3D Tool Setting Probes
BLUM Measuring Cycles

For Tool Setting Probes

Tool Measurement

Tool Length Measurement

Tool Radius Measurement

Tool Breakage Detection

Axes Compensation

Programming instructions	**English**
Software	146432 | 146456 | 146460
License	146435 | 146459 | 146463
Version	V3C
Control	Heidenhain TNC426/430 | iTNC530 | TNC620 | TNC640

Copyright

We always supply the latest software version. If another version is requested, please contact our Service Department / Hotline.

Prior to the starting-up it is mandatory to check if the parameters and the software range are already used by other programmes. If so, the cycles must be adapted because overwriting used parameters and data can have an unpredictable influence on the machine and could cause damage.
After finishing the installation, the used parameters and the performed changes to program and machine must be documented.
Blum-Novotest GmbH is not liable for damages to the machine due to programming errors or faulty application of the measuring cycles. By installing the measuring cycles, the specified warranty exclusions are acknowledged.
We reserve the right to make technical modifications which will improve the product. All suggestions for improvement are gladly accepted.
The language version of the manufacturer (DE/EN) is decisive for the technical content.
Original operating instructions

Please read the manual carefully first, then start up the measuring system and the measuring cycles!

Order number Software until 2018-03: P05.8000/-030.330
Order number Software as from 2018-04: 146432

Order number Software until 2018-03: P05.8000/-030.410
Order number Software as from 2018-04: 146456

Order number Software until 2018-03: P05.8000/-030.450
Order number Software as from 2018-04: 146460

Contents

1. Introduction

1.1 Explanation of warnings

CAUTION!
indicates hazards which may cause injuries unless avoided.

NOTICE
signifies measures to avoid material damage.

1.2 Notes

Control:	Heidenhain TNC426/430 | iTNC530 | TNC620 | TNC640
From software version:	TNC 426/430 280476-04
	iTNC 530: 340420-01, 340480-01, 340490-01, 606420-01
	TNC 620: 340560-03, 734980-01, 817600-01
	TNC 640: 340590-01

Control-specific characteristics of the TNC 620/640 are marked in green.

1.3 Legend

Additional advice

Further applicable documents, related literature
Please also observe the documentation of the machine manufacturer.

1.4 Further applicable documents

Programming instructions

Control-specific wiring diagrams
These are available at Blum-Novotest.

1.5 Abbreviations

MCS	Machine coordinate system

1.6 Intended use

The BLUM measuring cycles can be used with the following BLUM measuring systems:

- Z-MT / ZX-Speed / ZX-Speed IR / ZX-Speed RC / TC54-20 / TC64-20

2. Introduction

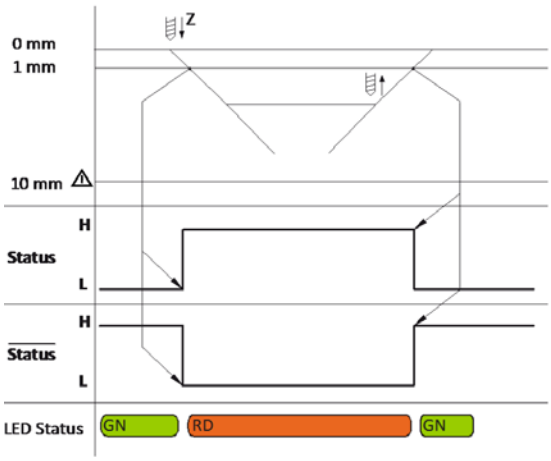
The tool setting probe is a tactile high-precision measuring system for automated measurement of the tool geometry within the working area of a machining centre working under normal operating conditions. This helps to avoid further damage to the workpiece or to the following tool.
Advanced possibilities for tool setting, breakage detection and contour monitoring of rotating tools are available with the non-contacting BLUM laser measuring system. Further information can be obtained from our technical service department.

2.1 Operating mode

The tool setting probe is a high-precision switch and is built into a rigid body together with the corresponding electronic parts. When the measuring system is deflected, the status of the probe changes at a certain defined position.

Blum offers two kinds of technologies. The cable bound as well as the infrared or radio tool setting probes. Both systems are designed for operating voltage 24V DC. The output signals have the same voltage level. The cable bound systems are ready for operation immediately after the operating voltage is switched on. Infrared and radio systems have to be switched on and off before and after measurement. The following output signals are available:

- The **Output STATUS** issues the following signal levels depending on the probe deflection:
  0 V with not actuated tool setting probe (probe system not deflected)
  24 V with actuated tool setting probe (probe system deflected)
- The **Output STATUS INVERSE** issues the following signal levels depending on the probe deflection:
  24 V with the tool setting probe not operated (probe system not deflected)
  0 V with actuated tool setting probe (probe system deflected)



Notes concerning measuring sequence

NOTICE

Risk of collision

▶ Limit measuring block overtravel within 2 mm

☞ Pay attention to maximum feed rate of the machine manufacturer

☞ Calibration and tool measurement at the same constant feed rate and rotation speed

For further information concerning the corresponding tool setting probe, please refer to the operating instructions.

2.2 Program overview

The following files are saved on the PLC partition of the control:

PLC:\BLUM\ Z_PROBE_0.TAB	Parameter table for system-specific definitions
PLC:\OEMCY3\NC\ ..\OEMCYC\TREE3\..\NC\..	Directory path for cycle files
561.CYC	Cycle 561 - Calibration and temperature compensation
562.CYC	Cycle 562 – Tool length measurement and breakage detection
563.CYC	Cycle 563 – Tool length measurement and radius measurement, breakage detection in case of Z-3D probe systems only
578.H	Auxiliary program called by the cycle at the beginning (functions to be executed are defined by the machine manufacturer)
579.H	Auxiliary program called at the cycle end (performed functions are defined by the machine manufacturer!)
MCODE.H	Auxiliary program for entering and executing "non-variable" M functions especially for TNC426/430 280476 xx controls (performed functions are defined by the machine manufacturer!)
RS232.H	Auxiliary program for executing RS232 functions
Z_CLEAN.H	Auxiliary program for cleaning measuring surface + tool
Z3D_POS.H	Auxiliary program for finding the probe element axis (plate centre) after manual pre-positioning in cycle calibration 561 where Q352=1 in case of Z-3D probe systems only

The following files are saved on the TNC partition of the control:

TNC:\BLUM\Z-PROBE\.. TNC:\BLUM\Z-PROBE\..	Directory path for sample programs
BLUM561.H	Example program for probe system calibration using cycle 561
BLUM562.H	Example program for length measurement using cycle 562
BLUM563.H	Example program for length and radius measurement with cycle 563 in case of Z-3D probe systems only
INFO.A	Information on Blum software version
T_RS232.H (not in case of TNC 620/640)	Program for testing the RS232 actuation IF59-A2
S_SWITCH.H (not with TNC 620/640)	Program for accepting the current soft-end switches from MP910.x and MP920.x

The sample programs show which parameters have to be defined when calling the measuring cycles and how the measuring cycles can be called from a machining program.

After installation, the following BLUM measuring cycles are available on the control. The cycles are selected by the following keys and inserted into a machining program.

