HEIDENHAIN

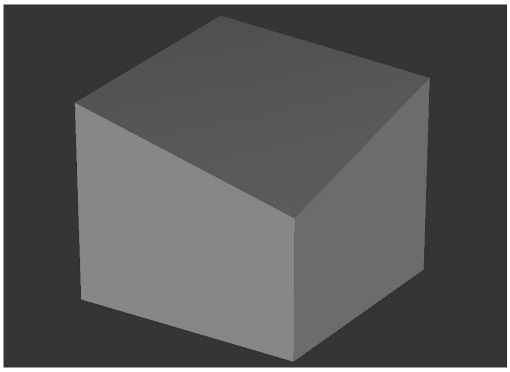
NC Solutions

Description of NC program 4020

English (en)
4/2017

1 Description of NC programs 4020_en.h and 40201_en.h

NC program for machining a 3-D ruled surface. The control moves the tool in five axes.

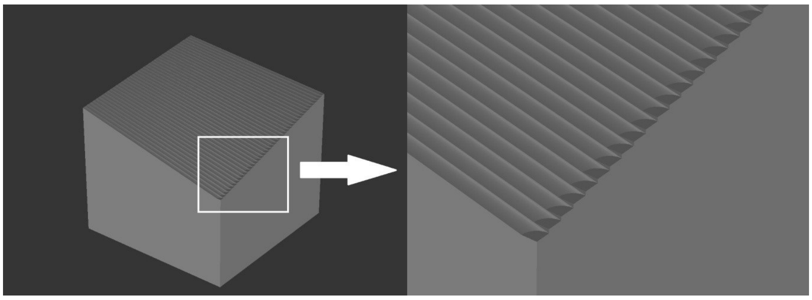


The NC program is operable on the following controls if software option 2 is activated (option no. 9):

- TNC 640
- TNC 620 as of NC software number 340 56x-03
- iTNC 530 as of NC software number 340 422-xx

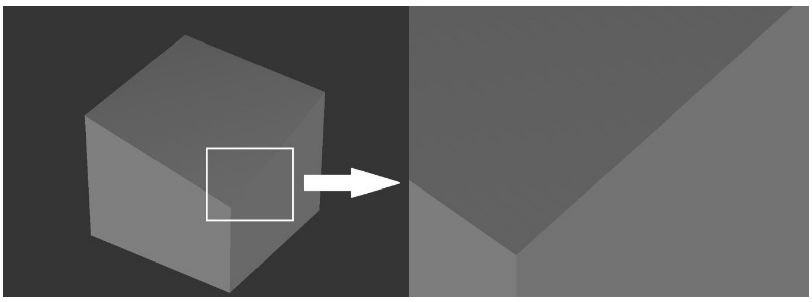
Requirement:

The 3-D ruled surface is to be machined with an end mill. If you execute this machining in three axes this results in contour innaccuracy, depending on the tool radius and number of milling paths. Edges are also created between the paths.



Solution:

In this NC program the control calculates a tool path in which the tool moves simultaneously in five axes. As a result the tool axis is always perpendicular to the surface to be machined. This movement minimizes any inaccuracies and edges.



Description of NC program 4020_en.h

In the NC program 4020_en.h, first define all parameters required for machining. Then define the workpiece blank and cycle call. In the NC program, the 40201_en.h NC program is then assigned in Cycle 12. Here you must modify the path specification if required.

The control positions the rotary axes to zero degrees. You must adapt the rotary axes to your kinematic configuration in the linear function programmed for this purpose. After moving to clearance height the control then calls a subprogram. The control carries out several calculations in this subprogram. After returning to the main program the control moves the tool to the starting position. With an M99, the control calls the NC program defined in Cycle 12.

