



HEIDENHAIN



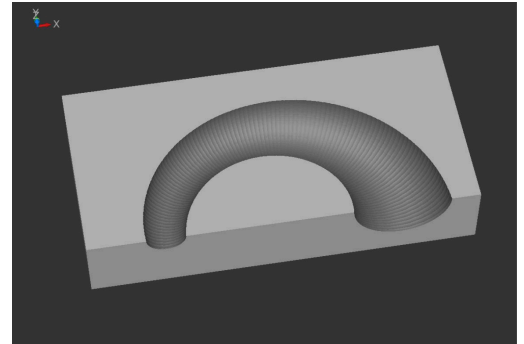
NC Solutions

Description of NC Program 3085

English (en)
9/2018

1 Description of NC programs 3085_en.h and 30851_en.h

NC program for machining a concave arc segment. For the machining operation, you define the angular lengths in the X/Y plane and the X/Z plane using parameters. In addition, you define the contour radius in the X/Z plane at the start of the contour and at the end of the contour. The control divides the contour to be machined into linear segments. You use further parameters to define into how many cuts the machining operation is divided by the control.



NC program 3085_en.h

You describe the workpiece blank in the BLK FORM at the beginning of the program. Then you define the tool. In this case a spherical cutter must be used. Then a further TOOL CALL is programmed. The control uses this tool call to shift the tool center point to the center of the sphere of the tool. If the tool you defined is measured at the center of the sphere, you must delete this NC block. Then the control positions the tool to a safe position in the Z axis.

In the next program section, you define all parameters required for machining. The first block contains the parameters with the values of the body to be machined, followed by the parameters for the cutting values. Subsequently, the control performs two calculations. Then it prepositions the tool in the X/Y plane to the center of the machining operation.

Then the control calls the NC program 30851_en.h. In this NC program, the control performs all of the calculations and positioning movements for the machining operation.

In the example program, the control machines the body in two steps. In the first call, it uses values for roughing and then values for finishing. After the return jump from NC program 30851_en.h into the main program, you define the cutting parameters for finishing. Then the control prepositions the tool again to the center. Then it jumps again into the NC program 30851_en.h, in which the control performs the machining operation.

After jumping back to the NC program 3085_en.h again, the control retracts the tool and ends the NC program.

Parameter	Name	Meaning
Q1	ARC CENTER X	X coordinate of circle center in the X/Y plane
Q2	ARC CENTER Y	Y coordinate of circle center in the X/Y plane
Q3	ARC CENTER Z	Z coordinate of circle center in the X/Z plane
Q6	STARTING RADIUS OF CONTOUR IN X/Z PLANE	Radius at the starting point of the contour in the X/Z plane
Q16	END RADIUS OF CONTOUR IN X/Z PLANE	Radius at the end point of the contour in the X/Z plane
Q7	STARTING ANGLE OF CONTOUR IN X/Z PLANE	Polar angle at the starting point of the contour in the X/Z plane (reference axis Z+)
Q17	END ANGLE OF CONTOUR IN X/Z PLANE	Polar angle at the end point of the contour in the X/Z plane (reference axis Z+)
Q8	STARTING ANGLE OF CONTOUR IN X/Y PLANE	Polar angle at the starting point of the contour in the X/Y plane (reference axis X+)
Q18	END ANGLE OF CONTOUR IN X/Y PLANE	Polar angle at the end point of the contour in the X/Y plane (reference axis X+)
Q10	RADIUS AT CENTER POINT OF CONTOUR IN X/Y PLANE	Radius of the center of the contour in the X/Y plane
Q25	SET-UP CLEARANCE	Distance in the tool axis that is maintained during prepositioning
Q26	NUMBER OF CUTS	Number of cutting paths in the X/Y plane
Q27	NUMBER OF LINEAR SEGMENTS PER CUT	Value indicating how many linear segments a cutting path is divided into by the control
Q20	FEED RATE FOR PLUNGING	Traversing speed in the Z axis
Q21	FEED RATE FOR MILLING	Traversing speed within the cut
Q22	FEED RATE FOR PREPOSITIONING	Traversing speed during prepositioning
Q23	OVERSIZE IN Z AXIS	Allowance by which the control shifts the machining operation in the tool axis
Q29	ROUGHING FACTOR	Value by which the infeed movements are multiplied. This reduces the number of cuts

NC program 30851_en.h

NC program for calculating and performing the individual positioning movements for machining a concave arc segment.

You define the parameters required for the calculation in the NC program 3085_en.h. You do not need to make any changes to this NC program.

