



HEIDENHAIN



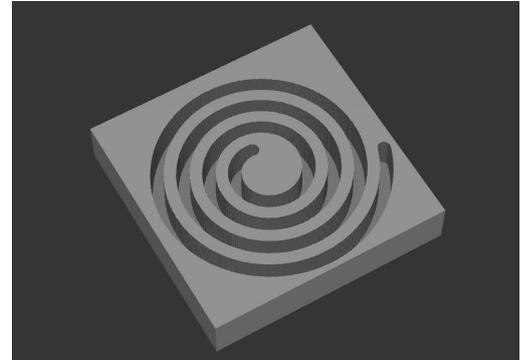
NC Solutions

Description of NC program 2035

English (en)
8/2017

1 Description of the NC program 2035_en.h

NC program for milling a spiral in the X/Y plane



Description

With this NC program, the control mills a spiral in the X/Y plane. The tool move on a path from the inside to the outside.

With parameters, you define whether the control:

- Calculates a tool path with tool radius compensation
- Calculates a lateral allowance
- Calculates the tool path in a clockwise or counterclockwise direction

In the first part of the NC program, you define the tool and all the parameters required for the calculation.

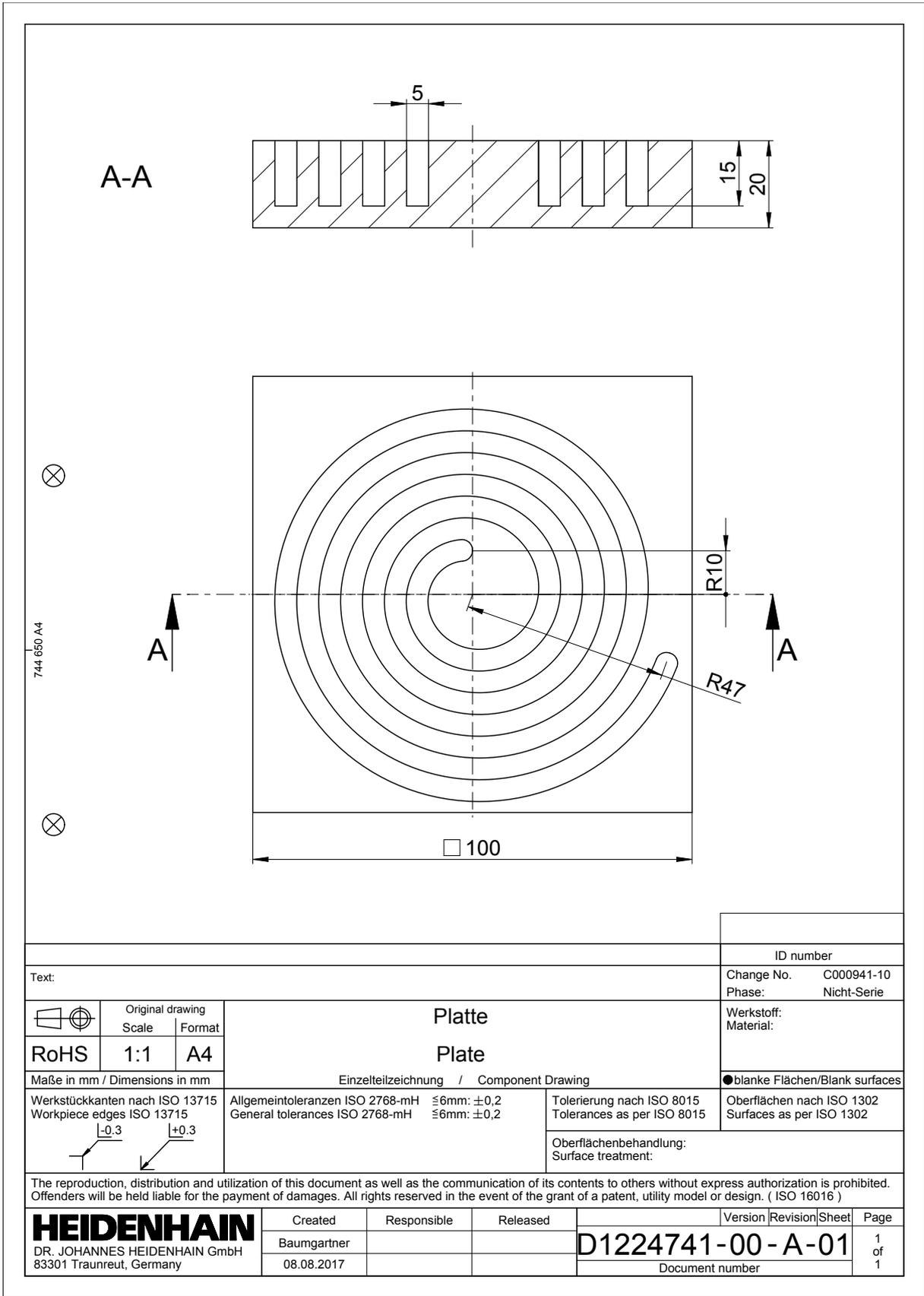
After you enter the parameters, the control pre-positions the tool in the center of the spiral and in the Z axis at the defined safety clearance. The control then calls a subprogram. In this subprogram, the control calculates the tool path and traverses this path. The control puts together the milling path from individual points. For each of these points, the control calculates the X and Y coordinates and approaches each point along a linear path. The pitch parameter allows you to define how many points the control calculates along a 360° path, thereby allowing you to influence the accuracy.

