



HEIDENHAIN



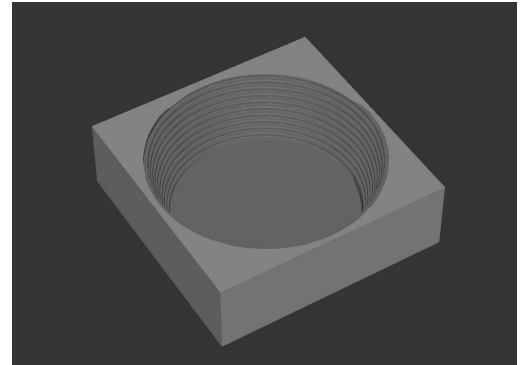
NC Solutions

Description of NC program 3170

English (en)
5/2017

1 Description of NC programs 3170_de.h and 31701_de.h

NC program for creating a tapered internal thread



3170_de.h NC program

At the program beginning the control first calls the NC program 31701_de.h. With this NC program the control machines the taper in which it then cuts the thread.

In the next step, you define in the NC program the tool and all parameters required for machining.

The machining goes from bottom to top.



With the parameter MACHINING DIRECTION you define whether the control is machining a left-handed thread or right-handed thread.

After entering these parameters the control calls a subprogram. In this subprogram the control calculates the tool path and traverses this path. The milling path consists of individual points. The control calculates the X, Y and Z coordinate for each of these points and approaches the point along a linear path. The pitch parameter enables you to define how many points the control calculates along a 360° path; this therefore influences the accuracy.

In the example program, some parameters are defined again after the first operation. The control calls the subprogram a second time and finished the thread groove.

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

Parameter	Name	Meaning
Q1	CENTER IN X	X coordinate of the circle center point
Q2	CENTER IN Y	Y coordinate of the circle center point
Q4	MACHINING DIRECTION	Direction of the milling path <ul style="list-style-type: none"> ■ +1 for counterclockwise milling path ■ -1 for clockwise milling path
Q5	PITCH	Number of calculated points in each 360° path
Q6	OUTSIDE RADIUS BELOW	Major radius of thread at start of milling path
Q7	STARTING ANGLE	Polar angle at the starting point of the milling path
Q8	TAPER ANGLE	Angle of the taper shaft
Q9	PITCH	Thread pitch
Q10	CLEARANCE HEIGHT	Safe Z position relative to the workpiece zero point
Q11	FEED RATE FOR PRE-POSITIONING	Traversing speed of tool during pre-positioning
Q12	FEED RATE FOR MILLING	Traversing speed of the tool on the helical path
Q3	Z-COORDINATE AT START	Z coordinate at the starting point of the milling path
Q13	Z-COORDINATE AT END	Z coordinate at the end of the milling path
Q14	OVERSIZE IN THE X/Y PLANE	Oversize in the X/Y plane
Q16	LATERAL SAFETY CLEARANCE	Distance traversed by the control when pre-positioning in the X/Y plane