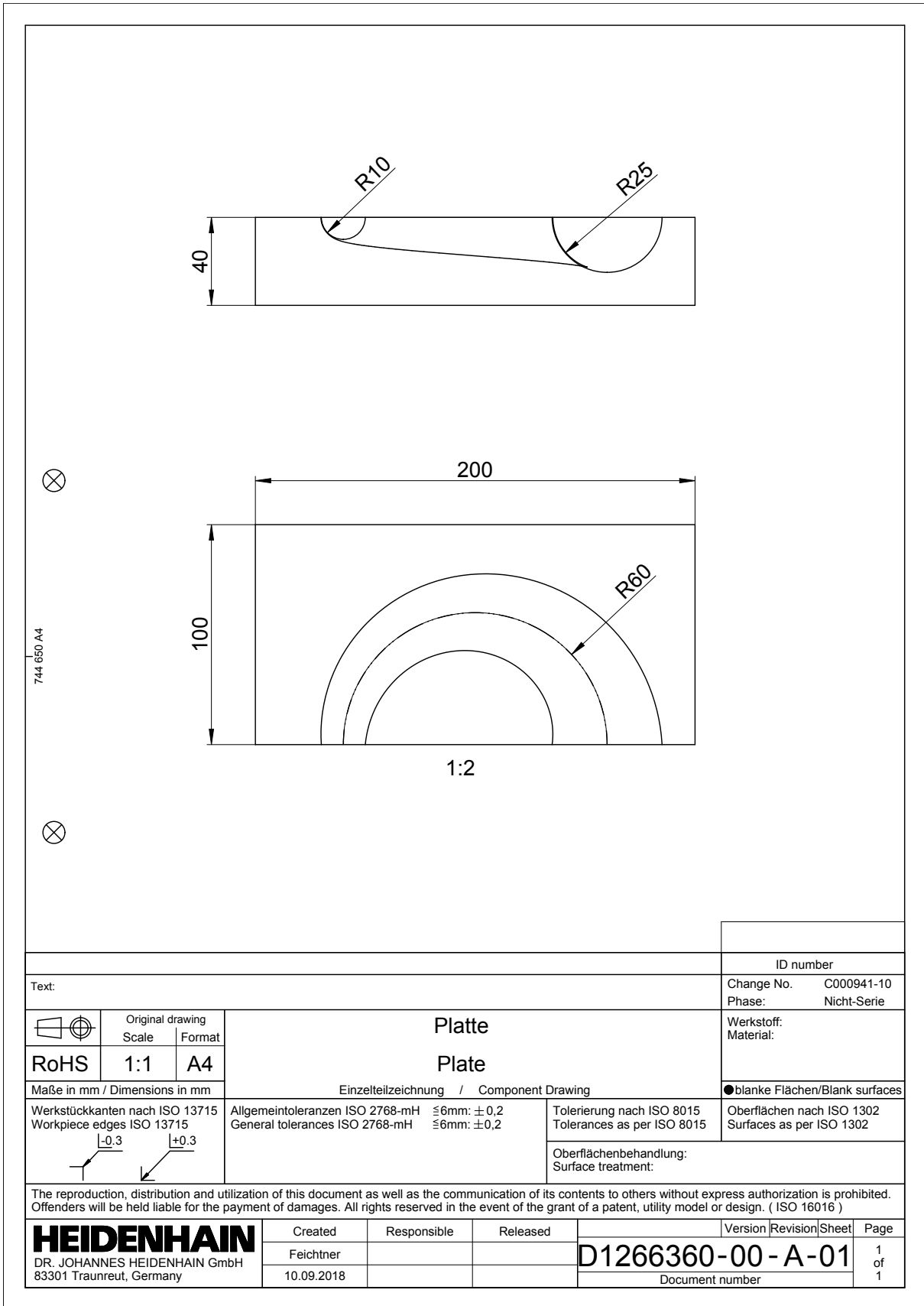
In the first program section, the control performs several calculations for defining the current values and the step values. Then it shifts the datum to the center of the contour to be machined.

In the next step, the control rotates the coordinate system to the starting angle of the contour using Cycle 10. The control then prepositions the tool. After this, it defines the center of the cutting path in the X/Z plane and prepositions the tool to the starting point. A program section repeat follows in which the control calculates and approaches the new angle in the X/Z plane. The control repeats this program section until the end angle has been reached in the X/Z plane.

Subsequently, the control updates the counter of the number of cuts, the radius to be machined and the angle of the cutting path. Then the control checks whether the number of cuts has been reached. If the number has been reached, it jumps to the end of the program. If the number has not been reached, it rotates the coordinate system to the updated angle. Then it approaches the new starting point. After this, the program section in which the control calculates and approaches the next cutting path is repeated again.

After every repetition, the control checks whether it will end the NC program or jump to the repetition again.

The control repeats the program sequence until the number of cuts has been reached. It then retracts the tool and ends the NC program.



Text:		ID number	
Change No. C000941-10		Phase: Nicht-Serie	
Werkstoff: Material:		Werkstoff: Material:	
Maße in mm / Dimensions in mm		Einzelteilzeichnung / Component Drawing	
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		Oberflächen nach ISO 1302 Surfaces as per ISO 1302	
Oberflächenbehandlung: Surface treatment:		●blanke Flächen/Blank surfaces	
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 10.09.2018	Responsible Released Version Revision Sheet Page D1266360-00 - A-01 1 of 1
		Document number	