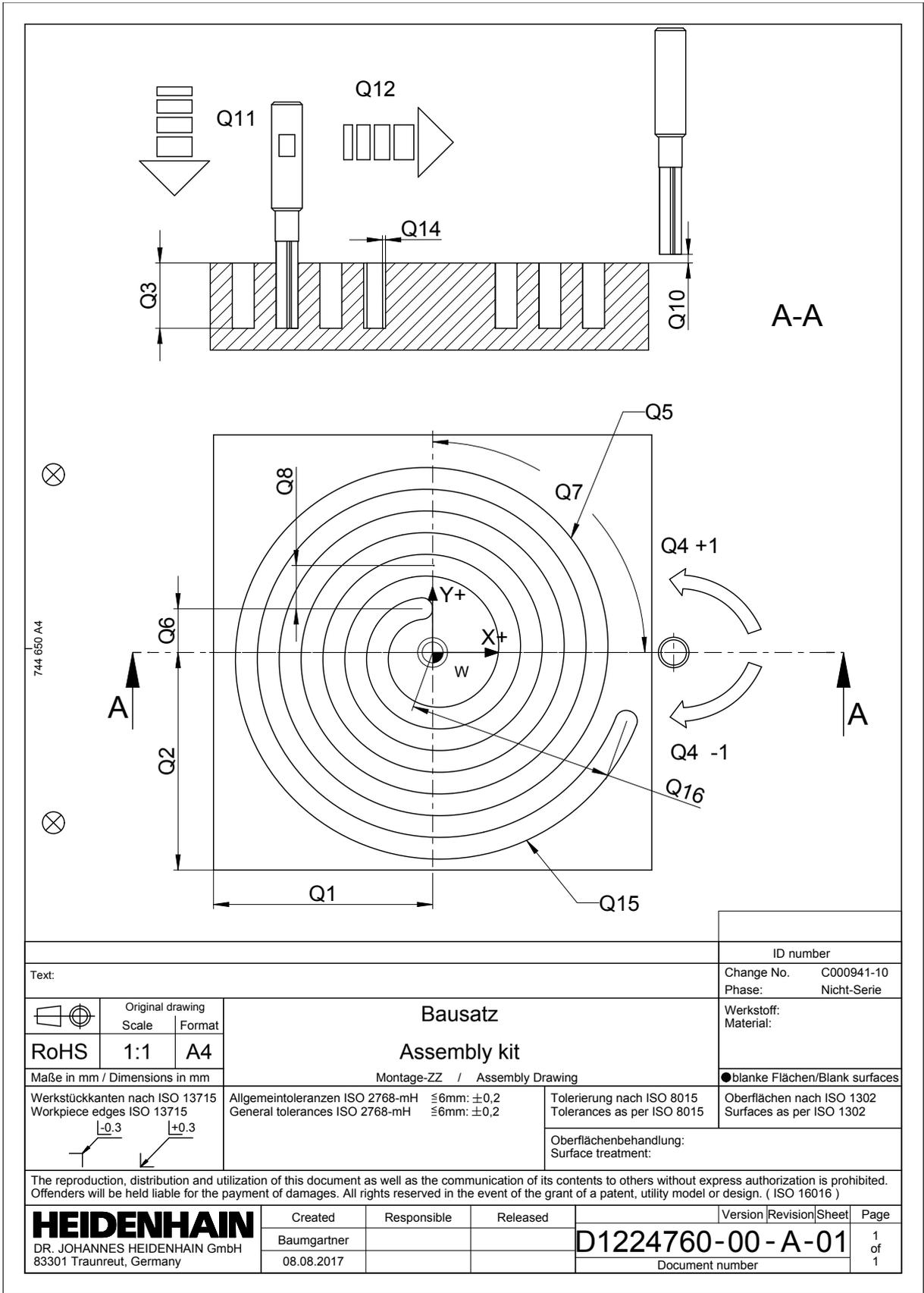
When the tool reaches a point, the control checks the following two possibilities:

