HEIDENHAIN

Interpolation Turning and Milling with the TNC 640

Instructor: Michael Wiendl

Company: Dr. Johannes HEIDENHAIN GmbH
Position: Trainer for NC Programming

Application

- Creation of rotationally symmetric contours in any machining plane
- There are two variants for machining:

→ Interpolation turning (**coupled** spindle)

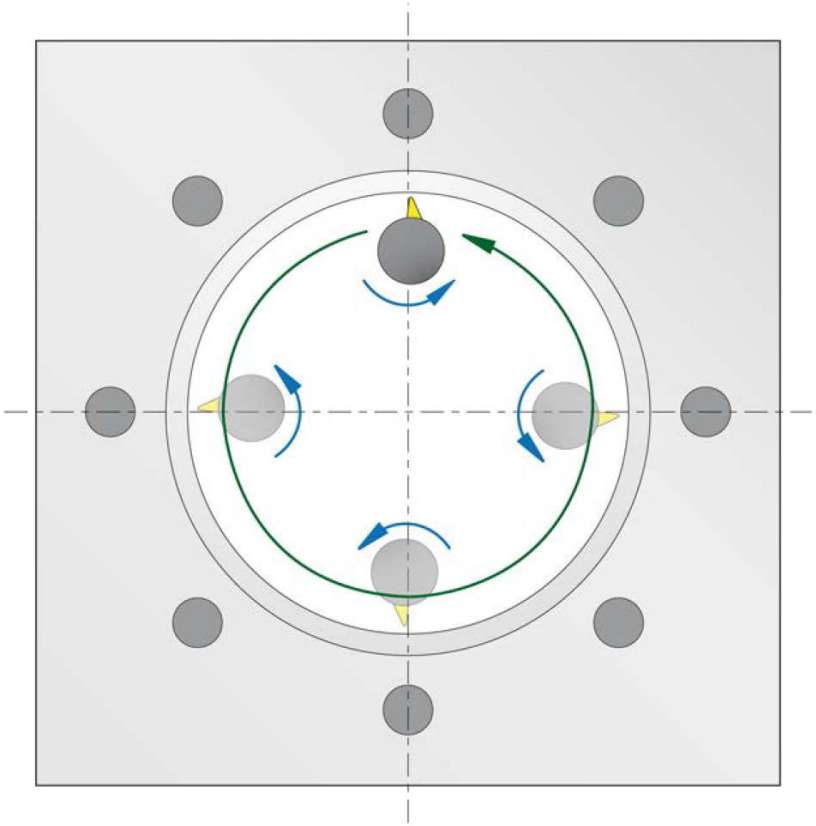
→ Interpolation milling (spindle is **not coupled**)

Interpolation turning

- During a circular motion, the cutter orients itself away from the center for **inside machining operations**
- During a circular motion, the cutter orients itself toward the center for **outside machining operations**

Interpolation milling

- Machining of rotationally symmetric contours with a milling tool



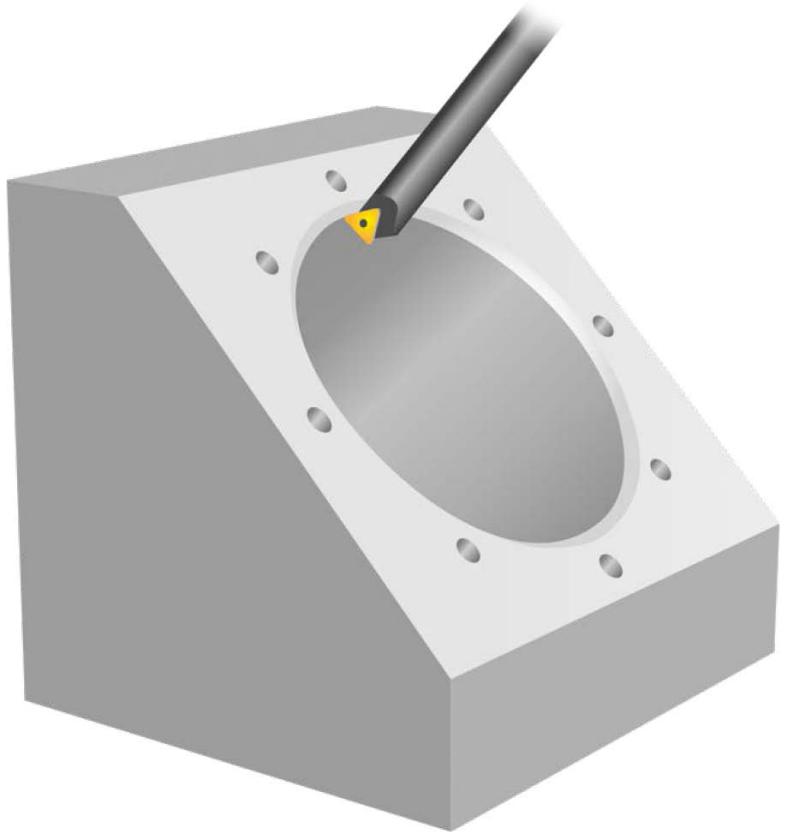
Programming:

Two cycles are available for programming:

- Cycle 292 *INTERPOLATION TURNING, CONTOUR FINISHING*
- Cycle 291 *INTERPOLATION TURNING, COUPLING*

→ Cycle 292 also needs the contour to be described in a LBL as well as the assignment via Cycle 14

→ Option 98 is required for interpolation turning

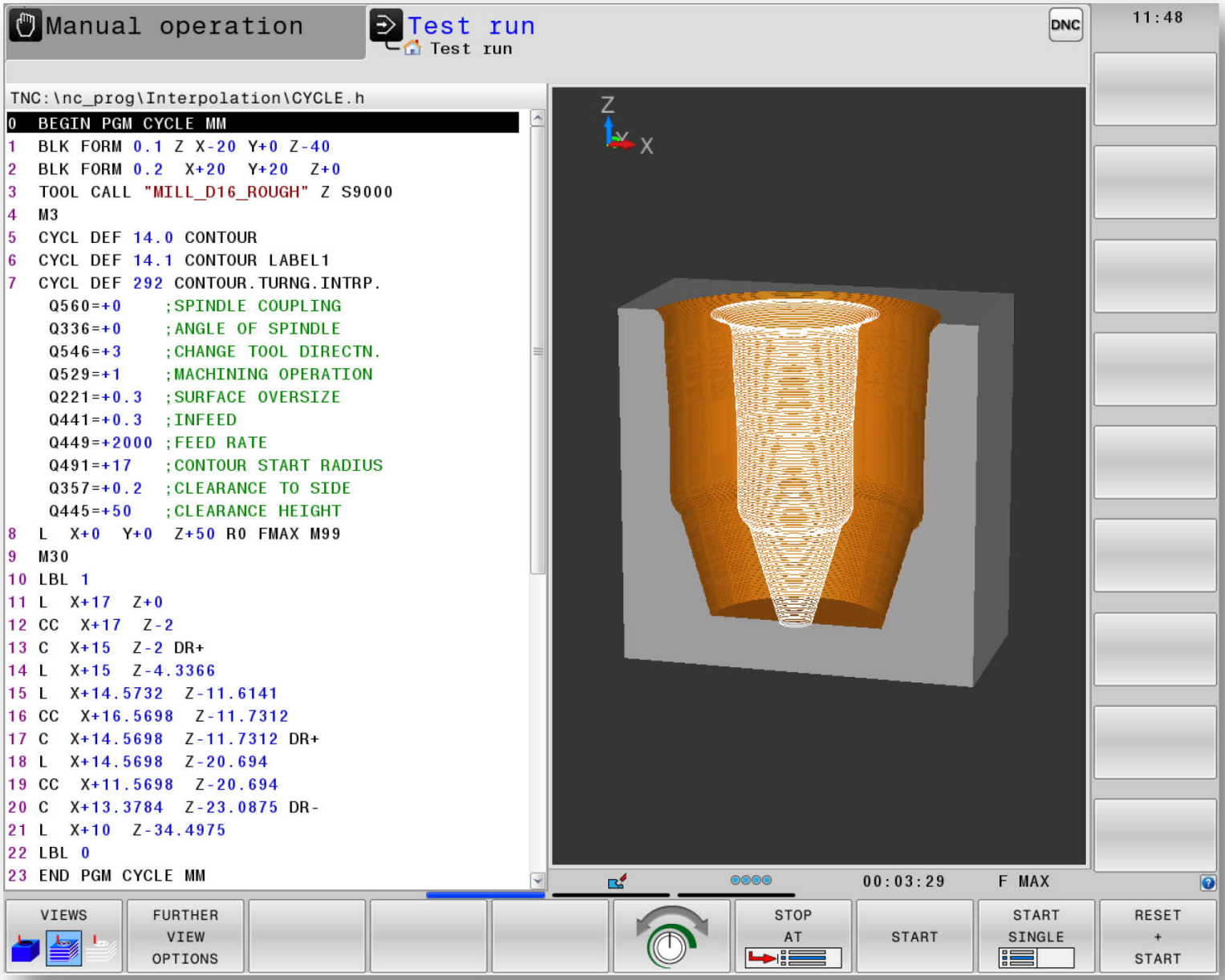


HEIDENHAIN

Programming

Cycle 292

Application Example – Cycle 292

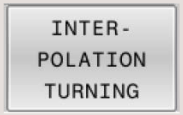


Programming – Cycle 292

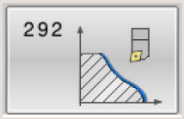
Programming:

- CYCL DEF key

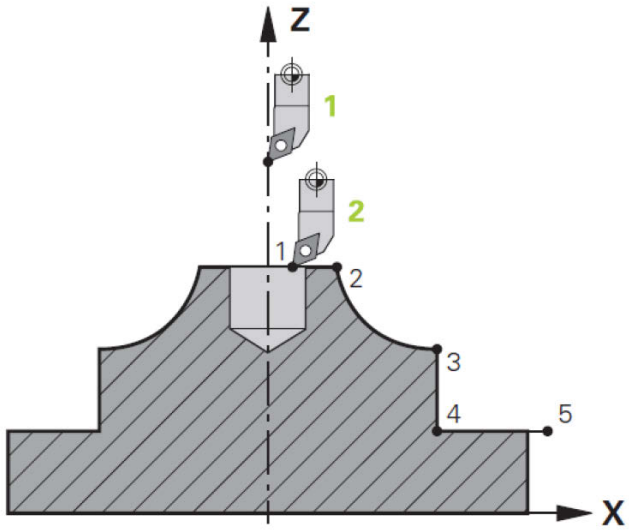
- SPECIAL CYCLES soft key

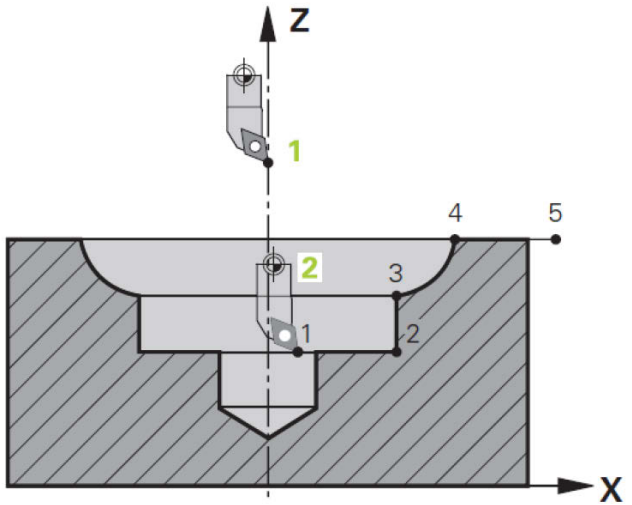


- INTERPOLATION TURNING soft key



- Cycle 292





Contour:

- Cycle 292 uses a contour description to generate a rotationally symmetric machining operation in the Z/X plane (tool axis Z)
- You program the contour in a subprogram
- In the program you use a Cycle 14 to assign this contour to Cycle 292

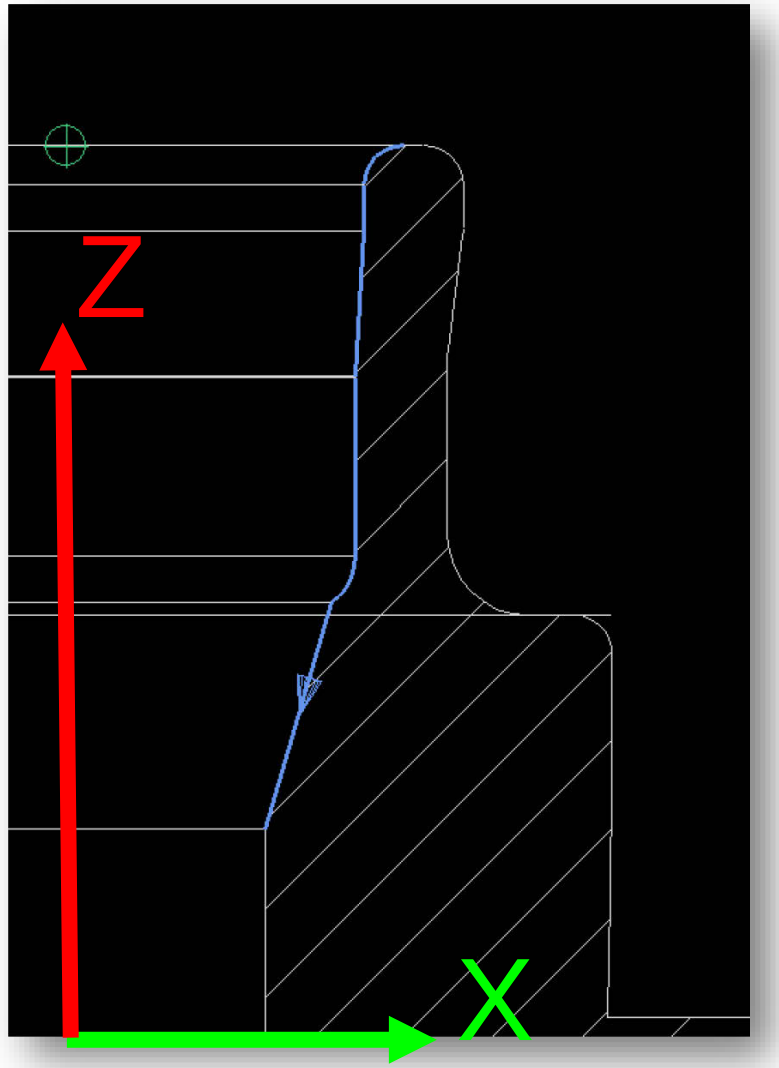
The following must be noted when programming the contour:

→ Contour description includes a radius (X axis)

→ No back cutting

→ Monotonously rising or falling contour description

→ Programming direction = Machining direction



Programming – Cycle 292

Contour:

- Cycle 292 uses a contour description to generate a rotationally symmetric machining operation in the Z/X plane (tool axis Z)
- You program the contour in a subprogram
- In the program you use a Cycle 14 to assign this contour to Cycle 292

The following must be noted when programming the contour:

→ Contour description includes a radius (X axis)

→ No back cutting

→ Monotonously rising or falling contour description

→ Programming direction = Machining direction

```
LBL 1
L   X+17   Z+0
CC   X+17   Z-2
C   X+15   Z-2 DR+
L   X+15   Z-4.3366
L   X+14.5732   Z-11.6141
CC   X+16.5698   Z-11.7312
C   X+14.5698   Z-11.7312 DR+
L   X+14.5698   Z-20.694
CC   X+11.5698   Z-20.694
C   X+13.3784   Z-23.0875 DR-
L   X+10   Z-34.4975
LBL 0
```

Tools:

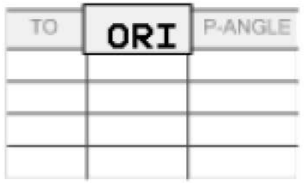
Spindle coupling off, Q560=0

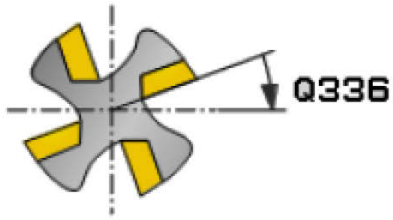
- Milling: Define the milling cutter in the tool table (tool.t) as usual

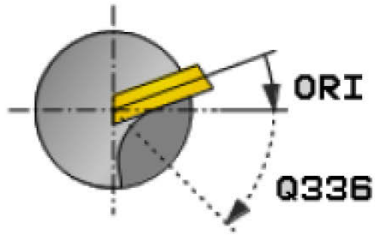
Spindle coupling on, Q560=1

- Turning:
  - Define the turning tool in the tool table (tool.t) as a milling tool
  - Define the milling tool in the tool table (tool.t) as a milling tool (in order to then use it as a turning tool)
  - Define the turning tool in the turning tool table (toolturn.trn)

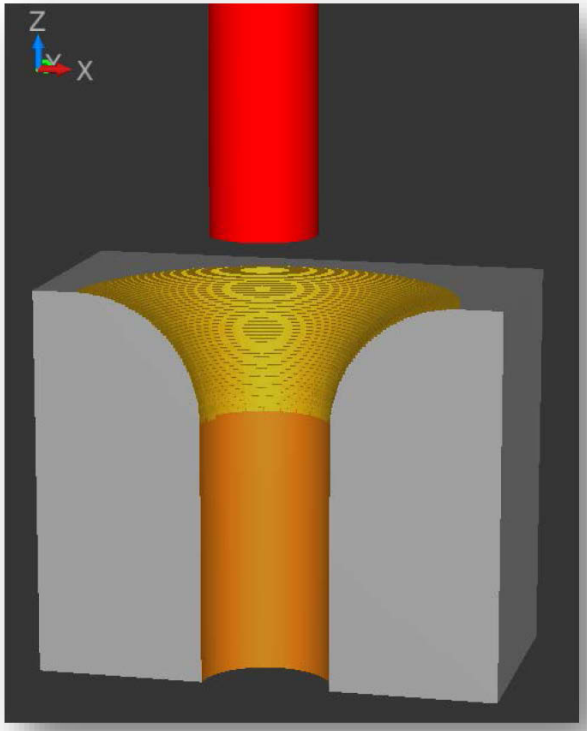
→ See the manual for the tool definition in tool.t and toolturn.trn

