



HEIDENHAIN



NC Solutions

Description of NC Program 4235

English (en)
11/2017

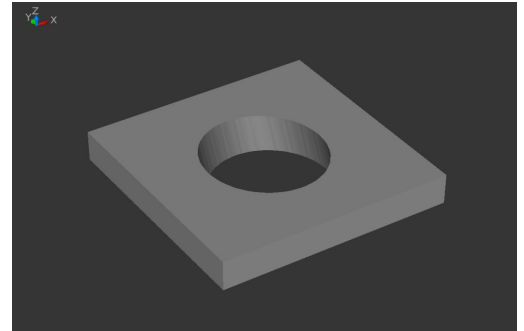
1 Description of NC program 4235_en.h

NC program for machining an inclined hole.



The NC program can be run on the following controls if software option 2 (option 9) is activated:

- TNC 640
- TNC 620 as of NC software number 340 56x-03
- iTNC 530 as of NC software number 340 422-xx



Requirement:

An inclined hole is to be milled. Use a short tool so as to avoid oscillations. Also, the traverse path at the lower edge should be kept to a minimum, in order to avoid a collision with the chucking equipment.

Solution:

In this NC program the control first calculates a core hole, which it then machines perpendicular to the workpiece surface. Then it inclines the tool and moves it on an elliptical path in order to machine the hole contour. The control positions the tool in the Z axis so that the cutting edge moves along the lower edge of the hole.

Description of NC program 4235_en.h

At the beginning of NC program 4235_en.h you define the tool and all of the parameters required for machining. Subsequently, the control performs several calculations. Depending on the result of the calculations, it jumps to two subprograms in order to reverse the algebraic sign of the values.

After these calculations the control first positions the tool to a clearance height and then moves it to the machining center. Cycle 252 CIRCULAR POCKET is subsequently defined. The control uses this cycle to create the perpendicular core diameter for the inclined machining operation. Some parameters of the cycle were already defined with the previously calculated parameters, and you define the rest directly in the cycle.

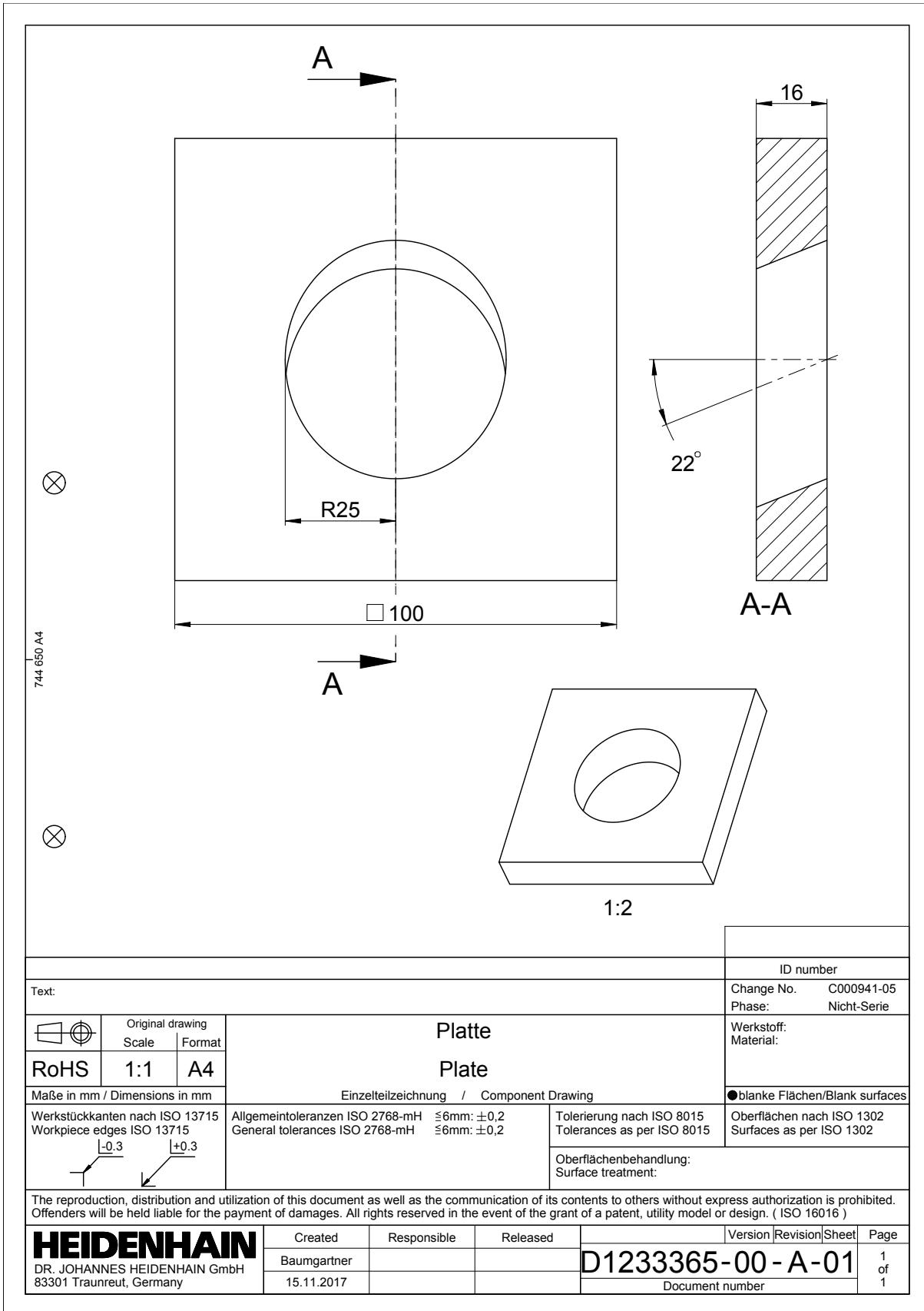
Once the cycle has been executed, the control shifts the datum to the center as well as to the surface of the workpiece. Then it shifts the datum in the Y axis incrementally by the offset between the upper contour and the lower contour.