- Key "TOUCH PROBE" (on the operating panel)
- Soft key symbol "BLUM Z-3D CYCLES" (1st soft key level, 2nd soft key line)

- Soft key symbols with cycle numbers "561...563" (2nd soft key level, 1st soft key line)

2.3 General information

2.3.1 General

- In the file TNC:\BLUM\Z-PROBE\ INFO.A, the BLUM software version installed on the control can be read out.
- The program examples in directory TNC:\BLUM\Z-PROBE\BLUM56*.H show how the measuring cycles are called from a machining program.
- The tool measurement is executed by the measuring block TCH PROBE 3.0 MEASURING, referring to the tool holder reference point in the machine coordinate system (REFERENCE SYSTEM:1), regardless of whether cycle 19 or M128 is active (precondition is iTNC530 higher than version 340 422 05). In older iTNC530 versions, an error message is output when 3D-ROT is active.
- All travel movements (except the measuring block) can be influenced with the override potentiometer.
- If the cycle is interrupted due to error messages, the values already measured correctly will not be entered in the tool table.
- Calibration and measurement must always be performed when the machine is at operating temperature. A heating-up phase at a mean spindle speed, with NC axes moved and perhaps with coolant switched on is recommended. This ensures that the same environmental conditions for the measuring system are always in place.
- To achieve high in-process security and reliability, repeat measurements will generally be performed in case of an error. In conjunction with a reasonable evaluation strategy a high measurement precision is achieved.
- At the start of the cycle the last programmed spindle rpm S and the spindle status (M3, M4, M5) are buffered by the cycle. At the end of the cycle the previous spindle status is restored independently of the determined tool status (tool in working order / tool defective).
- The centric/eccentric measuring position for the tool length measurement is determined by parameter ROFFS in the tool table. If ROFFS = 0 or ROFFS < P46 (radius of the measuring surface), the tool is positioned in the centre. Pre-positioning to the safety clearance P10 is used as the measuring block from retract position P80 on. This detects an incorrectly entered tool length. This method can be used for tool identification.
- Before the cycle is executed, a TOOL CALL command must always be executed in specifying the tool number, tool axis and spindle speed (spindle indexing angle), otherwise incorrect tool data might be read/written in the cycle.

```
...
TOOL CALL 2 Z S3000 FMAX
TCH PROBE 562 TOOL LENGTH
...
```

- The measuring surface and/or the tool is cleaned with a customer-specific configurable cleaning strategy (modifiable auxiliary program Z_CLEAN.H) before the length measurement if a cleaning nozzle is available. If the tool length is unknown, a rough measuring block is always executed before running the cleaning strategy to determine the appropriate cleaning position for the tool at a short distance over the measuring surface. The cleaning strategy can be run with different intensity and time expenditure depending on the measuring modes Q350=0,1,2 (measurement, check, control).

2.3.2 Tool change / spindle status

The tool change procedure must always be performed before a measuring cycle is called. When starting TOOL CALL, the tool number T# and the tool axis must always be specified. The spindle rpm Sxx as the temporarily valid spindle indexing angle / spindle rpm for tool measurement is optional.

Example of the tool change call:

```
...
TOOL DEF 5                              TOLL DEF 5.1
TOOL CALL 5 Z FMAX                      TOOL CALL 5.1 Z FMAX
...
```

In the case of Z-3D probe systems, measurements are performed, depending on the radius R, the measuring position ROFFS as well as the radius of the measuring surface P46, either with a stationary spindle (0 <= S <= 360 and ROFFS < P46 or ROFFS >= P46) or with a rotating spindle (S > 360 and ROFFS >= P46).

ROFFS < P46 (probe element radius for L measurement):
When 0 =< S <= 360 and ROFFS < P46, the S value is interpreted as the indexing angle and the spindle positioned to the specified angle position with cycles 13 and M19. The tool length is measured centrically in upright orientation.
When S > 360 and ROFFS < P46, the S value is interpreted as the spindle rpm. The spindle is indexed to the value of parameter "spindle indexing angle" (P18 in PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR]). The tool length is measured centrically in upright orientation.

ROFFS >= P46 (probe element radius for L measurement):
When 0 =< S < 360 and ROFFS >= P46, the S value is interpreted as the indexing angle and the spindle positioned to the specified angle position with cycles 13 and M19. For ROFFS the tool length is measured eccentrically in vertical upright position with the longest tool cutting edge being determined with a detailed search at 3 different angle positions with reduced angle increments to determine the specified indexing angle. This kind of measurement is useful for tools with one cutting edge of which the angle position with regard to the probe system is known.
When S = 360 and ROFFS >= P46, the S value is interpreted as the indexing angle and the spindle positioned to the specified angle position with cycles 13 and M19. For ROFFS the tool length is measured eccentrically in vertical upright position with the longest tool cutting edge and the associated angle position with regard to the probe system are at first roughly determined by a full search (full search = measurement at several angle positions via a full spindle rotation). The exact length of the tool cutting edge is then determined by a detailed search at the angle position found (detailed search = measurement at three angle positions with half of the angle increment by the highest value found). This kind of measurement is useful for tools with several cutting edges for which the angle position of the longest tool cutting edge with regard to the probe system is unknown and should not be measured while rotating due to sensitive cutting edges.
When S > 360 and ROFFS >= P46, the S value is interpreted as the spindle rpm. The tool length will be measured with ROFFS rotating. This kind of measurement is useful for tools with several cutting edges for which the angle position of the longest tool cutting edge with regard to the probe system is unknown and a stationary measurement is not desired (standard case).

At the end of the measuring cycle the last active spindle status is restored.

2.3.3 Read / write tool data

When a measuring cycle is called, the current tool data for the tool in the spindle is read from the tool table TOOL.T. The tool status TL in the tool management system must be released to start the measuring cycle.

Tool length parameter in the tool management system:

The variable value L from the tool management is interpreted by the measuring cycle as the nominal value length L_{TAB}, and the variable value DL as a real measurable length wear value DL_{TAB}. The variable $DL_{TOOLCALL}$ from the last TOOL CALL block can be used to implement a fictitious length compensation (e.g. when machining with an allowance or when length compensation is required depending on the measured cutting edge geometry). The control calculates the total tool length for the machining program:

$L = L_{TAB} + DL_{TAB} + DL_{TOOL\ CALL}$

The measuring cycle calculates the real tool length:

$L = L_{TAB} + DL_{TAB}$ (without $DL_{TOOL\ CALL}$!)

With $L = 0$, $L < L_{min}$ or $L > L_{max}$ the tool length is assumed to be unknown. In this case a rough search process with the spindle at standstill is performed for the first measuring block. The start position of the measuring block is ($P90 + L_{max} + P10$), and the aim position is ($P90 + L_{min} – P12$) with regard to the measuring surface of the probe element. This ensures that both the longest and the shortest tool within this measuring path emits a trigger signal to determine the approximate length. All other scanning cycles for taking precise measurements are then run with reference to this roughly determined tool length.
With $L \geq L_{min}$ or $L \leq L_{max}$ the tool length is assumed to be known. In this case pre-positioning to the safety clearance P10 is run as the measuring block with the spindle at standstill. The start position of the measuring block is ($P90 + L_{max} + P10$), and the aim position is ($P90 + L – P10$) with regard to the measuring surface of the probe element. If the tool length is defined correctly, no trigger signal is expected within this measuring path. Only after this monitored pre-positioning was completed, the proper tool length measurement is executed.
An equal distribution of the total tool length into the 3 stated tool length parameters is certainly possible and does not affect the machining program, but is not useful when the Blum measuring cycles are used, as only L_{TAB} and DL_{TAB} are evaluated or determined in the measuring cycle. The procedure for evaluating and determining these tool length values and saving of the determined tool data in the tool management are hard-coded in the measuring cycle and cannot be modified.
The total length of the reference tool L+DL may be entered as 0 mm in the tool table (=zero tool) if $L_{min} < 0$ mm was entered in table Z_PROBE_*.TAB [*.ZPR]. The tool lengths are then determined relative to this zero length so that positive and negative tool lengths are possible.