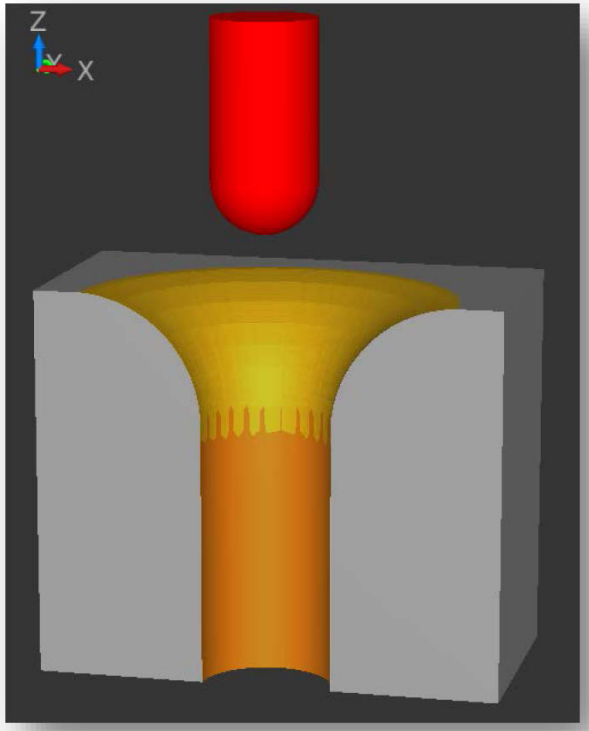


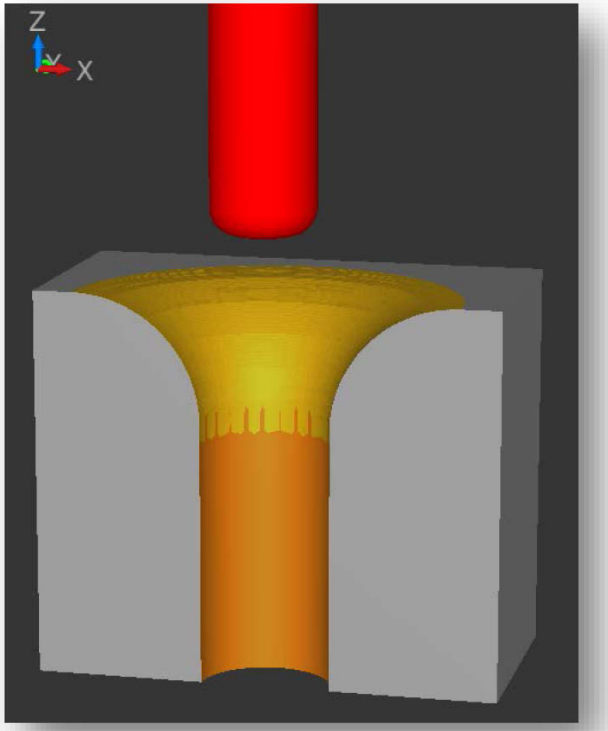




End mill	Ball-nose cutter	Toroid cutter
R>0 / R2=0	R = R2	R2>0<R







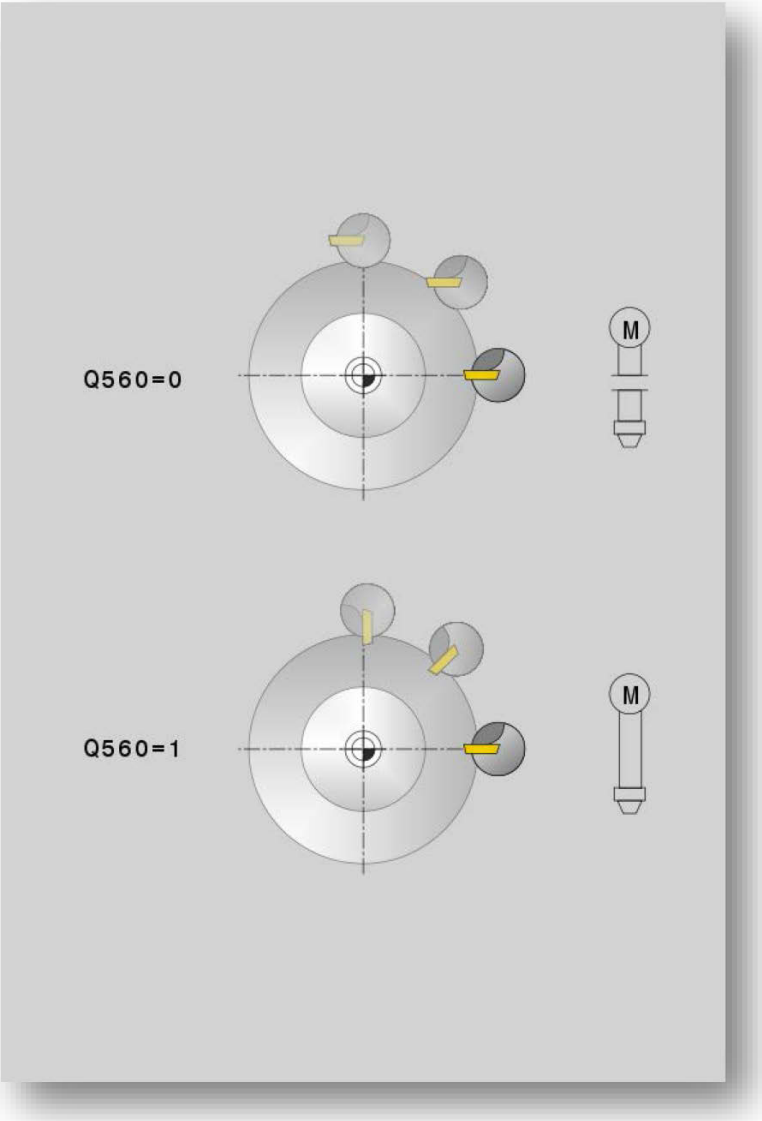
Programming – Cycle 292

Programming:

- You use **Q560** to specify whether the spindle should be coupled or not during machining

→ Interpolation turning **Q560 = 1**

→ Interpolation milling **Q560 = 0**

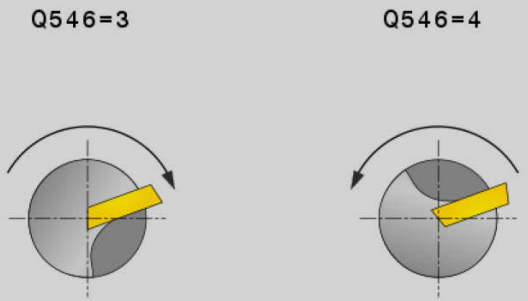


Programming:

- You use **Q545** to specify the machining direction

→ For interpolation turning you have to enter 3 or 4 here, depending on the position of the cutting edge

→ For interpolation milling, the entries 3 and 4 specify whether the contour is machined with climb milling or with up-cut milling



Programming:

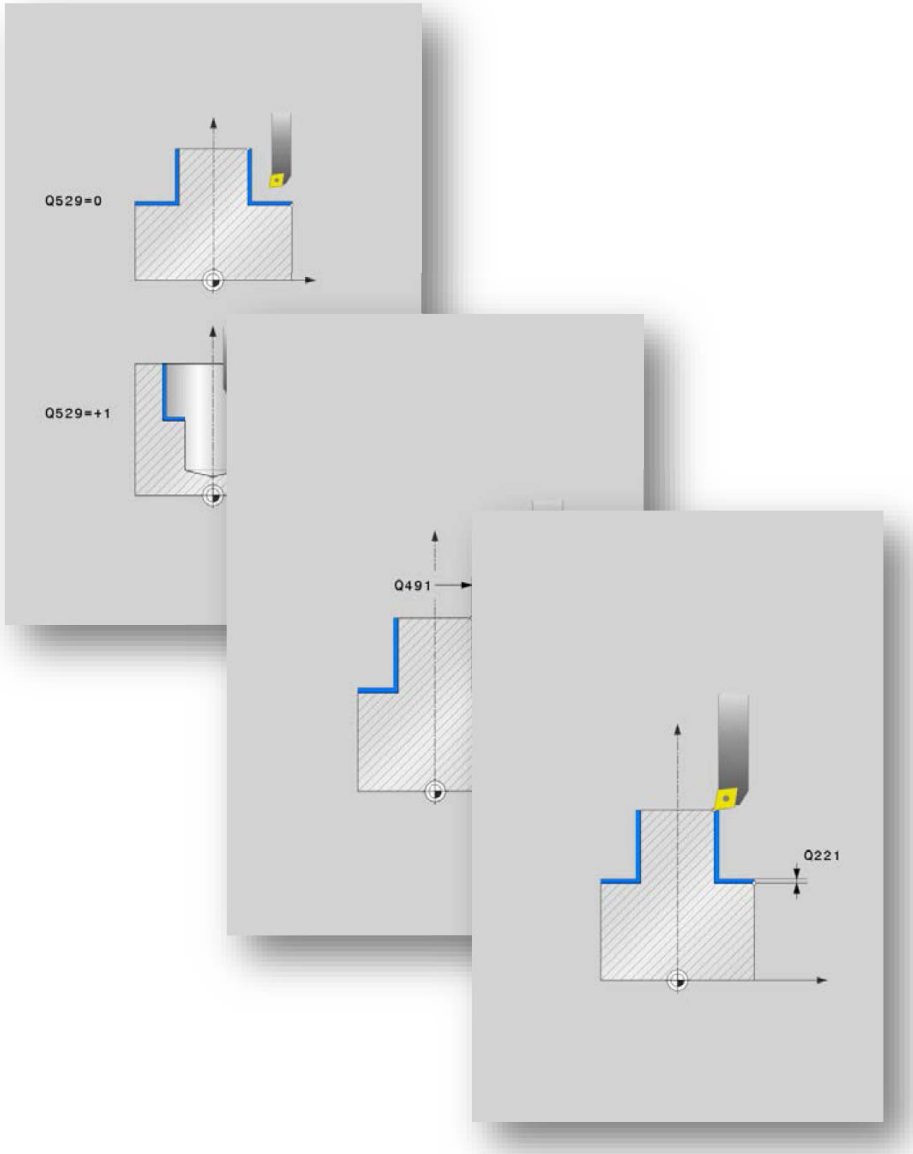
- You use **Q529** to specify the side to be machined

→ Outside machining **Q529 = 0**

→ Inside machining **Q529 = 1**

- In **Q491** you program the starting point for machining. This information is contained in the contour label

- In **Q221** you define the oversize for this operation



Programming – Cycle 292

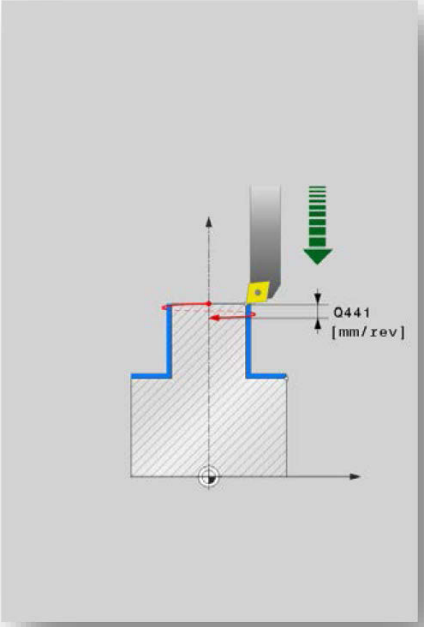
Programming:

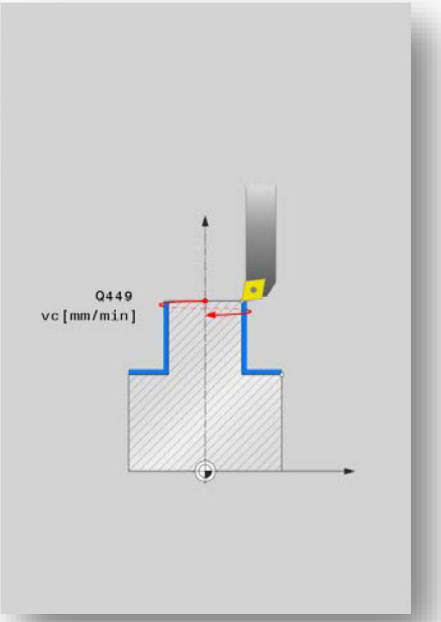
- You define the infeed per revolution in **Q441** (mm/rev)

- **Q449** Feed rate in reference to the contour starting point
  The feed rate is adapted as follows depending on the type of machining (Q529):

→ **Q529 = 1**: Feed rate of the tool center-line path is reduced for inside machining