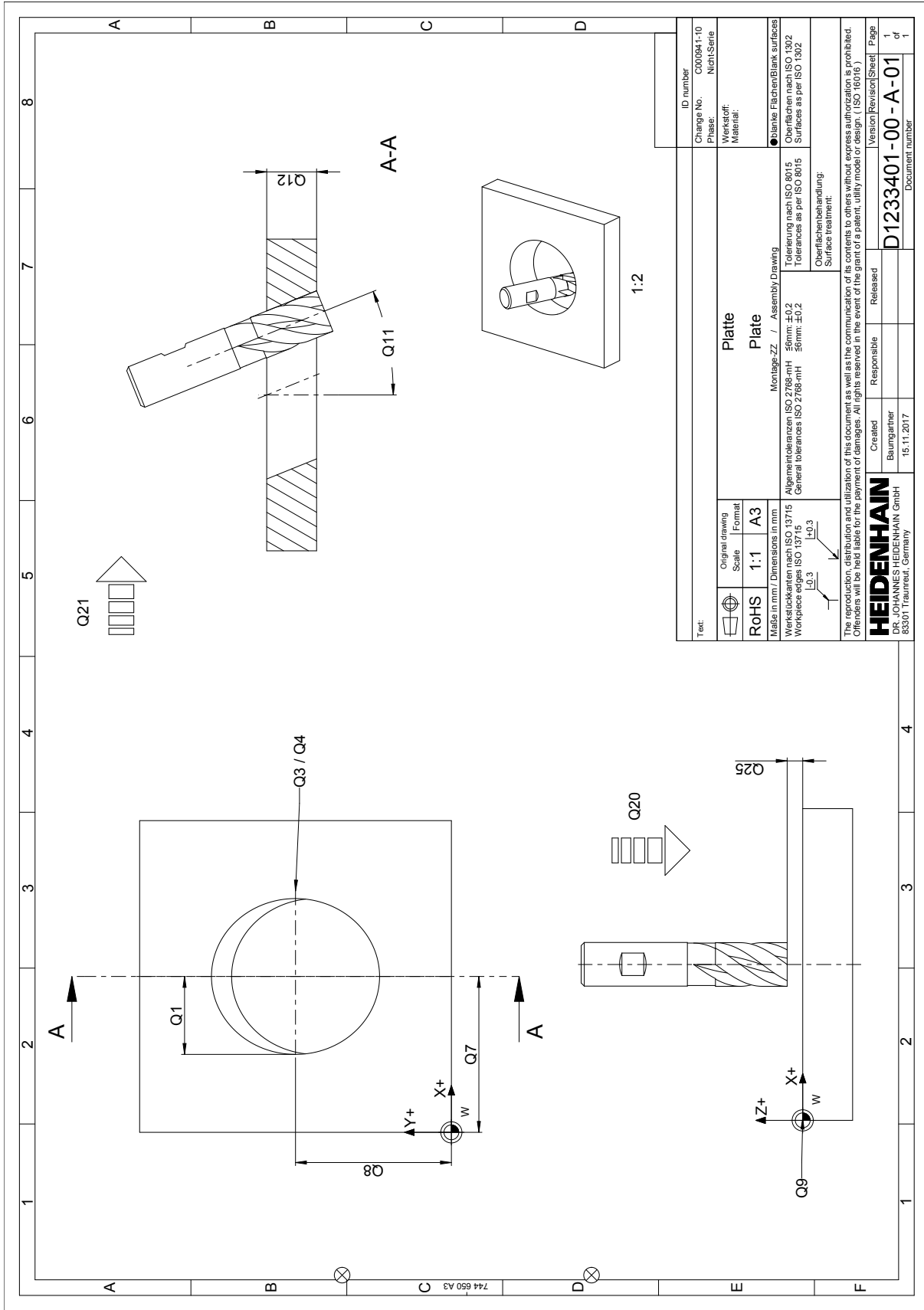
The control then sets the pole to X0 and Y0, and positions the tool there. After that it positions the tool in the Z axis to the safety clearance. Next the control activates the TCPM function, in order to lead the tool center point along the programmed path when positioning the rotary axes. In the next program step, the control tilts the tool to the inclination angle that you defined.

