HEIDENHAIN

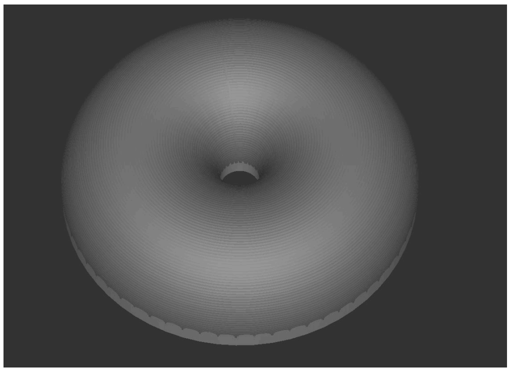
NC Solutions

Description of NC Program 7025

English (en)
3/2020

1 Description of NC program 7025_en.h

NC program for externally machining a horizontal, torus-shaped workpiece along contour lines.



NC program 7025_en.h

At the beginning of the program, define all parameters required for machining. Then define the blank form and the tool. The NC program was created for machining with a ball-nose cutter. In order to shift the tool center point from the south pole of the tool to the center of the sphere, a second **TOOL CALL** block is programmed after the tool call. In this NC block, the control uses the DL function to correct the tool length by the active tool radius to the center of the sphere. If ball-nose cutters in your machine tool are measured relative to the center of the sphere, then you must delete this NC block.

Subsequently, the control performs several calculations. It calculates:

- The compensated circle radius
- The X coordinate of the center of the circle radius
- The stepping angle between two contour lines
- The number of required program section repeats in order to machine all of the contour lines

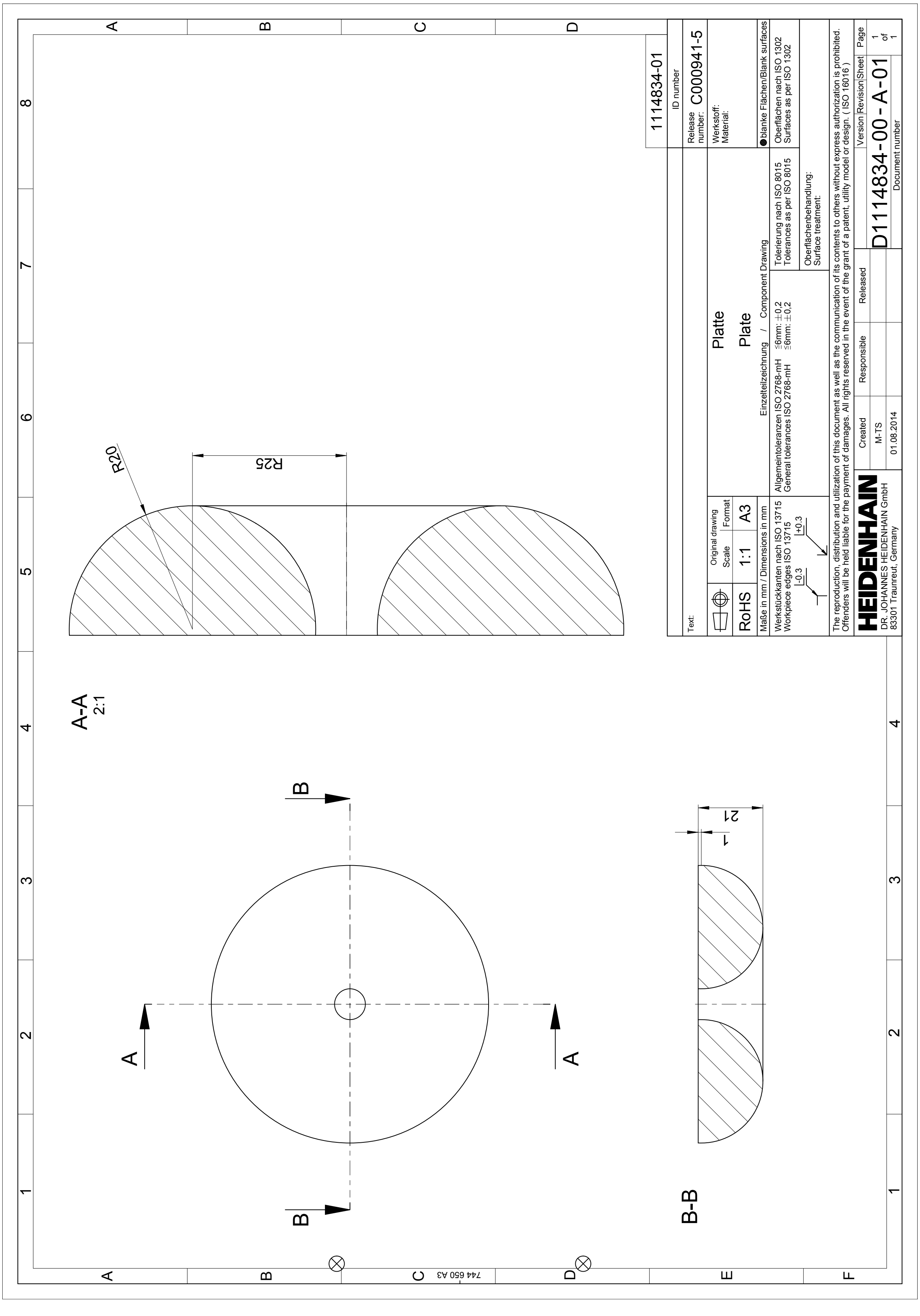
Then the control positions the tool to the center of the machining process. In the next step, it moves the tool to a safe Z position. Subsequently, it defines the circle center point at the center of the circle radius in the X/Z plane.