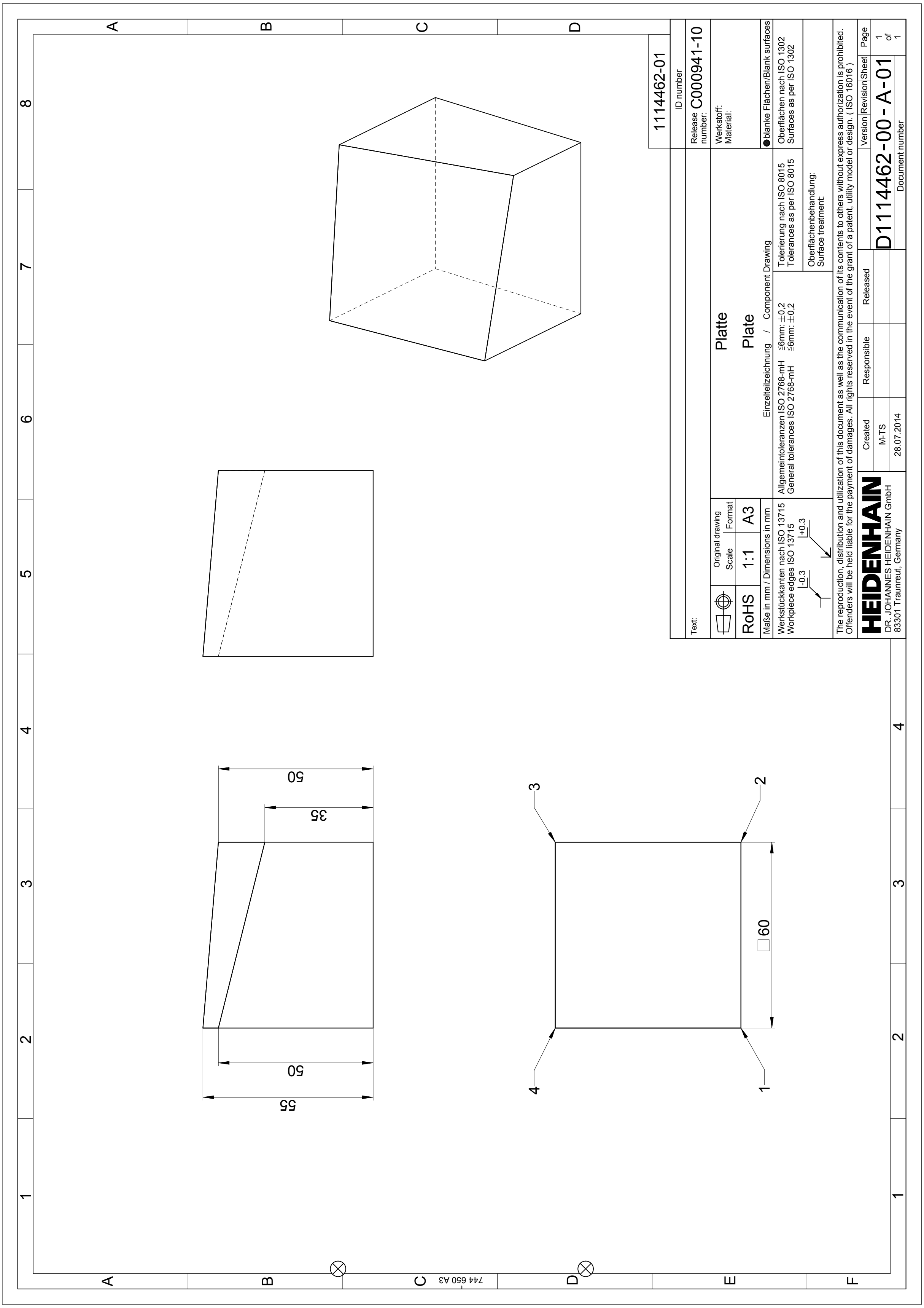
After the control has executed this NC program and returned to the 4020_en.h NC program, it positions the rotary axes back to zero degrees. You must again modify the axes to be traversed to your kinematic configuration in this NC block. The control then resets the TCPM function via the M129 command.

