



HEIDENHAIN



NC Solutions

Indexing a Rotary Table

Changing NC Programs of the iTNC 530 for
Compatibility with Successor Controls

English (en)
10/2023

Additional information

Copyright

All media files (e.g. texts, images, graphics and videos), including their parts, distributed or made available as part of the course are copyright-protected.

Insofar as the individual media file does not contain expressly deviating provisions, the following applies to use: Without the prior written consent of DR. JOHANNES HEIDENHAIN GmbH, the media files must not be passed on to other persons (including to other employees of the participating company) and must not be duplicated, modified, archived, stored on a server, included in newsgroups, used in online services, stored on CD-ROMs or other data media, or used in printed publications. This also applies to the use of excerpts or sections of the media files. In addition, the general Terms of Use for the web pages of HEIDENHAIN shall also apply.

Copyright © DR. JOHANNES HEIDENHAIN GmbH.

Exclusion of liability

DR. JOHANNES HEIDENHAIN GmbH cannot guarantee the correctness, accuracy and completeness of the information, and therefore cannot assume liability for any type of loss or damage.

Table of contents

1	About this document.....	4
2	Scenarios.....	5
3	Possible solutions.....	11

1 About this document

This document helps you modify the iTNC 530's NC programs for indexing a rotary table so that they are compatible with the successor controls.

With the successor controls of the iTNC 530, some of the NC functions for indexing a rotary axis have changed and necessitate a different procedure. You can adapt the NC programs of the iTNC 530 for use on all successor controls.

This document describes typical scenarios combining inclined machining with the indexing of a rotary table.

More information on the functions used is provided in the respective User's Manuals matching the software version of your control.

Required modifications

NC programs published in the database are suggestions for solutions. The NC programs must be adapted before being used on a machine.

Modifications are always necessary for:

- Tools
- Cutting parameters
- Feed rates
- Clearance height
- Machine-specific positions (e.g. **M91**)
- Paths of program calls

Some NC programs depend on the machine kinematics. Adapt these NC programs to your machine kinematics before the first test run.

2 Scenarios

You can combine specific NC functions to index a rotary table and perform an inclined machining operation, for example. In this case, the behavior of the TNC 640 and TNC7 is incompatible with the behavior of the iTNC 530.

This document describes the following scenarios:

- NC program with an inclined machining operation
Further information: "NC program with M128, Cycle 7, and Cycle 10", Page 6
- NC program with an inclined machining operation and pre-positioning using **PLANE**
Further information: "NC program with M128, Cycle 7, Cycle 10, and PLANE SPATIAL", Page 8

2.1 NC program with M128, Cycle 7, and Cycle 10

If you program an inclined machining operation by using a datum shift of the rotary table, there will be incompatibilities with the successor controls of the iTNC 530:

Control	Result
iTNC 530	Correct
TNC 640 up to and including software-version 34059x-08	Correct
TNC 640 with software version 34059x-09 and later TNC7	Incompatible behavior and scrap

NC example

0 BEGIN PGM CASE_1 MM	
1	; Section with loaded values
2 Q1 = +6	; Number of operations
3 Q2 = Q1 - +1	; Calculation of repetitions
4 Q3 = -60	; Starting angle
5	; Beginning of main program
* - ...	
12 LBL 9	; Program-section repeat
13 CYCL DEF 7.0 NULLPUNKT	; Shift the datum of the rotary axis to the calculated angle
14 CYCL DEF 7.1 CQ3	
15 CYCL DEF 10.0 DREHUNG	; Rotate the reference system
16 CYCL DEF 10.1 ROTQ3	
17 M128	; Automatically compensate for tool inclination
18 L X+0 Y+10 Z+50 A-80 C+0 FMAX	; Pre-position and incline the tool
19 L X+0 Y+10 Z+10 F2000	; Tool infeed
20 L X+0 Y+90 Z-20 A-15 C+0	; Machine the contour and change the inclination
21 L X+0 Y+90 Z+50	; Retract the tool
22 L X+0 Y+10 Z+50 FMAX	; Pre-position the tool
23 M129	; Deactivate M128
24 Q3 = Q3 + (+360 / Q1)	; Calculate angle for next machining operation
25 CALL LBL 9 REPQ2	
26	; End the program, including resetting and retraction
* - ...	
37 M30	
38 END PGM CASE_1 MM	

Possible solutions

Possible solution	Further information
Recommendation: Change the NC program	Page 11
Workaround: Possible with software version 34059x-09 and later ■ Change the machine parameter ■ Change the preset table as needed	Page 12

2.2 NC program with M128, Cycle 7, Cycle 10, and PLANE SPATIAL

i This scenario describes the same machining process as the previous scenario. The **PLANE** function is only used to pre-position the rotary table.