- The end radius has not yet been reached: the control repeats the loop with point calculation and path contour
- The end radius has been reached: the tool travels along the Z axis to the set-up clearance

After machining, the control retracts the tool and terminates the program.

Parameter	Name	Meaning
Q1	CENTER X AXIS	X coordinate of the center of the spiral
Q2	CENTER Y AXIS	Y coordinate of the center of the spiral
Q3	DEPTH	Depth of the spiral
Q4	ROTATIONAL DIRECTION	Direction of the milling path <ul style="list-style-type: none"> ■ +1 for a counterclockwise milling path ■ -1 for a clockwise milling path
Q5	PITCH	Number of points calculated on each 360° path
Q6	START RADIUS	Distance between the center point and the starting point of the spiral
Q16	END RADIUS	Distance between the center point and the end point of the spiral
Q7	STARTING ANGLE	Polar angle at the starting point of the spiral
Q8	SLOPE IN MM ON 360 DEGREE	Radius change of the spiral per 360° revolution
Q10	SET UP CLEARANCE	Safe Z position, referenced to the workpiece datum, which the control approaches in rapid traverse
Q11	FEED RATE FOR PLUNGING	Traversing speed of the tool in the Z axis
Q12	FEED RATE FOR MILLING	Traversing speed of the tool in the X/Y plane
Q14	ALLOWANCE FOR SIDE	Allowance in the X/Y plane; only takes effect when a radius correction is defined
Q15	RADIUS CORRECTION	Correction of the tool radius <ul style="list-style-type: none"> ■ 0 for no correction ■ +1 for outward correction ■ -1 for inward correction





Text:		ID number
Change No. C000941-10		Phase: Nicht-Serie
Werkstoff: Material:		●blanke Flächen/Blank surfaces
Tolerierung nach ISO 8015 Tolerances as per ISO 8015		Oberflächen nach ISO 1302 Surfaces as per ISO 1302
Oberflächenbehandlung: Surface treatment:		

	Original drawing	Scale	Format
RoHS	1:1	A4	
Maße in mm / Dimensions in mm			
Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715		Allgemeintoleranzen ISO 2768-mH $\leq 6\text{mm}$: $\pm 0,2$ General tolerances ISO 2768-mH $\leq 6\text{mm}$: $\pm 0,2$	
		Tolerierung nach ISO 8015 Tolerances as per ISO 8015	

Bausatz
Assembly kit
Montage-ZZ / Assembly Drawing

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	Released	Version	Revision	Sheet	Page
	Baumgartner			D1224760-00-A-01			1 of 1
	08.08.2017			Document number			1