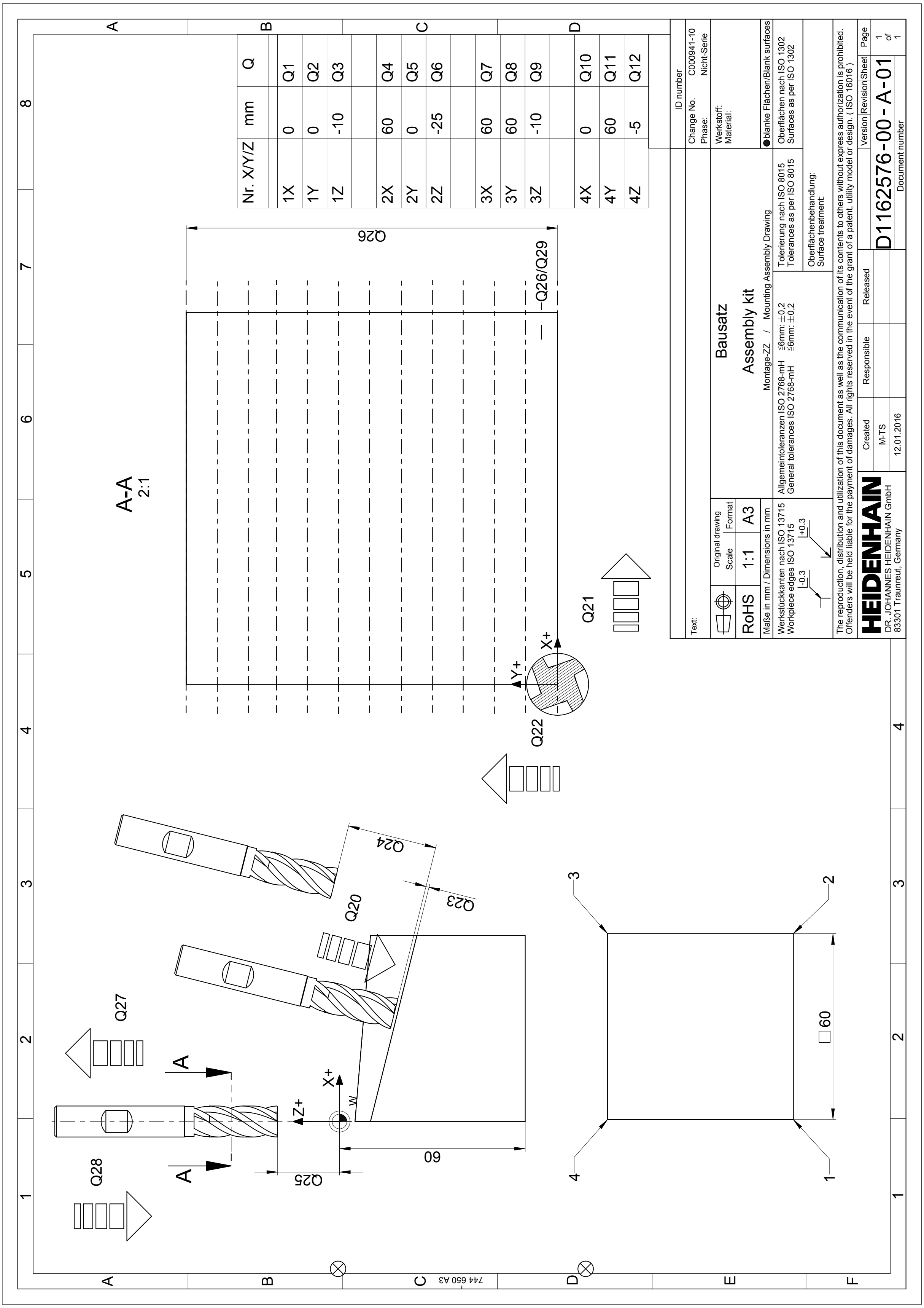
In the example program, a second tool call is defined for finishing. All parameters with different values for finishing are redefined. The control then traverses back to the starting point and calls the NC program defined in Cycle 12. After execution of this NC program the control returns the rotary axes to zero degrees. The control resets the TCPM function, withdraws the tool in the Z axis and terminates the NC program.