NOTICE

Risk of collision possible due to incorrect manual input of the tool length L in the tool management system.
In the case of an incorrect input, a false pre-position or target position is calculated and approached.

- ▶ Enter the correct tool length value L, DL.
  Maximum deviation from the real length measurement: 1…5 mm
- ▶ Do not change a correctly measured or entered value.

Tool radius parameter in the tool management system:

The variable value R from the tool management system is interpreted by the measuring cycle as a nominal value radius RTAB, the variable value DR as a real, measurable radius wear value DRTAB. The variable DRTOOLCALL from the last TOOL CALL block can be used to implement a fictitious radius offset (e.g. when machining with an allowance or when a radius offset is required, depending on the measured cutting edge geometry). The control calculates the total tool radius for the machining program according to the formula:

R = RTAB + DRTAB + DRTOOL CALL

The measuring cycle calculates the real tool radius from:

R = RTAB + DRTAB (without DRTOOL CALL!)

When R = 0 or R > Rmax, the tool radius is assumed to be unknown. In this case a rough search process with the spindle at standstill is performed for the first measuring block. The start position of the measuring block is (P91 + P47 + Rmax + P10), and the aim position is (P91 + P47 + Rmin – P12) with regard to the probe element. This ensures that both the largest and the smallest tool within this measuring path emits a trigger signal to determine the approximate radius. All other scanning cycles for taking precise measurements are then run with reference to this roughly determined tool radius.

When R > 0 or R ≤ Rmax, the tool radius is assumed to be known. In this case pre-positioning to the safety clearance P10 is run as the measuring block with the spindle at standstill. The start position of the measuring block is (P91 + P47 + Rmax + P10), and the aim position is (P91 + P47 + R + P10) with regard to the probe element. If the tool radius is defined correctly, no trigger signal is expected within this measuring path. Only after this monitored pre-positioning was completed is the actual tool radius measurement performed.

An equal distribution of the total tool radius into the 3 stated tool radius parameters is certainly possible and does not affect the machining program, but is not useful when the Blum measuring cycles are used, as only RTAB and DRTAB are evaluated or determined in the measuring cycle. The procedure for evaluating and determining these tool radius values and saving the determined tool data in the tool management are hard-coded in the measuring cycle and cannot be modified.

NOTICE

Risk of collision possible due to incorrect manual input of the tool radius R in the tool management system.

In the case of an incorrect input, a false pre-position or target position is calculated and approached.

▶ Enter the correct tool radius value R or DR.
Maximum deviation from the real length radius value: 1…5 mm

▶ Do not change a correctly measured or entered value.

At the end of the cycle run the measuring result is automatically written back in tool table TOOL.T.

- If the measurement is error-free, the tool data (length L, radius R and correction values DL) are updated and the tool status TL is set to valid/free.
- If the measurement is incorrect, the tool data (length L, radius R and correction values DL, DR, tool status TL) is not updated. The error text display causes the cycle and the machining program to be aborted.

2.3.4 Switching between mm / inches

It is easy to switch the control from mm to inches. If the machining program to be called is defined in inches (e.g. using BEGIN PRG xxx INCH and END PRG xxx INCH), all route and position-dependent constants will automatically be interpreted as inch values. The call parameters may be specified in inches; in this case, the result parameters, such as the tool length and radius, will be automatically converted by the control and entered into the tool table in inches. The default values for the Q-parameters will be rounded up or down in inches during input.

NOTICE

All parameters in the PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] table must always be entered in "mm", even if the control is switched to "inch".

2.3.5 Calling the measuring cycles in DIN / ISO code

The measuring cycles can also be called easily using the DIN / ISO code from a machining program. The cycles are selected with the following keys and entered in a machining program:

- Key "TOUCH PROBE" (on the operating panel)
- Soft key symbol "BLUM Z-3D CYCLES" (1st soft key level, 2nd soft key line)

- Soft key symbols with cycle numbers "561...563" (2nd soft key level, 1st soft key line)

Example of calling the measuring cycle G562 in DIN / ISO Code:

```
...
T6 M6 S3000
G562 Q350=0 Q361=2 Q362=0,010
...
```

2.3.6 Program test operating mode and block search

In the program test operating mode or when the block search is active, the BLUM measuring cycles are skipped. If valid tool data must be available in the tool table for workpiece machining, the data must be manually entered or determined beforehand by a separately run measuring cycle.

3. Description of the measuring cycles

3.1 Cycle 561 - Calibration and temperature compensation

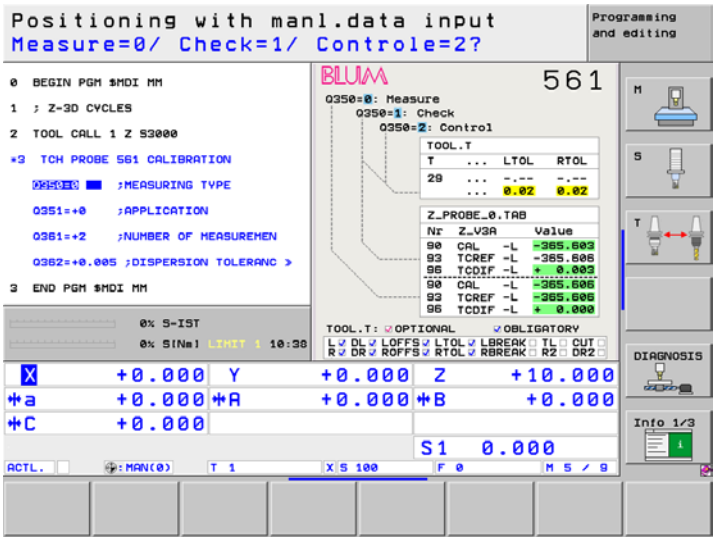
Cycle 561 is the program for calibration of the probe system and for calculation of the temperature drift in the available measuring axes with the aid of a reference tool.

3.1.1 Calibration of the probe system

Before the probe system can be used for measurement, the system must be calibrated. For this purpose, a wear-free calibration tool is used, e.g. a cylindrical pin (Ø > 4mm). The reference tool is clamped into a tool holder. The length of the reference tool must be determined precisely (µm resolution), either with a tool pre-setting device or in the machine with a dial gauge and then the data is stored in tool table TOOL.T. Based on these reference dimensions, the trigger point positions of the probe system are determined relative to the machine coordinate system. The measurement result is stored as the calibration value in line 90,91,92 and as the temperature reference value in line 93,94,95 in the table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR].