→ **Q529 = 0**: Feed rate of the tool center-line path is increased for outside machining





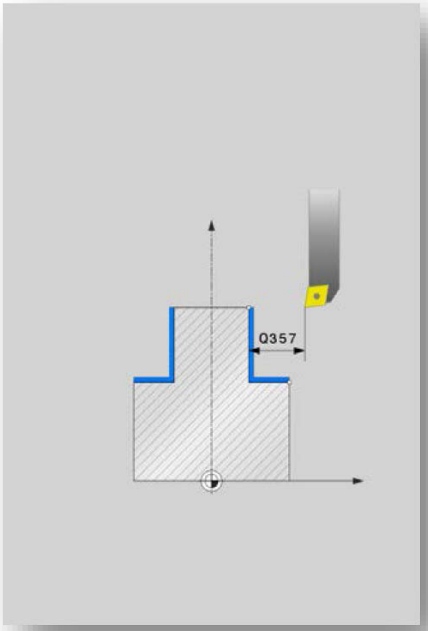
Programming – Cycle 292

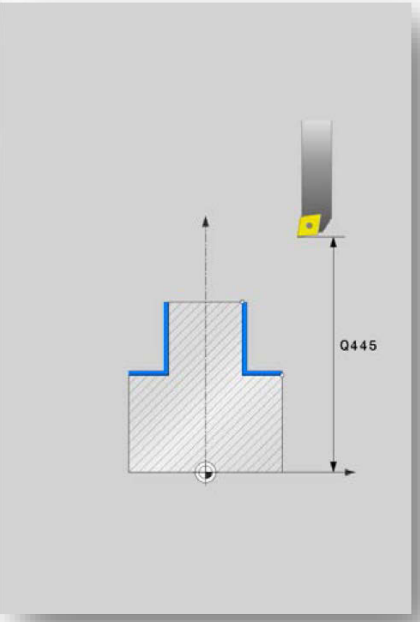
Programming:

- **Q357** Clearance to side
- **Q445** Clearance height that is positioned to after the machining operation

Cycle call:

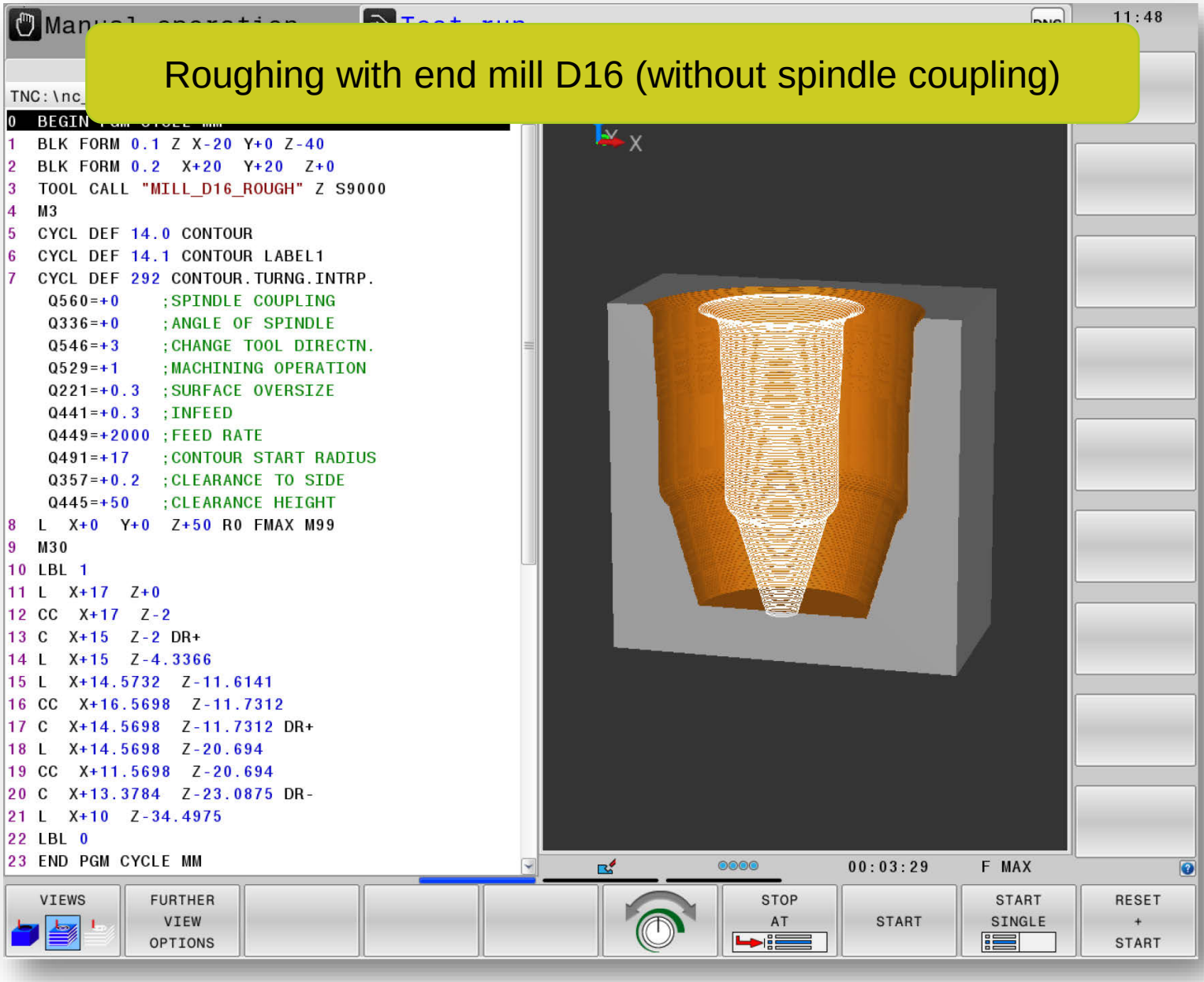
- Center the tool
- Call the cycle using one of the following functions:
  - M99
  - CYCL CALL
  - CYCL CALL POS
  - CYCL CALL PAT





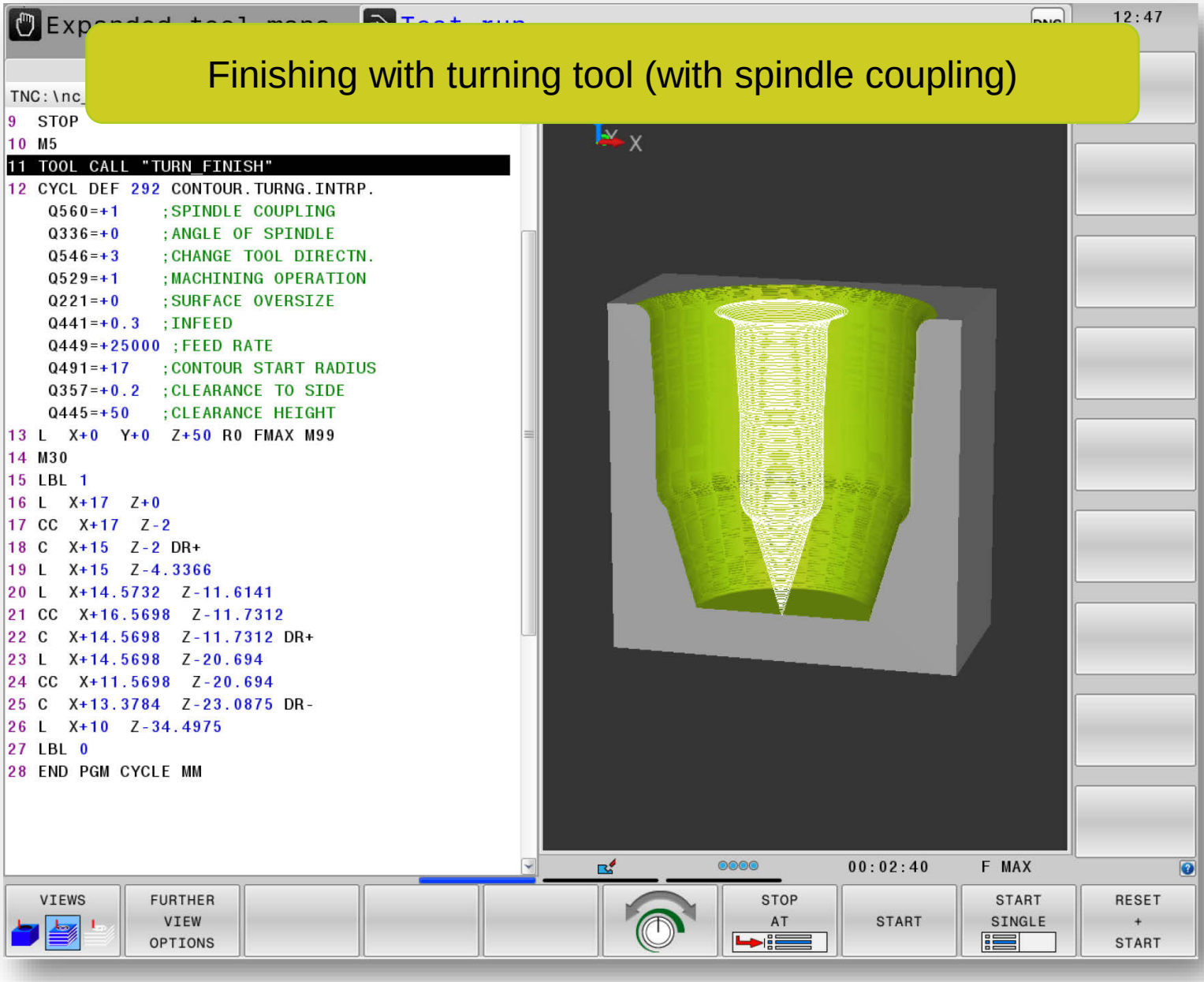
Application Example – Cycle 292

Roughing with end mill D16 (without spindle coupling)