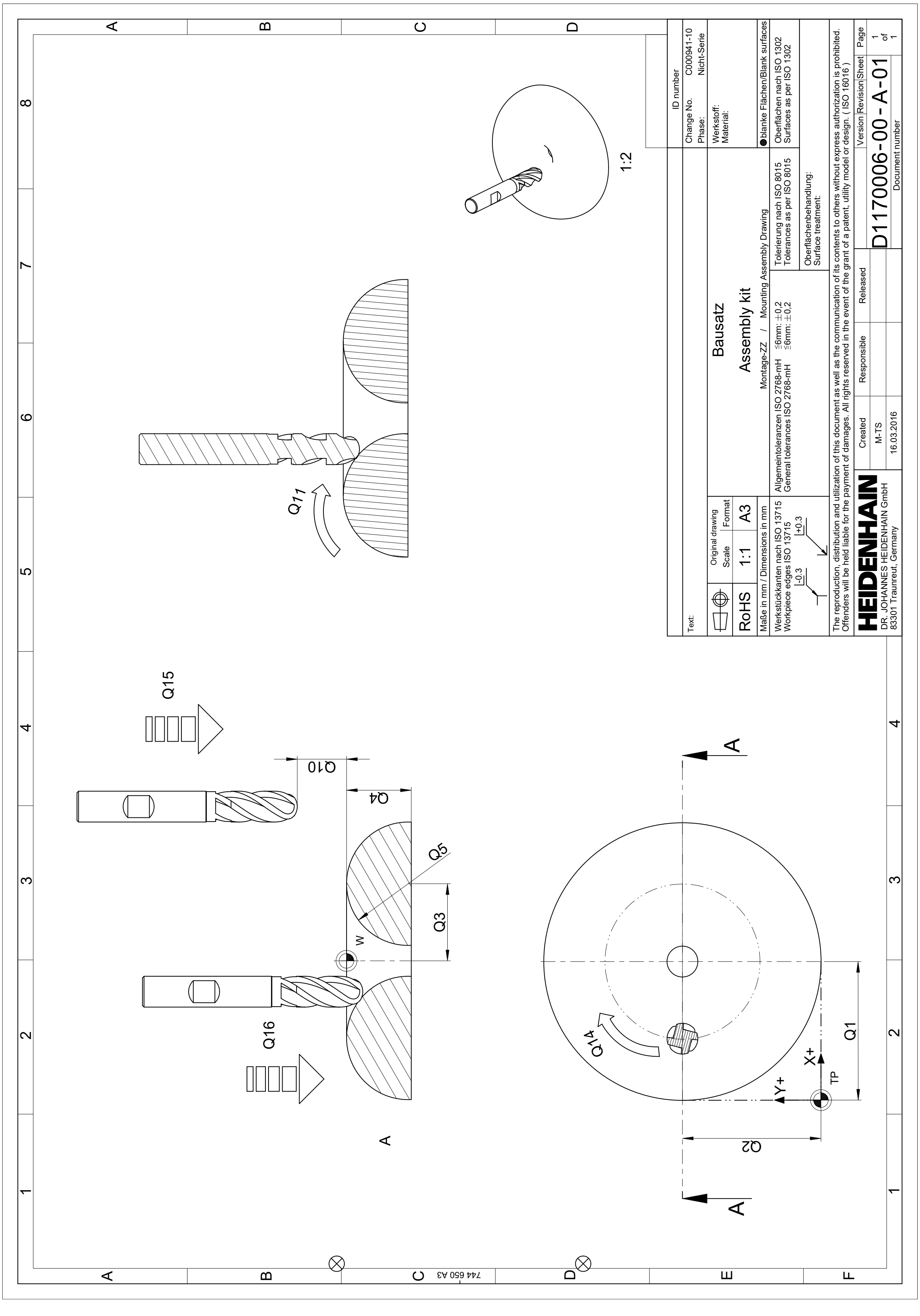
Then the control positions the tool to the starting point of the first contour line. Then it sets the circle center point at the center of the rotational radius in the X/Y plane. Subsequently, the control moves along a 360° circular path around the circle center point.