Example of calling the measuring cycle in the NC program:

```
...
TOOL CALL 1 Z S3000 FMAX
TCH PROBE 561 CALIBRATION
 Q350=0          ;MEASURING MODE
 Q351=0          ;MEASURING SCOPE
 Q361=3          ;NUMBER OF MEASUREMENTS
 Q362=0.01       ;PERM. SCATTERING TOLERANCE
 Q352=0          ;MANUALLY POSITIONED
...
```



The following Q parameters are used with the program call:

Q350	**Measuring mode**	
	Q350 = 0	Measuring: Storage of the new calibration value plus temperature reference value, no tolerance monitoring between the new and old calibration values
	Q350 = 1	Verification: Storage of the new calibration value and tolerance monitoring between the temperature reference value and the temperature shift value
	Q350 = 2	Check: no storage of the new calibration value; only tolerance monitoring between the temperature reference value and the temperature shift value
Q351	**Measuring scope**	
Q361	**Number of repeat measurements for determining the average**	
Q362	**Permitted scattering tolerance of the individual measured values for repeat measurement**	
Q352	**Manual pre-positioning**	
optional	*(Only necessary for initial commissioning or change of the mounting position of the probe system)*	

3.2 Temperature compensation of the NC axes

For increased accuracy requirements, the temperature drift of the machine can be compensated with the aid of the probe and the reference tool. The cycle uses the reference dimensions of the reference tool to determine the trigger point positions of the probe system changed by displacement. These are stored in the table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] as new calibration values and compared with the reference positions from the TC reference measurement (Q350=0). The difference determined is stored as the temperature shift value both in the Q parameter and in the table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] in line 96,97,98 and can be offset as an additive zero point offset. The machining program is then continued with the corrected values.

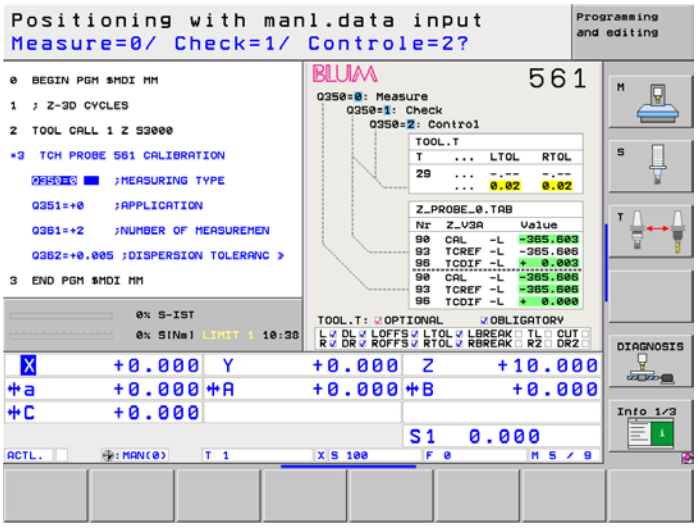
Notes:

The temperature compensation should be integrated into the machining process according to the following pattern:

1. Calibration of the probe system with cycle 561 (Q350=0 → TC reference measurement).
2. Measurement of all used tools.
3. Test machining of the first workpiece, until all workpiece dimensions are in compliance with the rated dimensions.
4. Depending on the machine's temperature drift and required accuracy, TC comparative measurement can be performed before or after any desired number of machining tasks.

Example of calling the measuring cycle in the NC program:

```
...
TOOL CALL 1 Z S3000        FMAX
TCH PROBE 561 CALIBRATION
 Q350=1            ; MEASURING MODE
 Q351=0            ;MEASURING SCOPE
 Q361=3            ;NUMBER OF MEASUREMENTS
 Q362=0.01         ;PERM. SCATTERING TOLERANCE
 Q351=0            ;MANUALLY POSITIONED
. . .
```



The following Q parameters are used with the program call:

Q350	Measuring mode: Reference measurement or comparative measurement	
	Q350 = 0	Measuring: Reference measurement with updating of the calibration value
	Q350 = 1	Verification: Comparative measurement with updating of the calibration value
	Q350 = 2	Check: Comparative measurement without updating of the calibration value
Q351	**Measuring scope**	
Q361	**Number of repeat measurements for determining the average**	
Q362	**Permitted scattering tolerance of the individual measured values for repeat measurement**	
Q352 optional	**Manual pre-positioning** Default value: Q352 = 0	

The following parameters must be defined in the tool table TOOL.T:

T	L	DL	R	DR	R2	DR2	TL	CUT	ROFFS	LOFFS	LTOL	RTOL	LBREAK	RBREAK
1	+80.5	+0.0	+6.0	+0.0	xxx	xxx		xxx	0.0	0.0	0.03	0.03	xxx	xxx

The following parameters must be defined in the tool table TOOL.T:

L=xxx.xxx	**Reference tool length**
R=xxx.xxx	**Reference tool radius**
DL=xx.xxx	**Tool compensation length**
DR=x.xxx	**Tool compensation radius**
LOFFS	**Axial offset for radius measurement position**
ROFFS	**Radial offset for length measurement position**
LTOL	**Max. permitted deviation in the length axis (for comparative measurement)** If the machine's thermal increase is greater than the permissible tolerance in this axis, and error message is output. The offset value (nominal-actual difference) is entered in table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 96. From here the value can be read and offset as additive zero point offset depending on the evaluation strategy. Value range: 0.0000 mm ≤ LTOL ≤ 0.9999 mm Default value: LTOL = 0.020 mm ... 0.050 mm
RTOL	**Max. permitted displacement in the radius axis/secondary axis (for comparative measurement)** If the machine's thermal increase is greater than the permissible tolerance in this axis, and error message is output. The offset value (nominal-actual difference) is entered in table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 97,98. From here the value can be read and offset as additive zero point offset depending on the evaluation strategy. Value range: 0.0000 mm ≤ RTOL ≤ 0.9999 mm Default value: RTOL = 0.020 mm ... 0.050 mm

Explanation:

The reference tool with T# =1 is changed into the spindle before the cycle 561 is called.
The tool is pre-positioned automatically (Q352=0) and collision-free via the retraction level in the centred start position above the measuring surface of the probe system. The trigger point positions of the probe system are calculated as the average from two repeated measurements (Q361) and the scatter of the single measuring values is monitored for compliance with the scattering tolerance of 0.01 mm (Q362). The trigger point positions measured are stored as the calibration value in line 90,91,92 and the temperature reference value (Q350=0) in line 93,94,95 or the temperature shift value (Q350=1) in line 96,97,98 in table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR]. This calibration data must not be deleted or changed.
At the end of the measuring cycle the retraction level is approached in rapid traverse.

The following Q parameters are determined or updated in the course of TC measurement:

Q185	Offset value in X axis
Q186	Offset value in Y axis
Q187	Offset value in the Z axis

Calculation: Offset value (line 96,97,98) = set value (line 90,91,92) – actual value (line 93,94,95)

Example of the calculation of the offset values in the following NC program:

(unless the function has already been implemented in the PLC by the machine manufacturer!)

```
...
TCH PROBE 561 CALIBRATION
 Q350=1       ; MEASURING MODE
 Q351=0       ; MEASURING SCOPE
 Q361=3       ; NUMBER OF MEASUREMENTS
 Q362=0.01    ; PERM. SCATTERING TOLERANCE
 Q352=0       ; MANUALLY POSITIONED
CYCL DEF 7.0 ZERO OFFSET; POSITIVE SIGN FOR WORKPIECE CORRECTION
CYCL DEF 7.1 IX+Q185 ; ADDITIVE ZERO POINT OFFSET IN AXIS X
CYCL DEF 7.2 IY+Q186 ; ADDITIVE ZERO POINT OFFSET IN AXIS Y
CYCL DEF 7.3 IZ+Q187 ; ADDITIVE ZERO POINT OFFSET IN AXIS Z
...
```

The following parameters must be defined in the tool table TOOL.T:

T	L		DL	R	DR	R2	DR2	TL	CUT	ROFFS	LOFFS	LTOL	RTOL	LBREAK	RBREAK
1	+80.5		+0.0	+6.0	+0.0	xxx	xxx		xxx	0.0	0.0	0.03	0.03	xxx	Xxx

Explanation:

The reference tool with T# =1 is changed into the spindle before the cycle 561 is called.
The tool is pre-positioned automatically (Q352=0) and collision-free via the retraction level in the centred start position above the measuring surface of the probe system. The trigger point positions of the probe system are calculated as the average from two repeated measurements (Q361) and the scatter of the single measuring values is monitored for compliance with the scattering tolerance of 0.01 mm (Q362). The trigger point positions measured are stored as the calibration value in line 90,91,92 and the temperature reference value (Q350=0) in line 93,94,95 in table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR]. This calibration data must not be deleted or changed. During later tool measurement, the calibration value is used for positioning of the tool relative to the probe element and for calculation of the tool data.
At the end of the measuring cycle the retraction level is approached in rapid traverse.

