



HEIDENHAIN



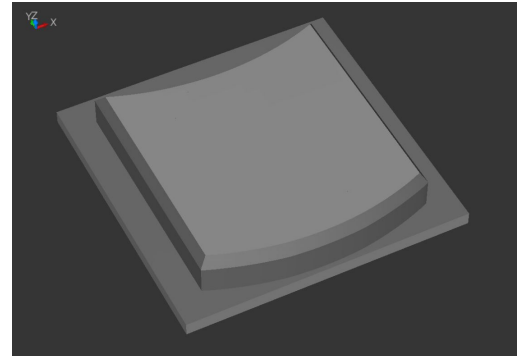
NC solutions

Description of NC program 3255

English (en)
6/2018

1 Description of NC program 3255_en.h

This NC program is for creating a chamfer on the upper edge of a contour.



Description

With this NC program, the control creates a chamfer on the upper edge of a contour. The control performs this machining operation in contour lines. You define the number of contour lines indirectly via the parameter DISTANCE BETW. CONTOUR LINES. This parameter allows you to influence the machining time and the surface quality of the chamfer to be machined.

Machining can be performed with an end mill, a ball-nose cutter, or a toroid cutter. For this purpose, the control reads tool radius 2 of the active tool from the tool table. Depending on the type of milling, the point of intersection is dependent on the chamfer angle. The NC program automatically calculates the contour lines based on the chamfer angle.

Program sequence of NC program 3255_en.h

At the beginning of the program, you define the tool for milling the contour. The control then positions the tool at a clearance height. The SL cycles 14, 270, and 25 are then defined for contour milling. For these cycles, the parameters must be configured in accordance with your application. The contour to be machined is described in the subprogram LBL 10. Then the control calls the cycles with the M99 function.

In the next step, you define the tool for machining the chamfer. Then the control retracts the tool to a clearance height. You then define the parameters needed for the chamfer. Then the control jumps to the CHAMFER subprogram.

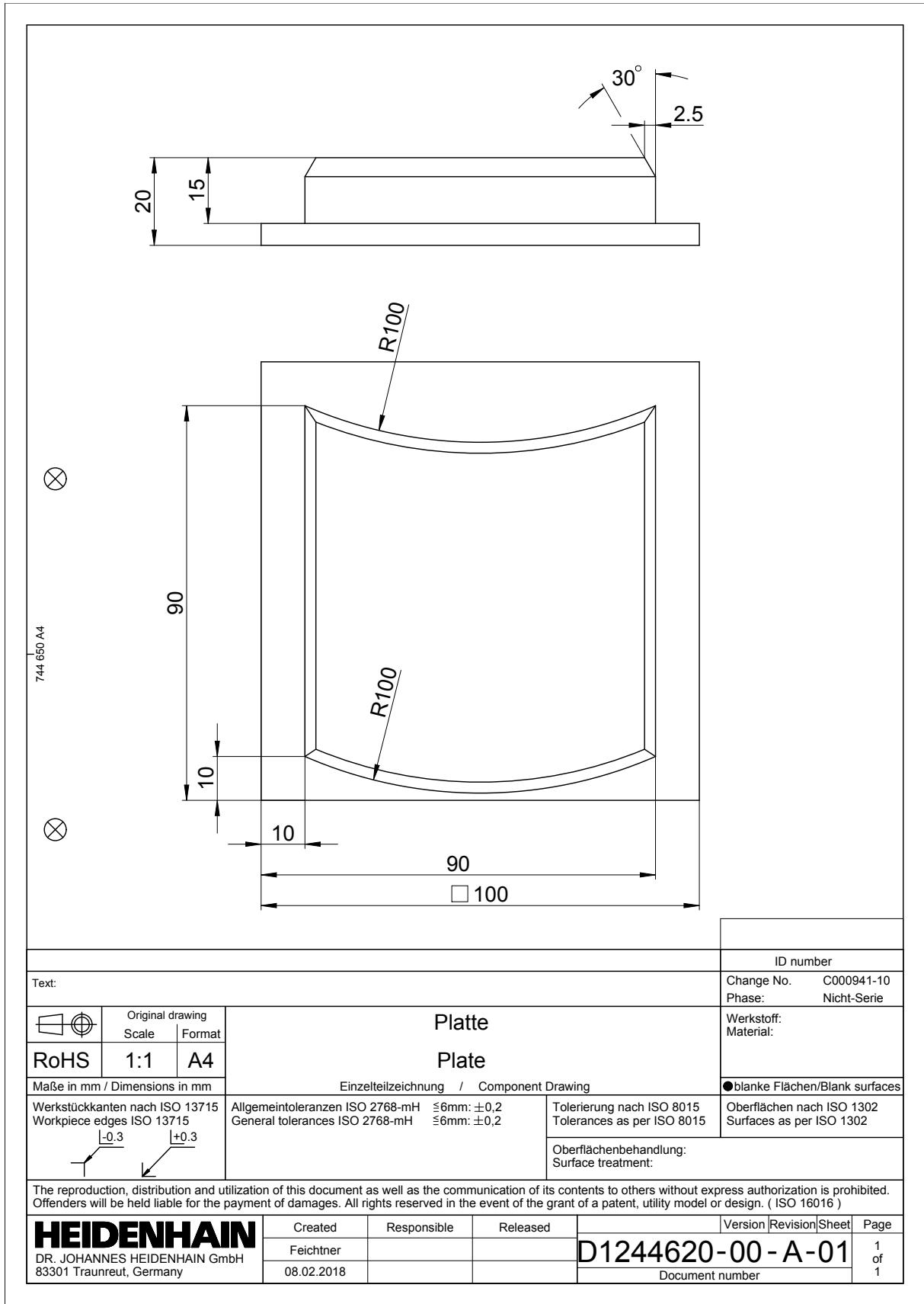
In the CHAMFER subprogram, the control performs all of the calculations and path contours needed for milling the chamfer. Since the parameters you have defined are used here, you do not need to change anything in the subprogram.