Parameter	Name	Meaning
Q1	X COORDINATE OF 1ST POINT	Absolute X coordinate of the first point
Q2	Y COORDINATE OF 1ST POINT	Absolute Y coordinate of the first point
Q3	Z COORDINATE OF 1ST POINT	Absolute Z coordinate of the first point
Q4	X COORDINATE OF 2ND POINT	Absolute X coordinate of the second point
Q5	Y COORDINATE OF 2ND POINT	Absolute Y coordinate of the second point
Q6	Z COORDINATE OF 2ND POINT	Absolute Z coordinate of the second point
Q7	X COORDINATE OF 3RD POINT	Absolute X coordinate of the third point
Q8	Y COORDINATE OF 3RD POINT	Absolute Y coordinate of the third point
Q9	Z COORDINATE OF 3RD POINT	Absolute Z coordinate of the third point
Q10	X COORDINATE OF 4TH POINT	Absolute X coordinate of the fourth point
Q11	Y COORDINATE OF 4TH POINT	Absolute Y coordinate of the fourth point
Q12	Z COORDINATE OF 4TH POINT	Absolute Z coordinate of the fourth point
Q20	FEED RATE FOR PECKING	Traversing speed of the tool in the Z axis
Q21	FEED RATE FOR MILLING	Traversing speed of the tool during machining in the X axis
Q22	FEED RATE FOR LATERAL STEPOVER	Traversing speed of the tool during lateral stepover in the Y axis
Q27	FEED RATE FOR RETRACTION	Traversing speed of the tool during retraction
Q28	FEED RATE FOR PRE-POSITIONING	Traversing speed of the tool during pre-positioning
Q23	ALLOWANCE FOR FLOOR	The oversize remaining after machining on the specified coordinates of the Z axis
Q24	INCREMENTAL HEIGHT DURING POSITIONING	Incremental value by which the control lifts off the tool from the surface for new positioning
Q25	SAFETY CLEARANCE	Incremental value at which the control pre-positions the tool in the Z axis at the first point
Q26	NUMBER OF CUTS	Number of machining paths in the Y axis
Q29	ROUGHING FACTOR	Value by which the control divides the number of cuts during roughing to traverse less machining paths

Description of NC program 40201_en.h

All calculations and tool movements required for machining are programmed in the 40201_en.h NC program. This NC program is structured independently of the kinematics, meaning that you do not need to change anything.


1114462-01
ID number
Release number: C00941-10
Werkstoff:
Material:
blanke Flächen/Blank surfaces
Oberflächen nach ISO 1302
Surfaces as per ISO 1302
Tolerierung nach ISO 8015
Tolerances as per ISO 8015
Oberflächenbehandlung:
Surface treatment:
Platte
Plate
Einzelteilzeichnung / Component Drawing
Allgemeintoleranzen ISO 2768-mH ≤6mm: ±0.2
General tolerances ISO 2768-mH ≤6mm: ±0.2
Original drawing
Scale 1:1
Format A3
RoHS
Maße in mm / Dimensions in mm
Werkstückkanten nach ISO 13715
Workpiece edges ISO 13715
-0.3
+0.3
Text:
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
M-TS
28.07.2014
Responsible
Released
D1114462-00-A-01
Document number
Version Revision Sheet Page
1 of 1
50
35
50
55
□60
1
2
3
4
744 650 A3


A-A
2:1
Nr. X/Y/Z | mm | Q
1X | 0 | Q1
1Y | 0 | Q2
1Z | -10 | Q3
2X | 60 | Q4
2Y | 0 | Q5
2Z | -25 | Q6
3X | 60 | Q7
3Y | 60 | Q8
3Z | -10 | Q9
4X | 0 | Q10
4Y | 60 | Q11
4Z | -5 | Q12
Q26
Q26/Q29
Q21
Q22
X+
Y+
Q24
Q23
Q20
Q27
Q28
Q25
60
□60
A
W
Z+
ID number
Change No. C000941-10
Phase: Nicht-Serie
Werkstoff:
Material:
blanke Flächen/Blank surfaces
Text:
Bausatz
Assembly kit
Montage-ZZ / Mounting Assembly Drawing
Original drawing
Scale 1:1
Format A3
RoHS
Maße in mm / Dimensions in mm
Werkstückkanten nach ISO 13715
Workpiece edges ISO 13715
Allgemeintoleranzen ISO 2768-mH ≤6mm: ±0.2
General tolerances ISO 2768-mH ≤6mm: ±0.2
Tolerierung nach ISO 8015
Tolerances as per ISO 8015
Oberflächen nach ISO 1302
Surfaces 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 M-TS 12.01.2016
Responsible
Released
D1162576-00-A-01
Document number
Version Revision Sheet Page 1 of 1
744 650 A3