Attention:

- Before calibration/measurement the reference tool and the measuring surface on the probe system should be cleaned carefully at the test point using a lint free cloth.
- Calibration and tool measurement must be performed with the machine at operating temperature (switch-on time at least 15 min with the spindle rotating).

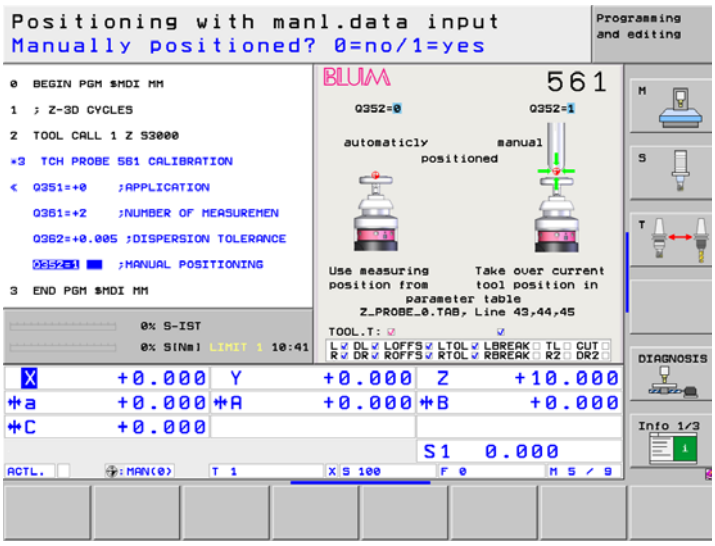
If the calibration parameters have been entered incorrectly, the following errors may occur:

- If dimensions of the reference tool are incorrectly specified in the tool table, the measuring results for the tool measurement are incorrect.
- If the calibration is missing or has not been run, incorrect or undefined measuring positions are approached during the subsequent tool measurement.

Initial commissioning or position change of the probe system:

At initial commissioning or after a change of the mounting position of the probe system, the tool tip of the reference tool must be prepositioned manually approx. 1mm in the centre above the measuring surface. Calibration cycle 561 must be executed with call parameter **Q352=1** to achieve that the manually preset measuring position is adopted by the cycle in all of the 3 axes and stored in table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 43,44,45. If parameter P7=1 is set, Z-3D probe systems also measure the centre of the probe surface by means of multiple approaches from different directions and the positions determined in this way entered in line 43,44,45.

```
...
TOOL CALL 1 Z S3000 FMAX
TCH PROBE 561 CALIBRATION
 Q350=0          ;MEASURING MODE
 Q351=0          ;MEASURING SCOPE
 Q361=3          ;NUMBER OF MEASUREMENTS
 Q362=0.01       ;PERM. SCATTERING TOLERANCE
 Q352=1          ;MANUALLY POSITIONED
...
```



3.3 Cycle 562 - Tool length measurement and breakage detection

Cycle 562 is the program for tool length measurement, wear measurement and breakage detection using the probe system. The cycle can measure known tools (tool length L≠0) or unknown tools (tool length L=0). The measurement result is written back in tool table TOOL.T under the current tool T#.

Please note:
For unknown tool data (L = 0 mm, R = 0 mm) there is always a risk of collision with the measuring system. Therefore, it is usually recommended to input rough tool geometry data L and R in tool table TOOL.T (input accuracy / maximum deviation < ±5mm).

The following Q parameters are used with the program call:

Q350	Measuring mode: see chapter 3.5.1
Q361	**Number of repeat measurements for determining the average**
Q362	**Permitted scattering tolerance of the individual measured values for repeat measurement**

The following parameters must be defined in the tool table TOOL.T:

L, DL	**Tool length, tool length compensation**	**(compulsory when Q350 = 1/ 2)**
R, DR	**Tool radius, tool radius compensation**	**(optional)**
ROFFS	**Radial measuring position for length measurement**	
LTOL	**Wear tolerance: Length**	**(only with Q350 = 1/ 2)**
LBREAK	**Breakage tolerance: Length**	**(only at Q350 = 2)**

Measuring sequence:
The tool must be loaded into the spindle before cycle 562 is called.
The tool is pre-positioned automatically and collision-free via the retraction level in the start position above the measuring surface of the probe system. The tool length is measured centrically or eccentrically with a multiple probing cycle as a function of the radial measuring position TT:ROFFS entered in tool table TOOL.T:

The tool is measured **centrically** with **the spindle at standstill**, i.e. the tool axis is positioned centrically towards the measuring surface if the following condition is true:

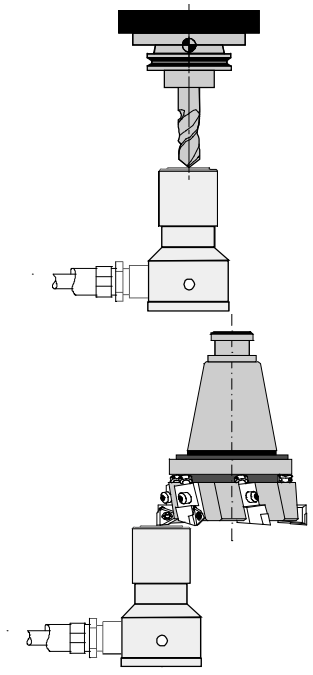
TT:ROFFS <= $R_{measuring\ surface}$ (=P46) or

TT:ROFFS = R and **R <= $R_{measuring\ surface}$**

The tool is measured **acentrically** with **orientated spindle**, i.e. the tool axis is positioned eccentrically by ROFFS towards the measuring surface if the following condition is true:

TT:ROFFS > $R_{measuring\ surface}$ (=P46)

Note:
For L=0 the first measuring block is centrically executed as a search run with spindle indexing S=0°. Then eccentric positioning by ROFFS is run and the spindle indexed again as a function of value $R_{measuring\ surface}$/ROFFS for each further measuring block until the tool has been rotated once by 360° and all cutting edges were measured on the measuring surface at least once. The longest tool cutting edge and the associated angle increment are stored in the machine.
Again 3 measurements with half of the angle increment are run about this longest tool cutting edge. The maximum tool length value determined in the process is entered in TOOL.T.



With **Q350=0 (measurement)** , the measured tool length is written in memory L and tool compensation DL=0 is set.
With **Q350=1 (check)**, the nominal tool length in memory L is not changed. The deviation between the currently measured actual tool length and nominal tool length is calculated and written in memory DL. Then the tool compensation DL is checked for compliance with the wear tolerance LTOL or LBREAK and the tool status TL is set.
With **Q350=2 (control)**, the tool length in memory L, DL is not changed. The deviation between currently measured actual tool length and nominal tool length is calculated and checked for compliance with the wear tolerance LBREAK and the tool status TL is set.
When exceeding the wear tolerance LTOL or LBREAK or the scattering tolerance Q362, the measurement is repeated (depending on parameter "number of repeated measurements" in table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 5, 6). Thus, it is possible to avoid to the greatest possible extent, unique sources of error or sources of error by chance, e.g. chips adhering to the tool or deposited on the measuring surface, which could be responsible for faulty results. The measured value is determined again under minimally changed environmental conditions, e.g. cleaning the tool or the measuring surface with compressed air, and the result is either corrected or confirmed.