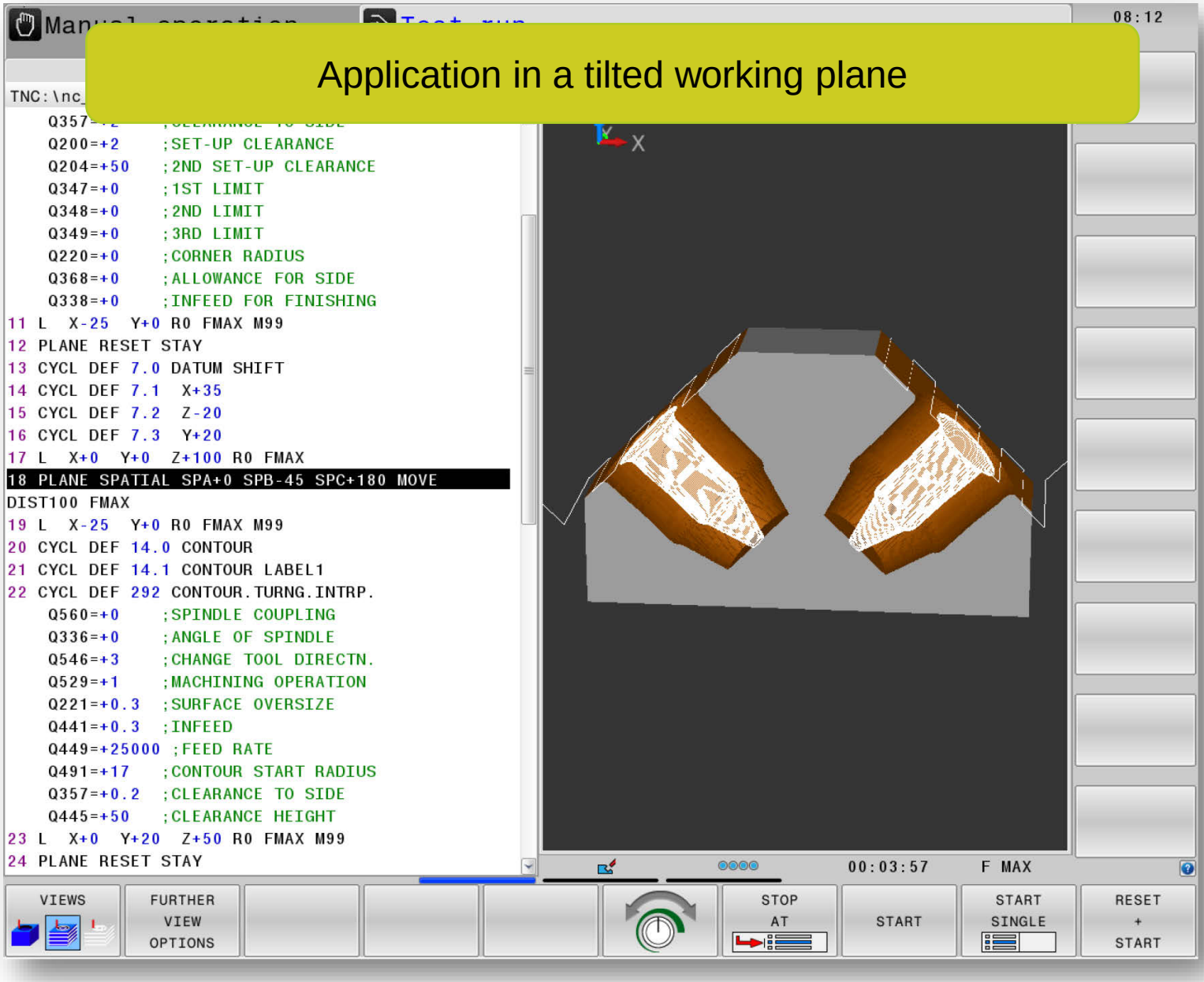
Application Example – Cycle 292

Finishing with turning tool (with spindle coupling)