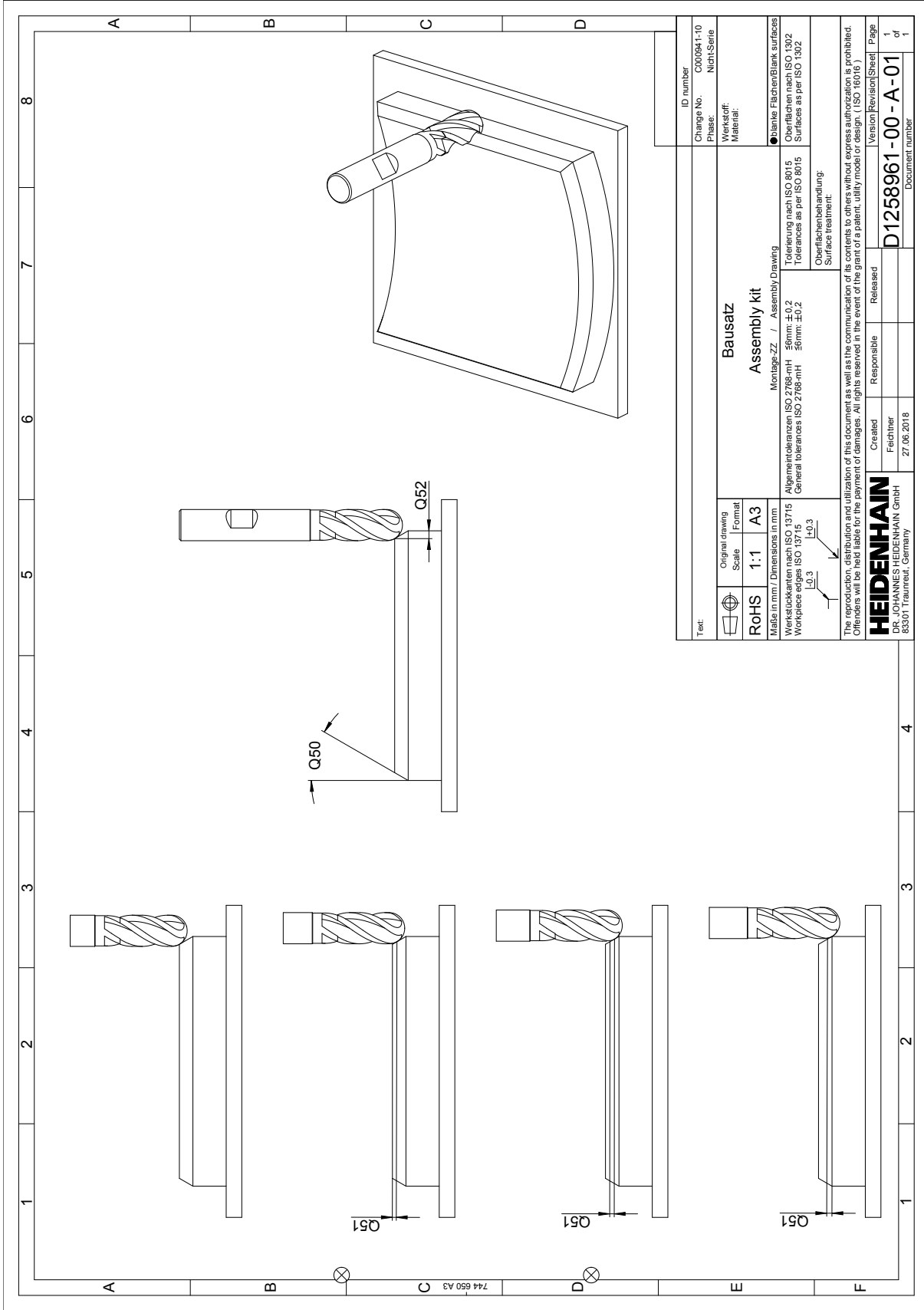
In the subprogram, cycles 14 and 270 are first defined. Then, with an FN18 function, the control reads the value of tool radius 2 of the active tool from the tool table. The control then calculates the Z coordinate for the first contour line and checks whether it is deeper than the total depth. If this Z coordinate is deeper than the total depth, the control jumps to LBL END1, where it machined the last contour line. If the Z coordinate of the first contour line is higher than the total depth, the control calculates the milling depth and the lateral allowance for the first contour line. During this process, the control takes into account the compensation for radius 2 of the tool being used.

After the calculations, the control machines the calculated contour line with Cycle 25. The control then repeats this program section until the calculated Z coordinate is less than the total depth. Then the control jumps to LBL END1.

In the END1 subprogram, the control calculates the last contour line and machines this path. Then the control jumps back to the main program where the control retracts the tool and ends the NC program.

Parameter	Name	Meaning
Q50	CHAMFER ANGLE	Angle of the chamfer relative to perpendicular
Q51	DISTANCE BETW. CONTOUR LINES	Distance between the two contour lines in mm
Q52	HORIZONTAL CHAMFER LENGTH	Length of the chamfer section in the X/Y plane





ID number		Change No. C000941-10	
Phase:		Nicht-Serie	
Werkstoff:		Material:	
●Blanke Flächen/Blank surfaces		Oberflächen nach ISO 1302	
Surfaces as per ISO 1302		Tolerances 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 Feichtner 27.06.2018	Released
Version		Revision/Sheet	
D1258961-00 - A - 01		1	
Document number		1	