⇨ In case of faultless tool, the error marker Q199=0 is set. The machining of the parts can be continued without interruption.

⇨ In case of incorrect tool, the error marker Q199=1 or Q199=2 is set and the tool is locked in tool table TOOL.T. All further actions, e.g. machine stop or loading of a sister tool must be decided in the main program.

At the end of the measuring cycle the retraction level is approached in rapid traverse.

Following parameters in TOOL.T are set during the program flow:

L, DL	**Tool length, tool length compensation**	**(compulsory when Q350 = 0/ 1)**
TL	**Tool lock**	**(compulsory when Q350 = 1/ 2)**

The following Q parameters are set during the program sequence:

Q199	**Tool status parameter for tool breakage or tool wear**	
	Q199 = 0	Tool / cutting-edge OK, continue work piece machining
	Q199 = 1	Tool dimensions exceed simple wear tolerance LTOL < DL < (2 * LTOL) or LTOL < DL < LBREAK
	Q199 = 2	Tool dimensions beyond the double wear tolerance, cutting-edge broken DL > (2 * LTOL) or DL > LBREAK

3.3.1 Tool Length Measurement

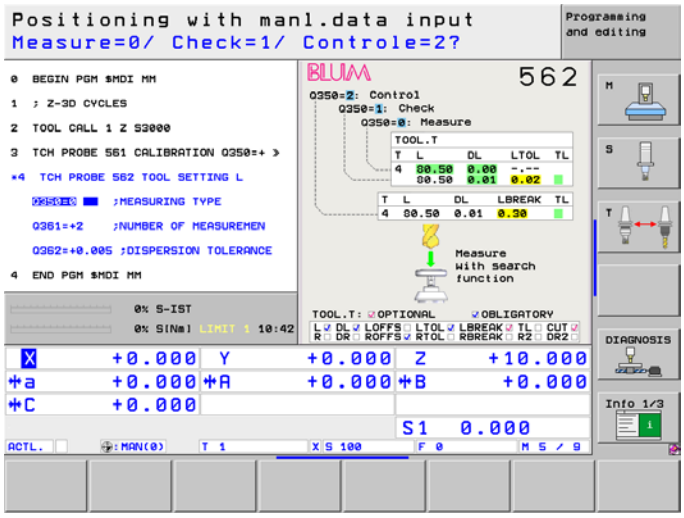
The tool length can be precisely measured by the probe system. When a cycle is called, a rough tool length L should be entered in tool table TOOL.T (accuracy: < ± 5mm).

Example of parameters in TOOL.T:

T	L	DL	R	DR	R2	DR2	TL	CUT	ROFFS	LOFFS	LTOL	RTOL	LBREAK	RBREAK
5	+120.0	+0.0	xxx	xxx	xxx	xxx		6	+8.5	xxx	+0.0	xxx	+0.0	xxx

Example of the program call:

```
...
TOOL CALL 1 Z S3000        FMAX
TCH PROBE 562 TOOL LENGTH
 Q350=0              ; MEASURING MODE
 Q361=3              ;NUMBER OF MEASUREMENTS
 Q362=0.01           ;PERM. SCATTERING TOLERANCE
...
```



Parameter **Q350=0 (measurement)** is used in cycle 562 to initially measure the length of the tool. The roughly entered tool length L is read from the tool table and used for pre-positioning. At the end of the measurement the measurement tool length L is written back in tool table TOOL.T and the associated wear value DL=0 and the tool status Q199=0 are set.

Attention:

- The tool must be loaded into the spindle and be active before the measurement is run.
- The rough tool length L+DL should be entered in tool table TOOL.T. Wear tolerance LTOL=0 or breakage tolerance LBREAK=0 need not be set.
- For initial tool measurement call parameter Q350=0 must be set.
- For tools with a large diameter parameter TT:ROFFS ≥ 0 must be entered in TOOL.T. If ROFFS = 0 or ROFFS < $R_{measuring\ surface}$ positioning is centrally, and for ROFFS > $R_{measuring\ surface}$ the measurement is taken eccentrically. (ROFFS R is not permitted!).
- If a defined cutting edge length should be measured on the walls with a large diameter, the spindle indexing with S = 0 … 360 must be entered in the TOOL CALL block.

3.3.2 Tool wear measurement

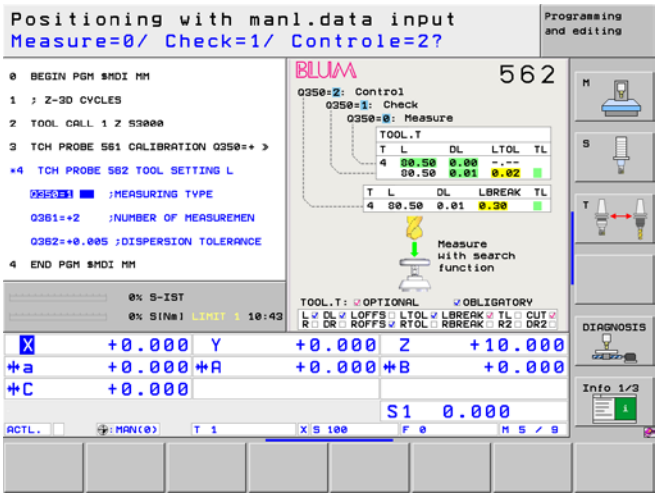
To ensure faultless part production, critical tools must be monitored for tool wear before or after each machining process. When a cycle is called, the exact reference tool length L and a permissible wear tolerance LTOL should be entered in tool table TOOL.T. It is optionally possible to define the breakage tolerance LBREAK.

Example of parameters in TOOL.T:

T	L	DL	R	DR	R2	DR2	TL	CUT	ROFFS	LOFFS	LTOL	RTOL	LBREAK	RBREAK
5	+120.2	+0.0	xxx	xxx	xxx	xxx		6	+8.5	xxx	+0.02	xxx	+0.5	xxx

Example of the program call:

```
...
TOOL CALL 1 Z S3000        FMAX
TCH PROBE 562 TOOL LENGTH
 Q350=1          ; MEASURING MODE
 Q361=3          ;NUMBER OF MEASUREMENTS
 Q362=0.01       ;PERM. SCATTERING TOLERANCE
...
```



Parameter **Q350=1 (check)** is used in cycle 562 to check the tool for wear. Reference tool length L in TOOL.T is not changed. The current tool length is measured, compared with the nominal specification L from the tool table and the difference is stored as the wear value in memory DL. If the tolerance LTOL or LBREAK is exceeded, the tool status TL or Q199 is set:

- Q199 = 2, if DL > (2*LTOL) or DL > LBREAK — TOOL.T: TL=L
- Q199 = 1, if LTOL < DL < (2*LTOL) or LTOL < DL < LBREAK — TOOL.T: TL=L
- Q199 = 0, if DL<LTOL

If the parameter "program stop at Q199>0" is set, the machining program is stopped when the tool is blocked and the error message "TOLERANCE EXCEEDED" is output.