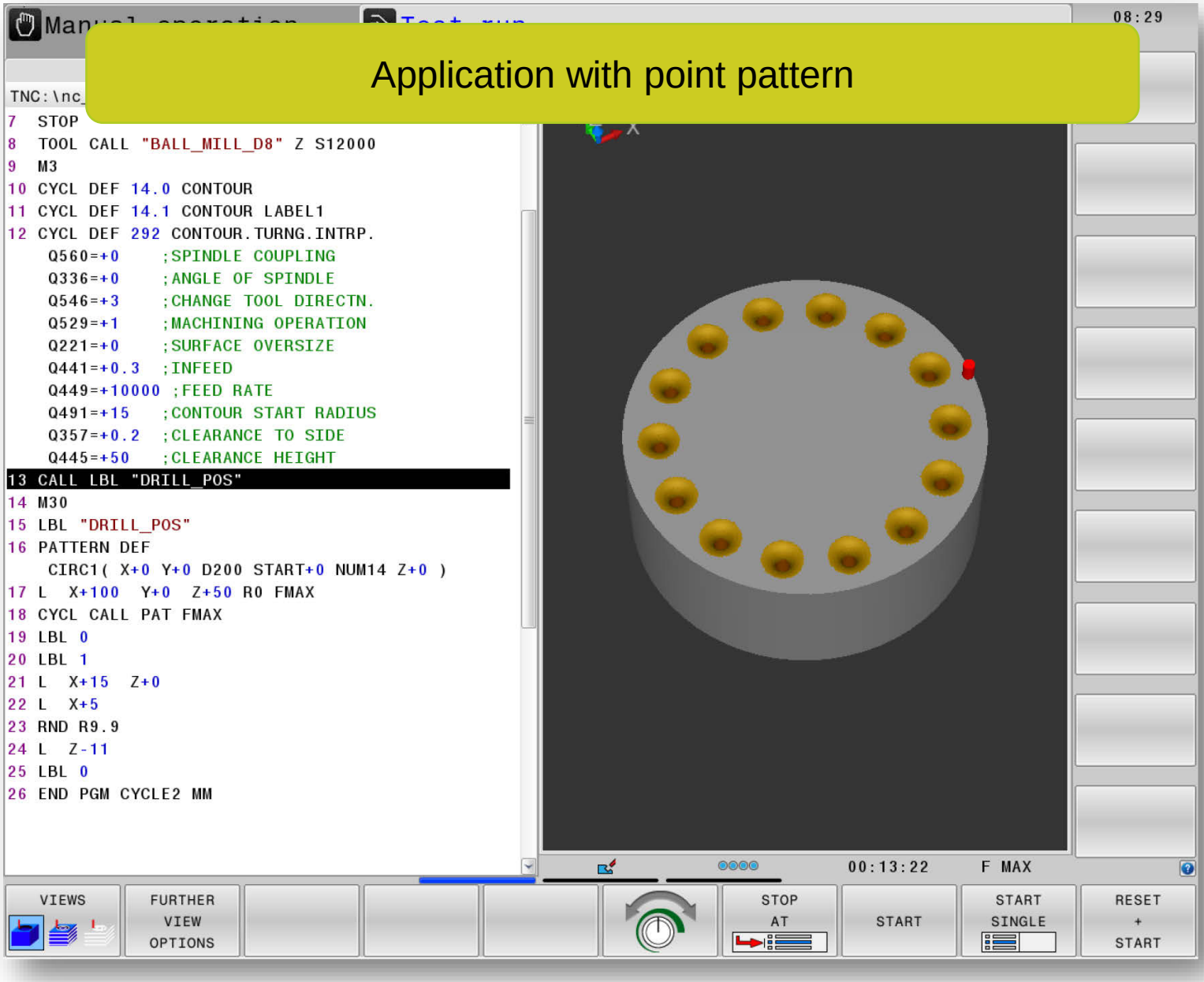
Application Example – Cycle 292

Application in a tilted working plane



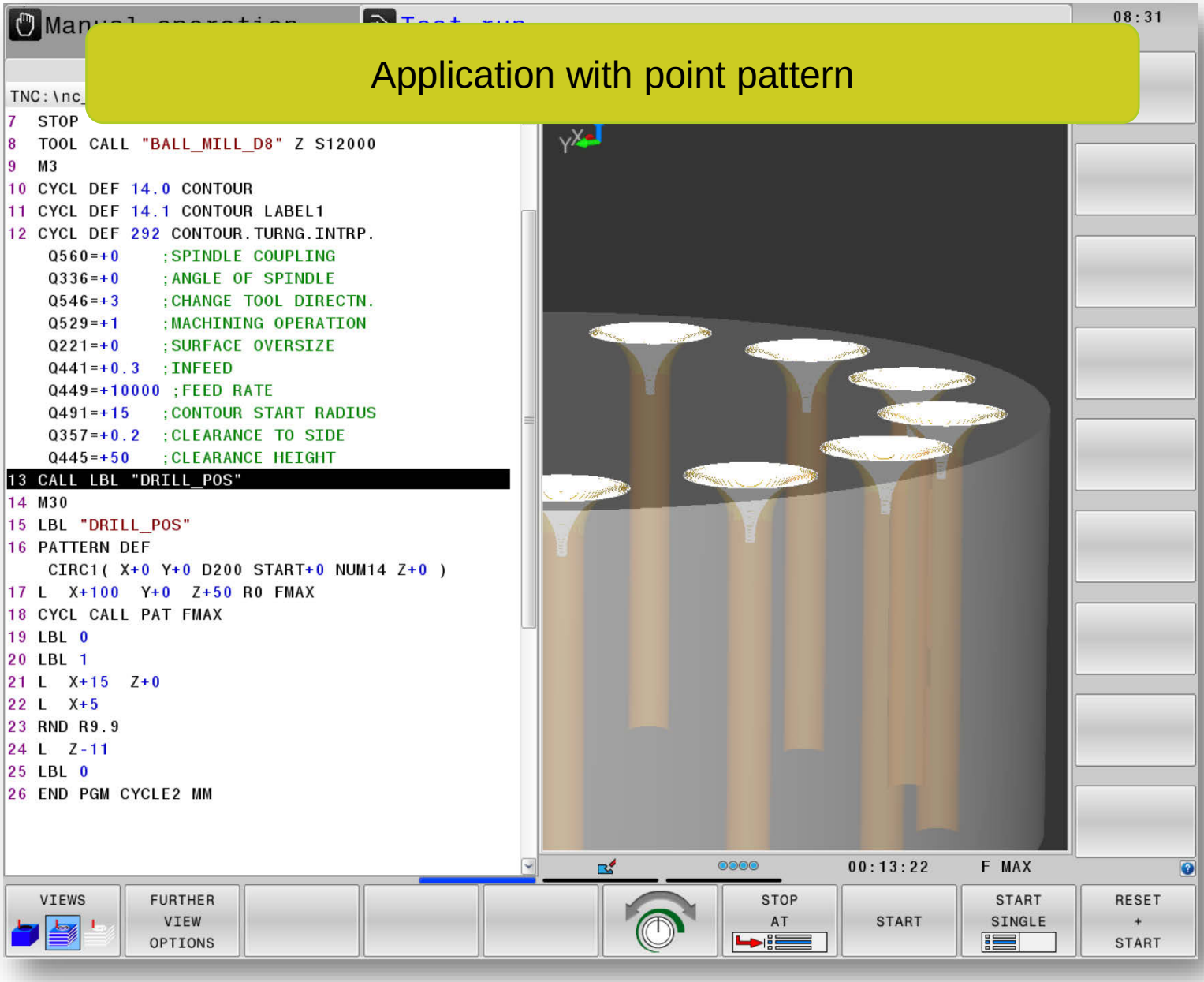
Application Example – Cycle 292

Application with point pattern



Application Example – Cycle 292

Application with point pattern



HEIDENHAIN

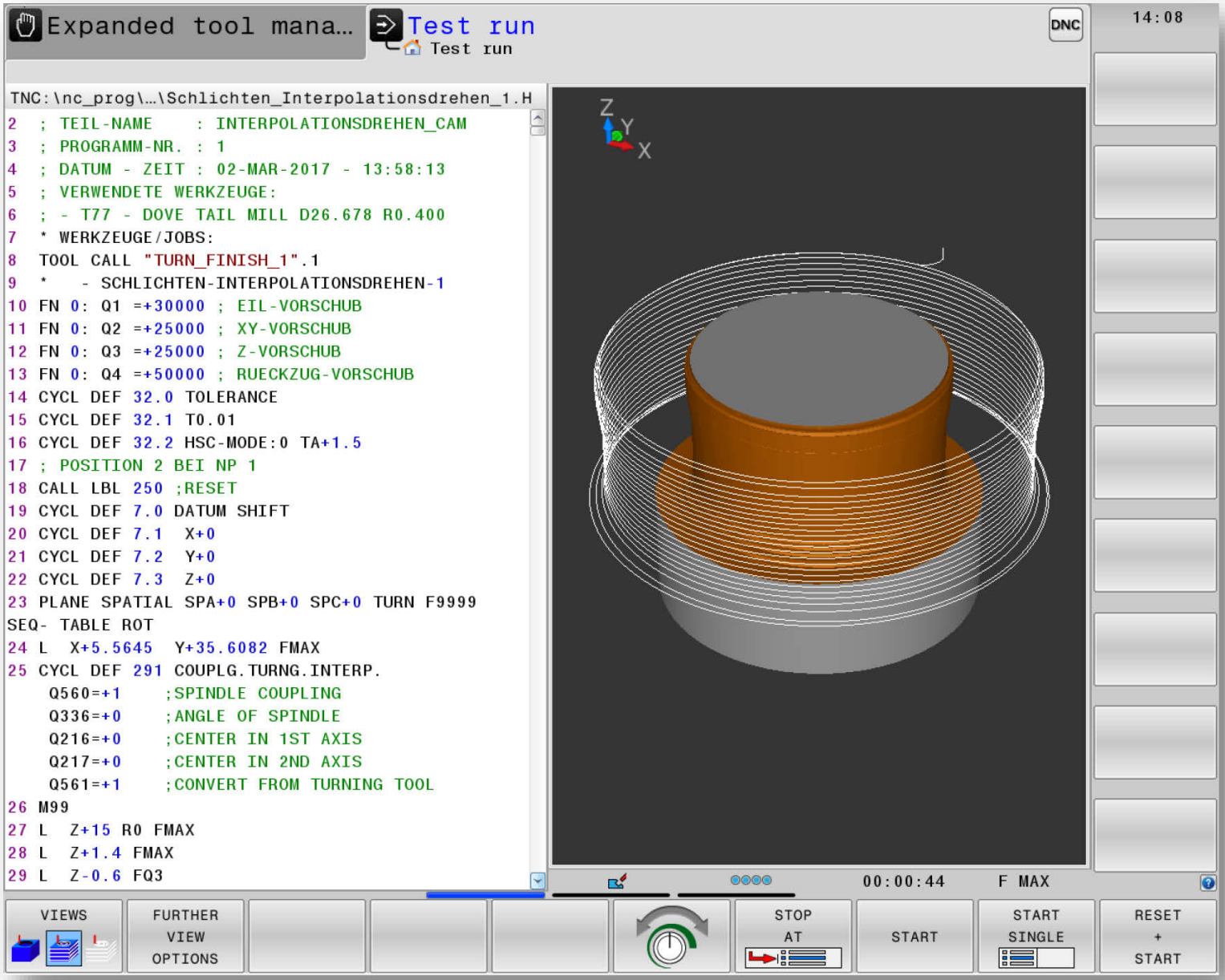
HEIDENHAIN

HEIDENHAIN

Programming

Cycle 291

Application Example – Cycle 291

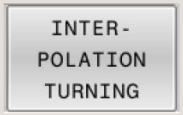


Programming – Cycle 291

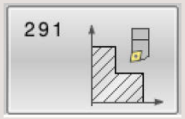
Programming:

- CYCL DEF key

- SPECIAL CYCLES soft key



- INTERPOLATION TURNING soft key



- Cycle 291

Cycle 291 is CALL-active, meaning that after you have programmed the cycle, you must call it with M99 or with CYCL CALL.

Programming – Cycle 291

Contour:

- Cycle 291 couples the spindle to a circular movement (center of rotation is defined in the cycle)
- The contour must be fully programmed, e.g. 3-D spiral path
- You must switch coupling on before machining
- You must switch coupling off after machining

Sample cases:

→ Contour description with back cutting

→ Contour description from CAM system

Please note:

→ Rotational direction (CW or CCW), machining direction (up or down), etc. are determined by the contour description.

```
L  X-6.7076  Y-32.2786  Z-1.1184
L  X-6.3538  Y-32.3518  Z-1.1211
L  X-5.9983  Y-32.4206  Z-1.1226
L  X-5.6423  Y-32.4861  Z-1.1253
L  X-5.2228  Y-32.5571  Z-1.1267
L  X-4.8027  Y-32.6238  Z-1.1299
L  X-4.3808  Y-32.6841  Z-1.1315
L  X-3.9583  Y-32.74  Z-1.1349
L  X-3.5964  Y-32.7827  Z-1.1363
L  X-3.2341  Y-32.8221  Z-1.1391
L  X-2.8709  Y-32.8568  Z-1.1405
L  X-2.5075  Y-32.8882  Z-1.1433
L  X-2.1434  Y-32.9149  Z-1.1448
L  X-1.7791  Y-32.9383  Z-1.1475
L  X-1.4143  Y-32.9568  Z-1.149
L  X-1.0493  Y-32.9722  Z-1.1518
L  X-0.6213  Y-32.9847  Z-1.1545
L  X-0.196  Y-32.9915  Z-1.157
L  X+0.2322  Y-32.9928  Z-1.1595
L  X+0.6575  Y-32.9885  Z-1.162
L  X+1.0211  Y-32.9801  Z-1.1635
L  X+1.3846  Y-32.9685  Z-1.1662
L  X+1.7476  Y-32.9522  Z-1.1677
L  X+2.1106  Y-32.9326  Z-1.1705
L  X+2.4729  Y-32.9082  Z-1.1719
L  X+2.8351  Y-32.8806  Z-1.1747
L  X+3.1964  Y-32.8484  Z-1.1762
L  X+3.5575  Y-32.8129  Z-1.1789
L  X+3.9846  Y-32.7652  Z-1.1815
```

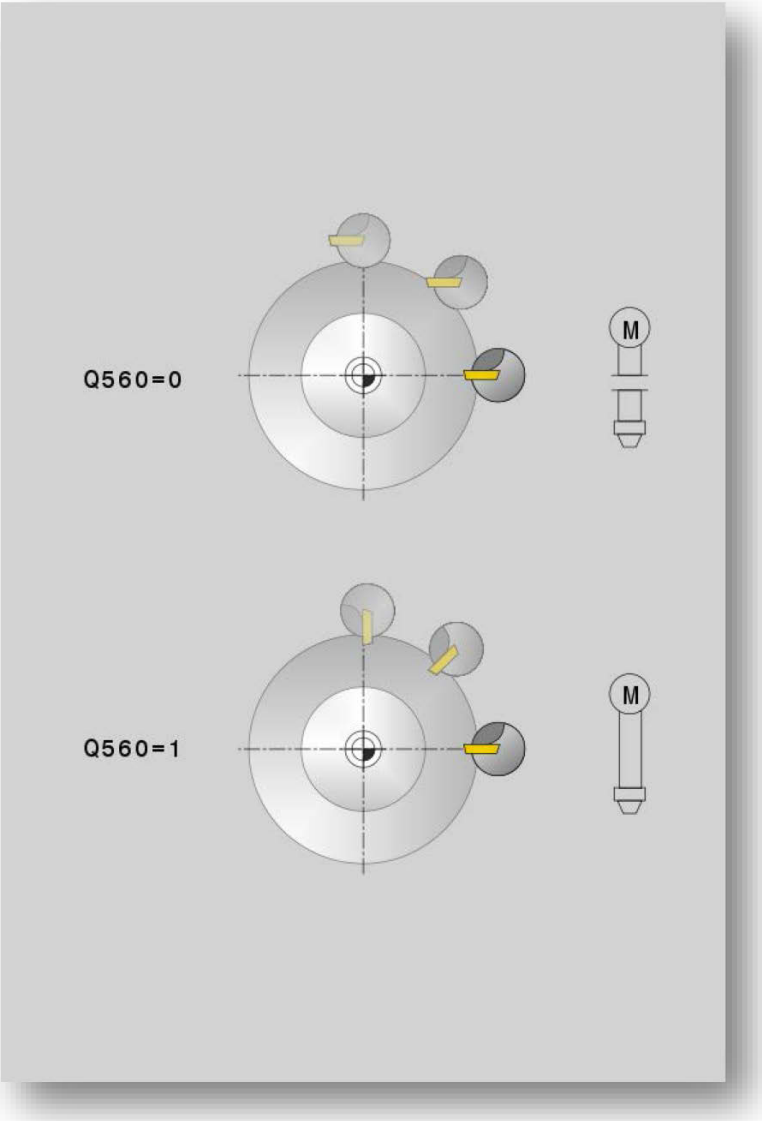
Programming – Cycle 291

Programming:

- You use **Q560** to specify whether the spindle should be coupled or not during machining

→ Interpolation turning **Q560 = 1**

→ Interpolation milling **Q560 = 0**

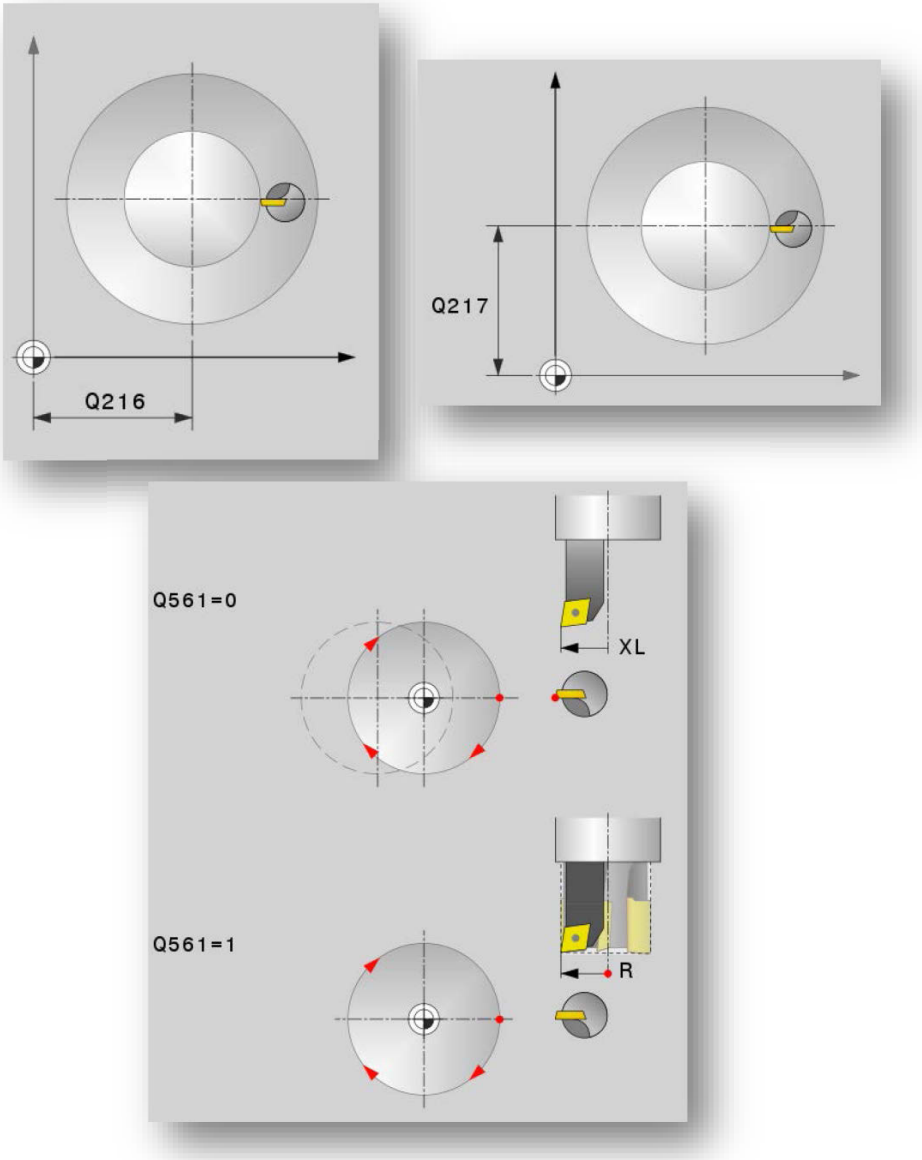


Programming – Cycle 291

Programming:

- With **Q216** (center of rotation in X) and **Q217** (center of rotation in Y), you define the center of rotation that the tool points to or away from.

- **Q561** supports you in the programming of the turning tool that was defined in toolturn.trn.
  - **Q561=0**: The value XL from toolturn.trn is used as XL.
  - → RR/RL can not be used
  - → Motion of the tool center point (TCP) must be programmed without spindle coupling
  - **Q561=1**: The value XL from toolturn.trn is interpreted as a radius R.
  - → RR/RL can be used
  - → Recommended variant



Programming – Cycle 291

Before machining:

- You must switch spindle coupling on before machining

```
19 CYCL DEF 7.0 DATUM SHIFT
20 CYCL DEF 7.1  X+0
21 CYCL DEF 7.2  Y+0
22 CYCL DEF 7.3  Z+0
23 PLANE SPATIAL SPA+0 SPB+0 SPC+0 TURN F9999
    SEQ- TABLE ROT
24 L  X+5.5645  Y+35.6082 FMAX
25 CYCL DEF 291 COUPLG.TURNG.INTERP.
    Q560=+1     ;SPINDLE COUPLING
    Q336=+0     ;ANGLE OF SPINDLE
    Q216=+0     ;CENTER IN 1ST AXIS
    Q217=+0     ;CENTER IN 2ND AXIS
    Q561=+1     ;CONVERT FROM TURNING TOOL
26 M99
27 L  Z+15 R0 FMAX
28 L  Z+1.4 FMAX
29 L  Z-0.6 FQ3
30 L FQ3
31 L  X+5.5552  Y+35.4841
32 L  X+5.5012  Y+35.123
33 L  X+5.408  Y+34.77
34 L  X+5.2768  Y+34.4293
35 L  X+5.1091  Y+34.105
```

After machining:

- You must switch spindle coupling off after machining

```
13945 L  X-3.9703  Y+41.4282
13946 L  X-3.9656  Y+41.9446
13947 L  X-3.957  Y+42.09
13948 L FQ4
13949 L  Z-21.5194
13950 L  Z-3.5194 FMAX
13951 L  Z+15 FMAX
13952 CYCL DEF 291 COUPLG.TURNG.INTERP.
    Q560=+0     ;SPINDLE COUPLING
    Q336=+0     ;ANGLE OF SPINDLE
    Q216=+0     ;CENTER IN 1ST AXIS
    Q217=+0     ;CENTER IN 2ND AXIS
    Q561=+1     ;CONVERT FROM TURNING TOOL
13953 M99
13954 CYCL DEF 32.0 TOLERANCE
13955 CYCL DEF 32.1
13956 CALL LBL 251 ;RESET TURN
13957 M30
13958 ;
```

HEIDENHAIN

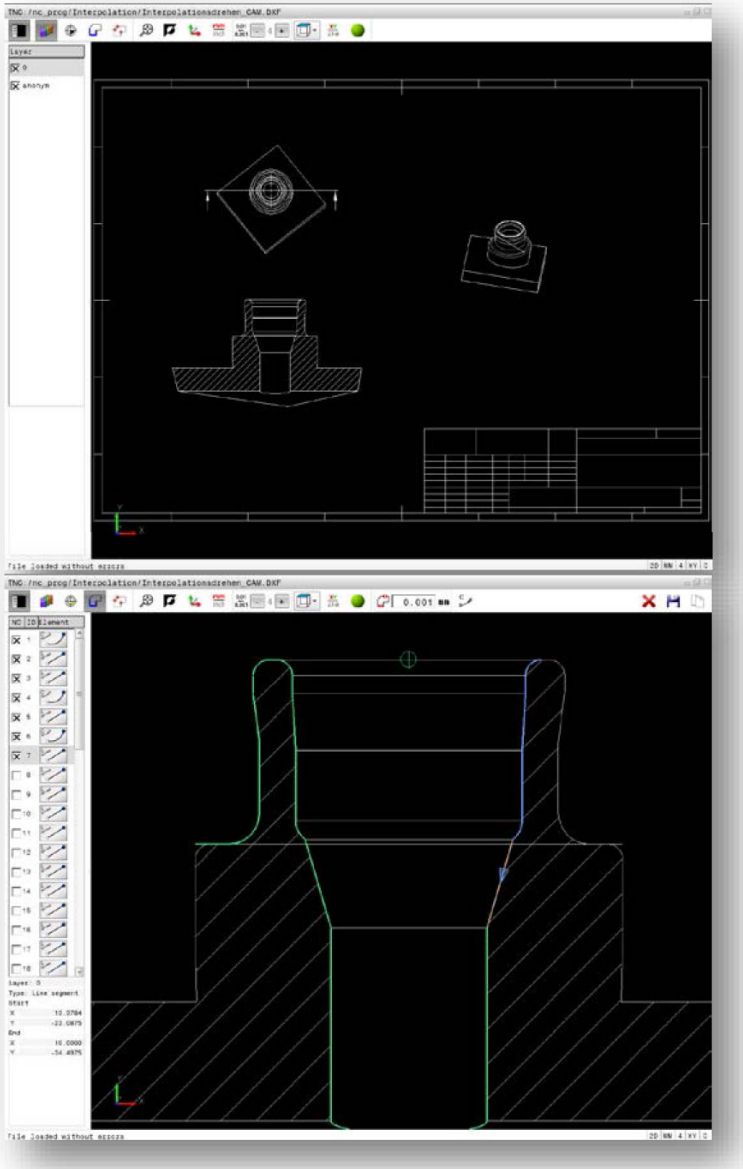
HEIDENHAIN

Tips and Tricks

Using a Contour from a DXF File

Contour creation:

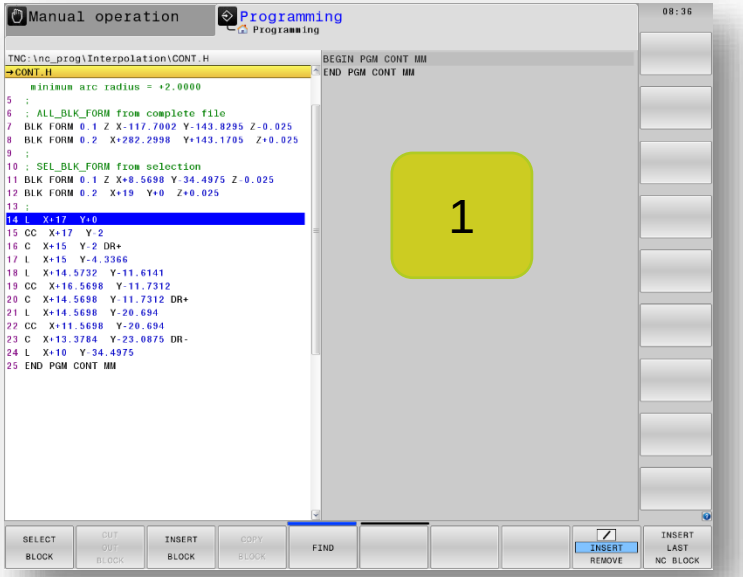
- DXF file with description of the workpiece contours
- Set the datum to the center of machining
- Now select the desired contour and save it