If you program an inclined machining operation by using a datum shift of the rotary table and you use **PLANE** for pre-positioning, there will be incompatibilities with the successor controls of the iTNC 530:

Control	Result
iTNC 530	Correct
TNC 640 up to and including software-version 34059x-08	Error message: Tilted working plane not allowed
TNC 640 with software version 34059x-09 and later TNC7	Incompatible behavior and scrap

NC example

0 BEGIN PGM CASE_2 MM	
1	; Section with loaded values
2 Q1 = +6	; Number of operations
3 Q2 = Q1 - +1	; Calculation of repetitions
4 Q3 = -60	; Starting angle
5 ;	; Beginning of main program
* - ...	
12 LBL 9	; Program-section repeat
13 CYCL DEF 7.0 NULLPUNKT	; Shift the datum of the rotary axis to the calculated angle
14 CYCL DEF 7.1 CQ3	
15 CYCL DEF 10.0 DREHUNG	; Rotate the reference system
16 CYCL DEF 10.1 ROTQ3	
17 PLANE SPATIAL SPA-80 SPB+0 SPCQ3 TURN F5000 SEQ- TABLE ROT	; Tilt the working plane
18 PLANE RESET STAY	; Mathematically reset the working plane
19 M128	; Automatically compensate for tool inclination
20 L X+0 Y+10 Z+50 A-80 C+0 FMAX	; Pre-position and incline the tool
21 L X+0 Y+10 Z+10 F2000	; Tool infeed
22 L X+0 Y+90 Z-20 A-15 C+0	; Machine the contour and change the inclination
23 L X+0 Y+90 Z+50	; Retract the tool
24 L X+0 Y+10 Z+50 FMAX	; Pre-position the tool
25 M129	; Deactivate M128
26 Q3 = Q3 + (+360 / Q1)	; Calculate angle for next machining operation
27 CALL LBL 9 REPQ2	
28	; End the program, including resetting and retraction
* - ...	
39 M30	
40 END PGM CASE_2 MM	

Possible solutions

Possible solution	Further information
Recommendation: Change the NC program	Page 11
Workaround: Possible with software version 34059x-09 and later <ul style="list-style-type: none"> ■ Change the machine parameter ■ Change the preset table as needed 	Page 12

Note

With software version 34059x-09, the behavior of Cycle **7** in combination with a rotary axis was changed to enable the simultaneous display of the workpiece and the machine in the simulation. Cycle **7** has the same effect as an offset value of the rotary axis. Both functions are used to align the workpiece on the machine table.

3 Possible solutions

3.1 Changing the NC program

If you change the NC program as specified, you can run it on all of the successor controls whose behavior will then be identical.

Measures

- ▶ Use **PLANE** functions with spatial angles for the following functions:
 - Tilting the working plane or inclination of tool
 - Indexing the rotary axis
- ▶ Use **FUNCTION TCPM** with spatial angles to change the tool inclination
- ▶ Remove the datum shift for the indexed rotary axis

NC example

0 BEGIN PGM SOLUTION MM	
1	; Section with loaded values
2 Q1 = +6	; Number of operations
3 Q2 = Q1 - +1	; Calculation of repetitions
4 Q3 = -60	; Starting angle
5	; Beginning of main program
* - ...	
13 PLANE SPATIAL SPA-80 SPB+0 SPCQ3 TURN MB MAX F500 SEQ- TABLE ROT	; Tilt the working plane
14 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	; Automatically compensate for tool inclination by using spatial angles
15 LBL 9	; Program-section repeat
16 PLANE SPATIAL SPA+0 SPB+0 SPCQ3 STAY SEQ-	; Mathematically tilt to calculated angle
17 L X+0 Y+10 Z+50 A-80 C+0 FMAX	; Pre-position and incline the tool
18 L X+0 Y+10 Z+10 F2000	; Tool infeed
19 L X+0 Y+90 Z-20 A-15 C+0	; Machine the contour and change the inclination
20 L X+0 Y+90 Z+50	; Retract the tool
21 L X+0 Y+10 Z+50 FMAX	; Pre-position the tool
22 Q3 = Q3 + (+360 / Q1)	; Calculate angle for next machining operation
23 CALL LBL 9 REPQ2	
24 ;	
25 PLANE RESET STAY	
26 M129	; Deactivate TCPM
27	; End the program, including resetting and retraction
* - ...	
36 M30	
37 END PGM SOLUTION MM	

3.2 Changing the machine parameter

The machine parameter **presetToAlignAxis** must be changed by the machine manufacturer, or changing by the machine operator must be enabled.

i If the machine parameter **presetToAlignAxis** is changed to **FALSE** for the rotary axis, the control will not consider offset values during program run. This can cause other NC programs leading to incorrect results.

Measures

- ▶ Change the machine parameter
 - If the machine manufacturer changes the machine parameter, you must also take into account the changed setting in other NC programs.
or
 - If the machine parameter is enabled for users, you need to check the machine parameter and adapt it as needed, before switching NC programs.
- ▶ Review the preset table

Solution

- Change the machine parameter **presetToAlignAxis** (no. 300203) as follows:
presetToAlignAxis_C = FALSE
- Review the preset table and, if needed, change it as follows:
C_OFFS = 0

Note

CFGREAD allows you to query the value of the machine parameter.