Attention:

- The tool must be loaded into the spindle and be active before the measurement is run.
- Tool length L must be entered in tool table TOOL.T. Wear tolerance LTOL > 0 must be set, optionally also breakage tolerance LBREAK > 0.
- For tool wear measurement call parameter Q350 = 1 must be set.
- For tools with a large diameter parameter TT:ROFFS ≥ 0 must be entered in TOOL.T. If ROFFS = 0 or ROFFS < $R_{measuring\ surface}$ positioning is centrally, and for ROFFS > $R_{measuring\ surface}$ the measurement is taken eccentrically. (ROFFS =R is not permitted!).
- If a defined cutting edge length should be measured on the walls with a large diameter, the spindle indexing with S = 0 … 360 must be entered in the TOOL CALL block.

3.3.3 Tool breakage detection

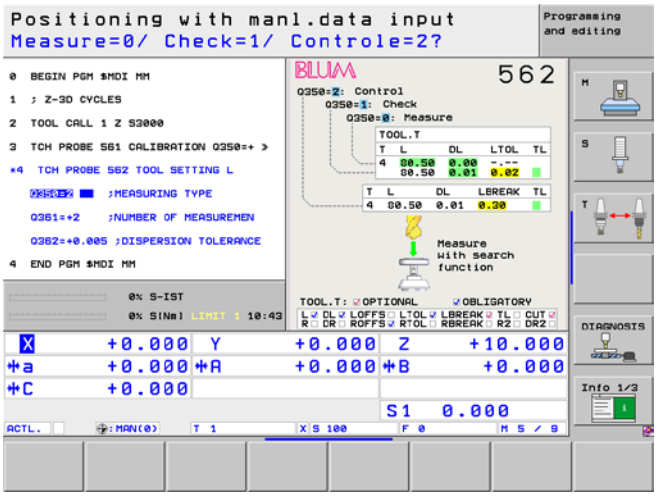
To ensure faultless part production, critical tools must be monitored for tool breakage or wear after machining. When a cycle is called, the exact nominal tool length L and a permissible breakage tolerance LBREAK must be entered in tool table TOOL.T.

Example of parameters in TOOL.T:

T	L	DL	R	DR	R2	DR2	TL	CUT	ROFFS	LOFFS	LTOL	RTOL	LBREAK	RBREAK
5	+80.5	+0.0	xxx	Xxx	xxx	Xxx		0	+0.0	xxx	+0.02	xxx	+0.5	xxx

Example of the program call:

```
...
TOOL CALL 1 Z S3000        FMAX
TCH PROBE 562 TOOL LENGTH
 Q350=2              ; MEASURING MODE
 Q361=3              ; NUMBER OF MEASUREMENTS
 Q362=0.01           ;PERM. SCATTERING TOLERANCE
...
```



Parameter **Q350=2 (control)** is used in cycle 562 to check the tool for tool shank breakage. The current tool length is measured, compared with nominal specification L from the tool table and the difference checked for compliance with the length tolerances. The tool data L and DL in TOOL.T is not updated. If breakage tolerance LBREAK is exceeded, the tool status TL or Q199 is set:

- Q199 = 2, if DL ≥ LBREAK TOOL.T: TL = L
- Q199 = 0, if DL < LBREAK

If the parameter "program stop at Q199>0" is set, the machining program is stopped when the tool was blocked and the error message "BREAKAGE TOLERANCE EXCEEDED" is output.

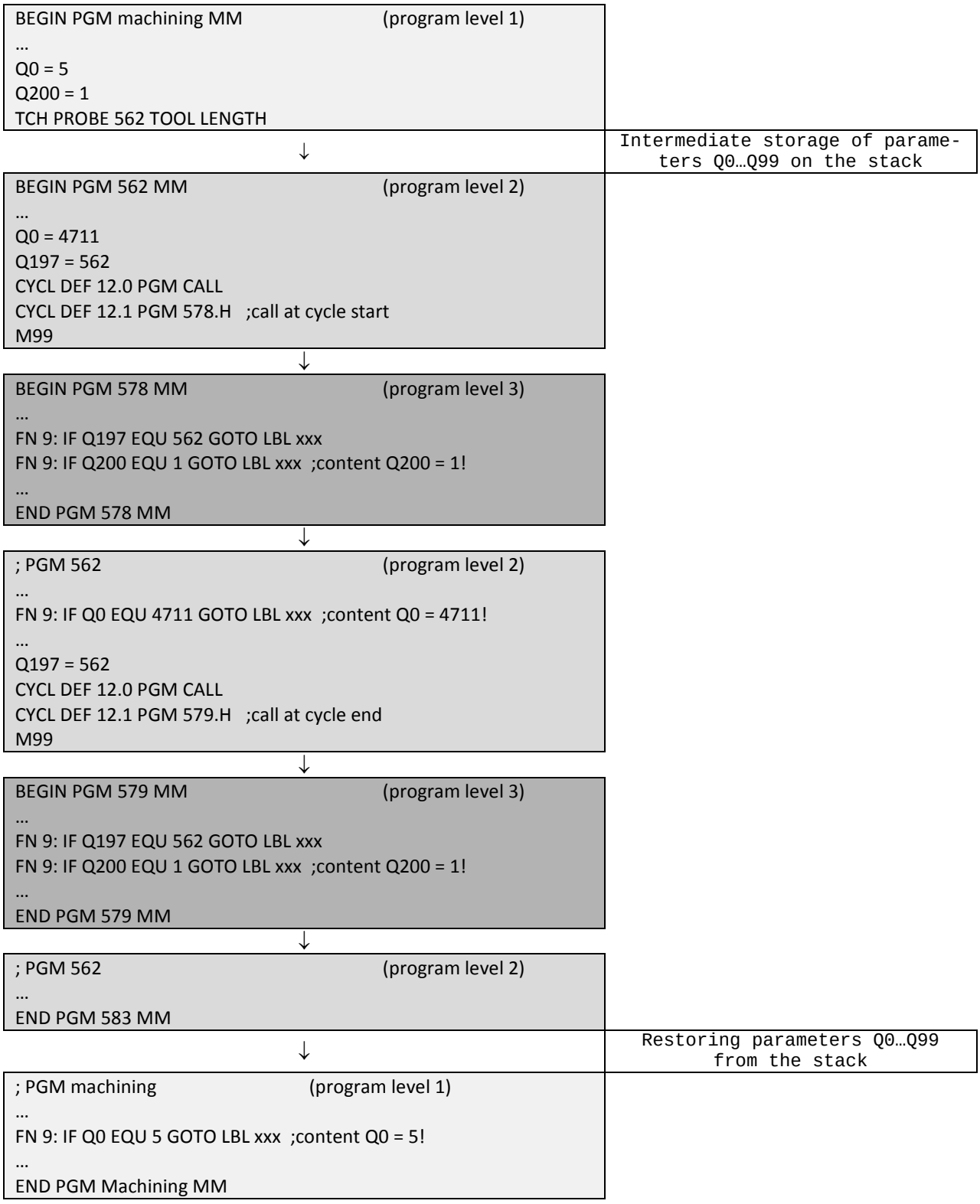
Attention:

- The tool must be loaded into the spindle and be active before the measurement is run.
- Tool length L must be entered in tool table TOOL.T. Breakage tolerance LBREAK > 0 must be set.
- For tool breakage detection call parameter Q350 = 2 must be set.
- For tools with a large diameter parameter TT:ROFFS ≥ 0 must be entered in TOOL.T. If ROFFS = 0 or ROFFS < $R_{measuring\ surface}$ positioning is centrally, and for ROFFS > $R_{measuring\ surface}$ the measurement is taken eccentrically. (ROFFS = R is not permitted!).
- If a defined cutting edge length should be checked for tools with a large diameter, the spindle indexing must be entered in the TOOL CALL block with S = 0 … 360 .