Then there is a jump to a subprogram. In this subprogram the control calculates the tool path for the elliptical contour and traverses this path. The control then calculates the X, Y, and Z coordinates for the starting point of the contour, and approaches this point. The control puts together the ellipse from individual linear paths. You define the number of linear elements of which the ellipse consists in a parameter. The control calculates the end points in the X, Y, and Z axes for each linear element. The calculation and the approach of the points are defined in a program section repeat that the control repeats until the number of defined linear elements is reached.

After this the control first positions the tool in the center of the hole, and then in the Z axis at the defined safety clearance. It then returns the rotary axes to their home positions and deactivates the TCPM function. As the final step in the subprogram, the control resets the datum shift. After jumping back to the main program, the control retracts the tool and ends the NC program.

Parameter	Name	Meaning
Q1	HOLE RADIUS	Radius of the hole to be created
Q3	STARTING ANGLE	Polar angle at which the inclined milling path begins
Q4	STOPPING ANGLE	Polar angle at which the inclined milling path ends
Q7	HOLE CENTER IN X	Center of the hole in the X axis
Q8	HOLE CENTER IN Y	Center of the hole in the Y axis
Q9	COORDINATE SURFACE	Coordinate of the workpiece plane
Q11	INCLINED ANGLE A	Inclination angle of the hole in the spatial angle SPA
Q12	MILLING DEPTH	Milling depth. Remember that the control moves the tool center to this depth. Depending on the tool radius and the inclination angle, the cutting edge might go deeper.
Q20	FEED RATE PLUNGING	Traversing speed of the tool in the Z axis
Q21	FEED RATE MILLING	Traversing speed of the tool in the X/Y plane
Q25	SAFE POSITION Z	Safe position in the Z axis
Q26	PITCH	The number of linear elements into which the control divides the elliptical path of the hole contour





Text:		ID number	
Change No.	C000941-10	Change No.	C000941-10
Phase:	Nicht-Serie	Phase:	Nicht-Serie
Werkstoff:		Werkstoff:	
Material:		Material:	
Platte		Platte	
Original drawing	Format	Montage-ZZ / Assembly Drawing	
Scale	A3		
RoHS	1:1	Tolerierung nach ISO 8015	
Maße in mm / Dimensions in mm		Tolerances as per ISO 8015	
Werkstoffkennlinie nach ISO 13715		General tolerances ISO 2768-mS	
Werkstückkanten nach ISO 13715		Surface treatment:	
Workpiece edges ISO 13715		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		Version	Revision/Sheet
DR. JOHANNES HEIDENHAIN GmbH		Created	15.11.2017
83301 Traunreut, Germany		Baugartner	
		Released	
		Responsible	
		Released	
		Document number	D1233401-00-A-01
		Page	1
		of	1