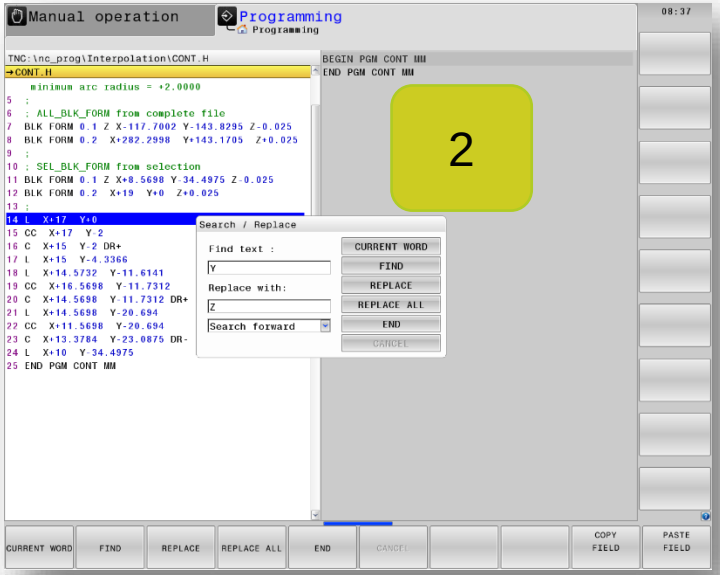


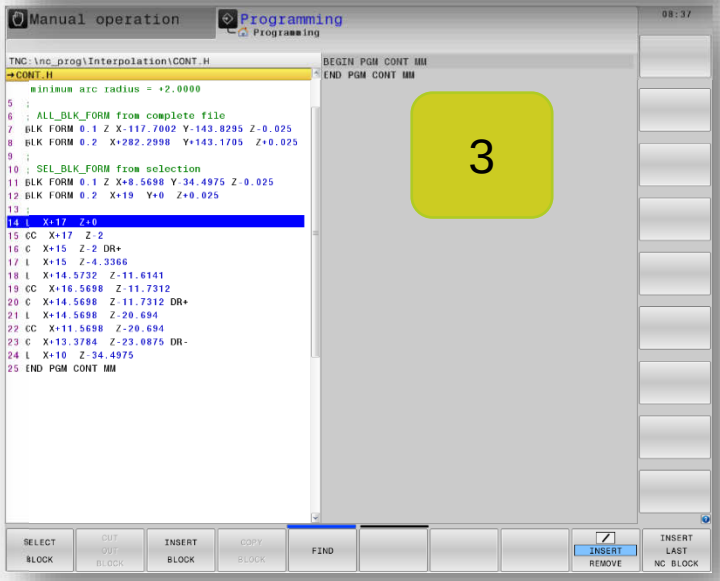
Using a Contour from a DXF File

Contour creation:

- In the machining program you use the FIND and the FIND and REPLACE functions to replace the Y coordinates with the Z coordinates







Increasing the Machining Accuracy

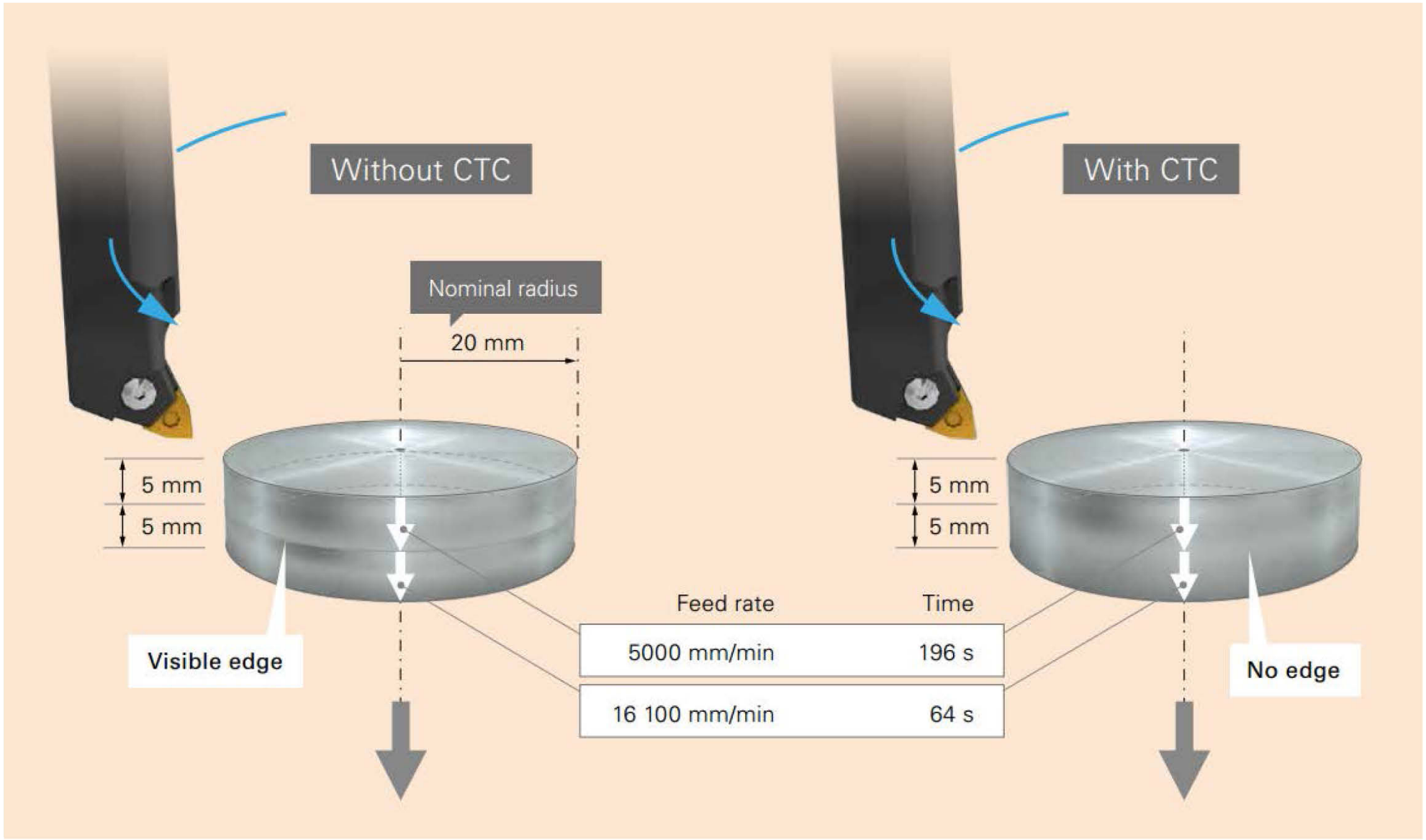
Interpolation turning in combination with Cross Talk Compensation (CTC):

- In order to reach the cutting speed, program very high feed rates for interpolation turning

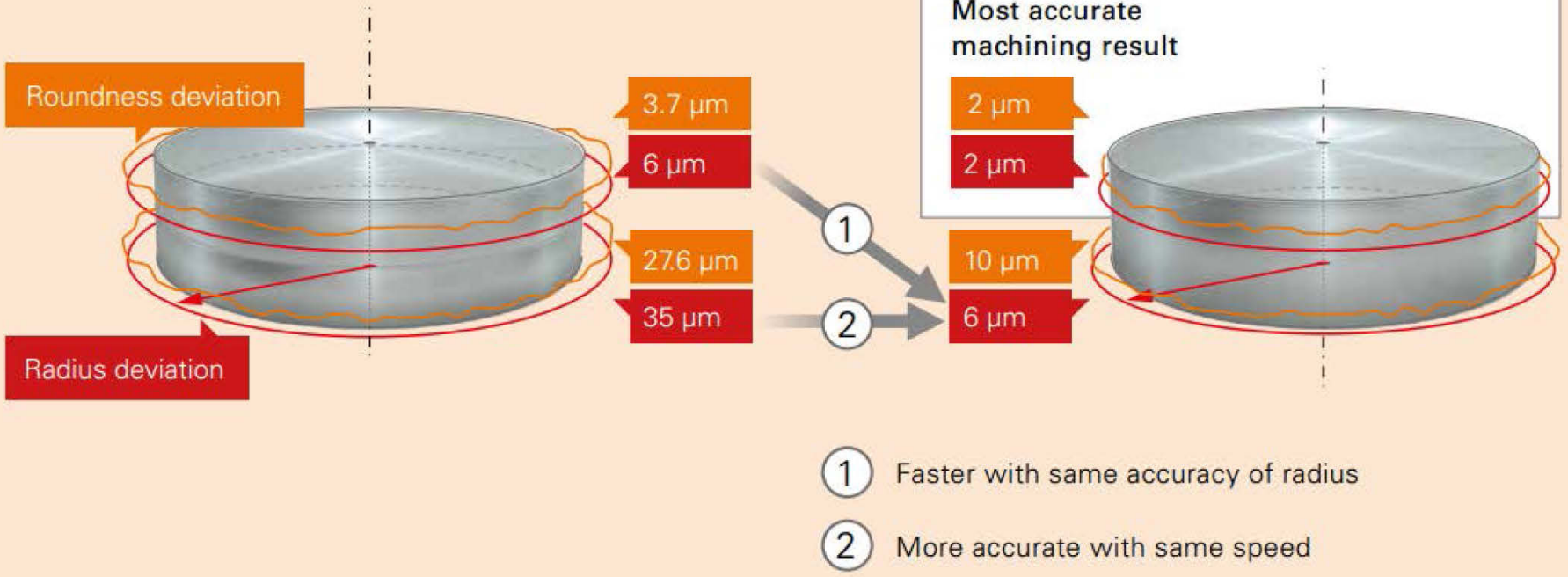
CTC OFF

CTC ON

Increasing the Machining Accuracy



Increasing the Machining Accuracy



Thank you very much for your attention!

Please do not hesitate to contact us should you have any questions:

Tel.: +49 8669 31-3103

E-mail: 3103@heidenhain.de

HEIDENHAIN

Thank you very much for your attention!

Michael Wiendl