequent machining operations. This program section repeat begins with the **LBL 2** jump label. Then the control calls the **LBL 1** subprogram. The control repeats this call until the calculated number of repetitions has been reached. Then it retracts the tool in the tool axis. Here it might also be necessary to define a different safe tilting position due to the machine kinematics. Subsequently, the control moves the rotary axes to the starting position. The last step in the LBL 3 subprogram is resetting the datum shift. Then the control ends the subprogram and jumps back to the main program.

The second defined subprogram is **LBL 1**. In this subprogram, the control first calls the **LBL 10** subprogram. Then the control incrementally shifts the datum in the X and Y axes by the defined machining clearance. Then the control tilts the working plane to the defined values with Cycle 19. Subsequently, the control positions the tool to the machining position and calls the machining cycle. After the machining process, the control calls the LBL 10 subprogram again. Then the control ends the subprogram and jumps back.

The third subprogram is **LBL 10**. This subprogram serves to reset the coordinate transformation between the machining processes. First, the control resets the tilting to the angles A0, B0, and C0 with Cycle 19 WORKING PLANE. In order to deactivate the tilting the working plane function, the cycle must be defined a second time without values. Then the control resets Cycles 11 SCALING FACTOR and 10 ROTATION. Then it ends the subprogram.

Parameter	Name	Meaning
Q10	COORDINATE OF FIRST HOLE IN X	X coordinate of the first hole, absolute value in the workpiece coordinate system
Q11	COORDINATE OF FIRST HOLE IN Y	Y coordinate of the first hole, absolute value in the workpiece coordinate system
Q12	DISTANCE BETWEEN HOLES IN THE X AXIS	Incremental distance of the holes in the X axis with regard to the workpiece coordinate system
Q13	DISTANCE BETWEEN HOLES IN THE Y AXIS	Incremental distance of the holes in the Y axis with regard to the workpiece coordinate system
Q14	NUMBER OF HOLES	Number of holes to be drilled
Q15	SPATIAL ANGLE A OF MACHINING	Spatial angle A by which the control tilts the working plane for the machining process
Q16	SPATIAL ANGLE B OF MACHINING	Spatial angle B by which the control tilts the working plane for the machining process
Q17	SPATIAL ANGLE C OF MACHINING	Spatial angle C by which the control tilts the working plane for the machining process
Q2	POCKET DEPTH	Pocket depth for the center hole with regard to the tilted working plane
Q3	POCKET DIAMETER	Pocket diameter for the center hole
Q18	FEED RATE FOR POCKET MILLING	Traversing speed of the tool when milling the pocket for the center hole
Q4	HOLE DEPTH	Hole depth with regard to the tilted working plane
Q19	FEED RATE FOR DRILLING	Traverse speed of the tool during drilling





ID number		C000941-10	
Change No.		Nicht-Serie	
Phase:			
Werkstoff:			
Material:			
●Blanke Flächen/Blank surfaces			
Oberflächen nach ISO 1302			
Surfaces as per ISO 1302			
Tolerierung nach ISO 8015			
Tolerances as per ISO 8015			
Oberflächenbehandlung:			
Surface treatment:			
<p>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)</p>			
<p>Original drawing Scale 1:1 Format A3</p>		<p>Bausatz Assembly kit</p>	
<p>Maße in mm / Dimensions in mm Werkstücktoleranzen ISO 2768-mH General tolerances ISO 2768-mH</p>		<p>Montage-ZZ / Assembly Drawing Tolerierung nach ISO 8015 Tolerances as per ISO 8015</p>	
<p>RoHS</p>		<p>Created Früher 28.06.2018</p>	
<p>HEIDENHAIN DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany</p>		<p>Released D1259028-00-A-01</p>	
<p>Version</p>		<p>Revision</p>	
<p>1</p>		<p>1</p>	
<p>of</p>		<p>of</p>	
<p>1</p>		<p>1</p>	
<p>Document number</p>		<p>Document number</p>	