31701_de.h NC program

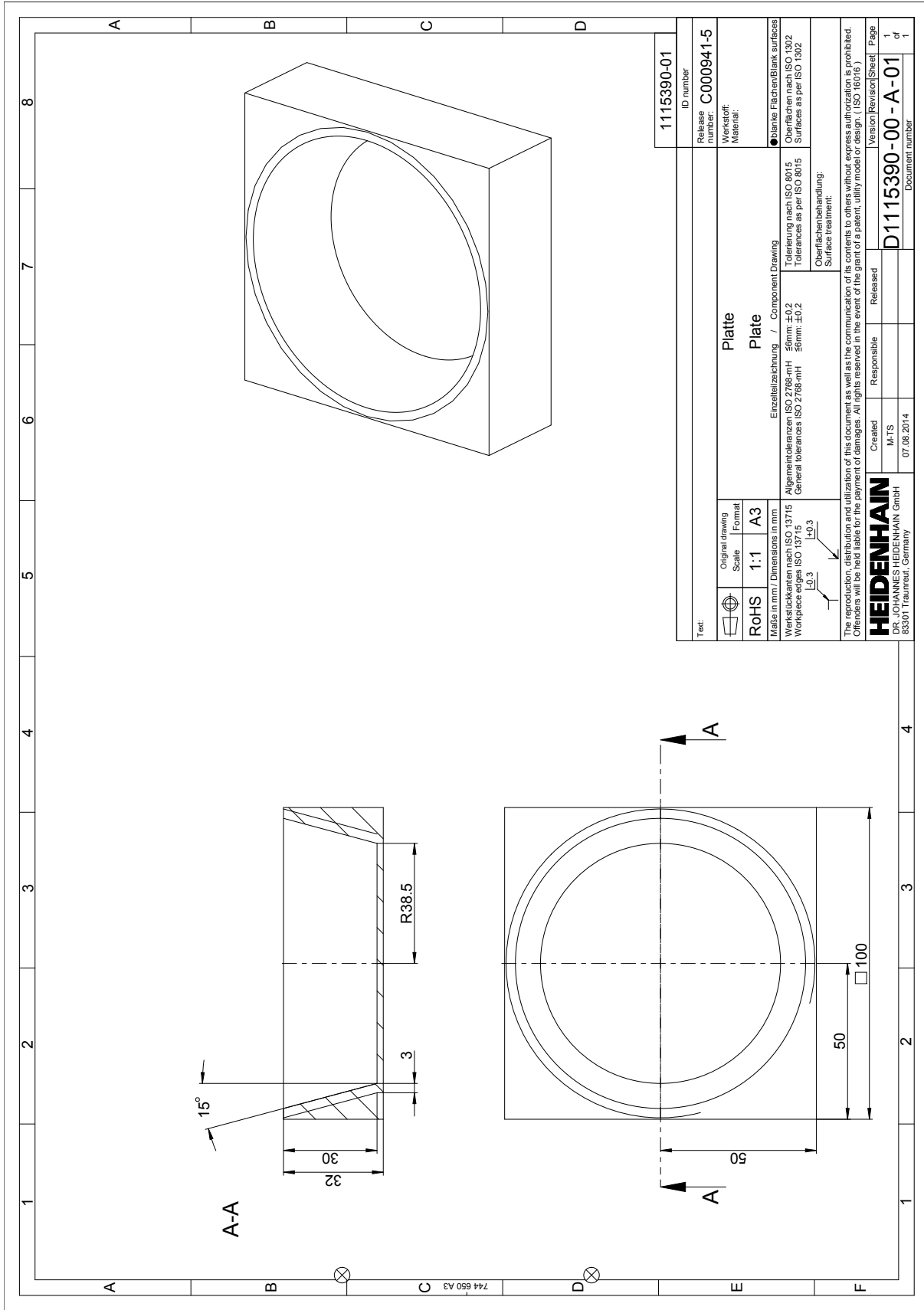
NC program for milling a taper. The control conducts the operation with tool paths in the X/Z plane.

At the program beginning you define the tool and all parameters required for machining.

After parameter input, the control makes some calculations. Then the control moves the tool to a clearance height, shifts the datum to the taper center and pre-positions the tool. In the next step the control positions the tool to the upper radius and moves it on the milling path in the X and Z axes. Then the control moves the tool in the X axis to the taper center and then in the Z axis to a safe position. This is followed by an incremental rotation of the coordinate system. The control repeats the sequence until the taper is completely machined.

After machining, the control resets the rotation and the datum shift, retracts the tool and ends the NC program.

Parameter	Name	Meaning
Q1	CENTER IN 1ST AXIS	X coordinate of the taper center
Q2	CENTER IN 2ND AXIS	Y coordinate of the taper center
Q3	COORDINATE Z ABOVE	Z coordinate of the taper top edge
Q5	UPPER RADIUS	Radius at the taper top edge
Q4	COORDINATE Z BELOW	Z coordinate at the taper bottom edge
Q6	LOWER RADIUS	Radius at the taper bottom edge
Q7	SAFETY CLEARANCE	Safe Z position, with respect to the taper top edge
Q8	FEED RATE FOR MILLING	Tool traversing speed during machining
Q9	FEED RATE FOR RETRACTION	Tool traversing speed during re-positioning
Q10	PITCH	Number of cuts in the X/Z plane



ID number 1115390-01		Release number: C000941-5	
Werkstoff: Material:		●blanke Flächen/Blank surfaces	
Einzelteilzeichnung / Component Drawing		Oberflächen nach ISO 1302 Surfaces as per ISO 1302	
Platte		Tolerierung nach ISO 8015 Tolerances as per ISO 8015	
Platte		General tolerances ISO 2768-mH ±0.2	
Maße in mm / Dimensions in mm		±0.3	
Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715		±0.3	
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)			
Original drawing Scale	Format	Created	Released
1:1	A3	M-TS 07.08.2014	
Responsible		Version/Revision/Sheet	
HEIDENHAIN DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany		D1115390-00 - A - 01 1 1	