Then a jump label is set for the program section repeat. The repetition begins with the definition of the circle center point at the center of the circle radius in the X/Z plane. Then the control positions the tool to the starting point of the next contour line on a circular path around the circle center point. For this positioning, the control moves incrementally with the calculated stepping angle between two contour lines.

Then the control sets the circle center point at the center of the rotational radius again. Subsequently, it moves along a 360° circular path for the next contour line. Then the program section repeat is called. After the calculated number of repetitions has been reached, the control retracts the tool. Then it ends the NC program.

Parameter	Name	Meaning
Q1	CENTER IN 1ST AXIS	The X coordinate of the center of the rotational diameter
Q2	CENTER IN 2ND AXIS	The Y coordinate of the center of the rotational diameter
Q3	TORUS RADIUS R	Rotational radius of the torus
Q4	Z-COORDINATE AT TORUS CENTER	The Z coordinate of the center of the circle diameter
Q5	TORUS RADIUS r	Circle radius of the torus
Q10	CLEARANCE HEIGHT	The Z coordinate for safe positioning
Q11	NUMBER OF CONTOUR LINES	Number of milling paths that the control calculates for the machining process
Q14	FEED RATE FOR MILLING	Traversing speed of the tool during machining in the X/Y plane
Q15	FEED RATE FOR PRE-POSITIONING	Traversing speed of the tool during pre-positioning
Q16	PLUNGING FEED RATE	Traversing speed of the tool in the tool axis


A-A
2:1
R20
R25
B-B
B
A
1114834-01
ID number
C000941-5
Platte
Plate
Einzelteilzeichnung / Component Drawing
RoHS
1:1
A3
HEIDENHAIN
DR. JOHANNES HEIDENHAIN GmbH
83301 Traunreut, Germany
M-TS
01.08.2014
D1114834-00-A-01
Document number


Q1
Q2
Q3
Q4
Q5
Q10
Q11
Q14
Q15
Q16
TP
X+
Y+
1:2
Bausatz
Assembly kit
HEIDENHAIN
DR. JOHANNES HEIDENHAIN GmbH
83301 Traunreut, Germany
D1170006-00-A-01
Created
M-TS
16.03.2016
Responsible
Released
Change No.
C000941-10
Phase:
Nicht-Serie
Scale
1:1
Format
A3
RoHS
Maße in mm / Dimensions in mm
Montage-ZZ / Mounting Assembly Drawing