3.4 Program levels and program nesting

Passing Q parameter values from the machining program (program level 1) to the cycle (program level 2) or to the auxiliary program 578.H and/or 579.H (program level 3) or from 578.H to 579.H is only possible through global Q parameters Q200 ... Q399, not through the local Q parameter Q0 ... Q99.

Example for transfer of Q parameter values:



3.5 Parameter overview

When starting a measuring cycle, parameter table PLC:\BLUM\ Z_PROBE_*.TAB or Z_PROBE_*.ZPR (for TNC 620/640) is read and the system-specific data written into the Q parameters Q200...Q299. Here, they are available to be read by the user. During commissioning, the parameters were assigned with valid values by the machine manufacturer. The parameters Q0...Q99 are internal local Q parameters and only valid in the measuring cycles. The external global Q parameters with the same Q numbers are not changed or overwritten by the measuring cycles (precondition MP7251=0).

3.5.1 Description of parameters in the cycle call

Reserved Q-parameters for special TNC functions:

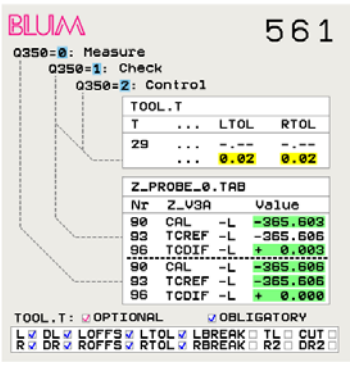
Q para	Definition / Meaning
Q150… Q169	Auxiliary parameters in PGM Z3D_POS.H
Q185	**Temperature compensation value / offset value in the X-axis**
Q186	**Temperature compensation value / offset value in the Y-axis**
Q187	**Temperature compensation value / offset value in the Z-axis**
Q190	**Current error number** When displaying a BLUM error text, parameter Q190 can be used to give a more precise error diagnostic. The value before the decimal point identifies the indicated error text number, the three-digit value after the decimal point specifies the exact cause of the error. Example: Q190 = 16,061 16.xxx Error text: "Error parameter table" xx.061 Error at parameter P61 in line 61 Value range: 0 ≤ Q190 ≤ 20 [24] Note: If you have any queries or would like an exact error analysis please contact BLUM Service.
Q197	**Current cycle number** Transfer of current BLUM cycle number to the auxiliary programs 578.H and 579.H Value range: 561 ≤ Q197 ≤ 563
Q199	**Tool status parameter for tool breakage, tool wear and cutting-edge monitoring** (set by the BLUM cycle) Q199 = +0 Tool dimensions within the permitted tolerance LTOL / RTOL, LBREAK / RBREAK Tool released (TL = no entry) Q199 = +1 Tool dimensions outside of the allowed tolerance LTOL / RTOL Tool locked (TL = tool locked) Q199 = +2 Tool dimensions outside of the allowed tolerance LBREAK / RBREAK Tool locked (TL = tool locked)
Q200 … Q299	Parameter P0-P99 from parameter table Z_PROBE_*.TAB or Z_PROBE_*.ZPR for TNC 620/640 in directory PLC:\BLUM\..

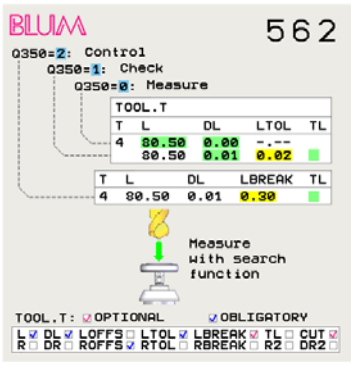
Global Q-parameter:

Q para	Definition / Meaning
Q348	**Set-tool length** For post-processor applications the path movement of the tool is frequently calculated for a pre-set nominal tool length. As a minimum path correction, only the deviation from this nominal length can be determined and entered in the tool table. If the tools are inserted according to this procedure, the predefined nominal length must be transferred to the BLUM cycle for each cycle call in parameter Q348. In connection with: $L_{Real} = L_{TOOL.T} + DL_{TOOL.T} + Q264 + Q266 * Q348$ with Q264 = additive tool length compensation (line 64 in Z_PROBE_*.TAB [*.ZPR]) with Q266 * Q348 = nominal tool length from post-processor application (parameter 66) Value range: -999.999 mm ≤ Q348 ≤ +999.999 mm Default value: Q348 = +0.000 mm **(Input in mm, no inch values allowed!)**
Q349	**Nominal tool radius** For post-processor applications the path movement of the tool is frequently calculated for a pre-set nominal tool radius. As a minimum path correction, only the deviation from this nominal radius can be determined and entered in the tool table. If the tools are inserted according to this procedure, the predefined nominal radius must be transferred to the BLUM cycle for each cycle call in parameter Q349. In connection with: $R_{Real} = R_{TOOL.T} + DR_{TOOL.T} + Q265 + Q267 * Q349$ with Q265 = additive tool radius compensation (line 65 in Z_PROBE_*.TAB [*.ZPR]) with Q267 * Q349 = target tool radius from post-processor application (parameter 67) Value range: -999.999 mm ≤ Q349 ≤ +999.999 mm Default value: Q348 = +0.000 mm **(Input in mm, no inch values allowed!)**

Call parameter

Q para	Definition / Meaning
Q350	**Measuring mode**





Specification whether the tool is to be measured again or whether an already measured tool is to be checked for wear or breakage.

When **Q350=0 (measurement)**, the cycle calculates the real actual for the length / radius, overwrites the value L / R in the tool memory TOOL.T and sets the tool compensation DL=0 / DR=0.
When **Q350=1 (verify)**, the measured actual value for the length / radius is compared with the specified set value L / R from the tool memory TOOL.T. The cycle calculates the deviation with the correct algebraic sign and enters it as tool compensation DL / DR in TOOL.T. If the difference value is greater than the permitted wear tolerance LTOL / RTOL or LBREAK, the tool is locked (status TL / RBREAK in TOOL.T)
When **Q350=2 (check)**, the measured actual value for the length / radius is compared with the specified set value L / R from the tool memory TOOL.T. If the difference value is greater than the permitted wear tolerance LTOL / RTOL or LBREAK / RBREAK, the tool is locked (status TL in TOOL.T). The tool data L / R or DL / DR in TOOL.T is not updated!

Attention:

The measuring mode Q350=1 / 2 (check / control) is automatically reset to Q350=0 (measurement) if condition $L_{min} < (L+DL) < L_{max}$ or Rmin < (R+DR) < Rmaxis not fulfilled.

Value range: +0 ≤ Q350 ≤ +2
Default value: Q350 = +0

Overview of the measuring modes for cycle 561, 561, 563

Measuring mode	Designation	Approach and measuring strategy	Tool data
0	Measurement	Search function to determine the tool data	• Length / radius may be unknown • Length L / radius is updated • Wear value DL/DR is set to 0 • Tool is **not** evaluated Q199 = 0
1	Verification	Search function to determine the tool data	• Length / radius must be measured • Length L / radius is **not** changed • Wear value DL/DR is updated • Tool is evaluated Q199 = 0/1/2
2	Check	Search function to determine the tool data	• Length must be measured • Length L is **not** changed • Wear values DL/DR is **not** changed • Tool is evaluated Q199 = 0/1/2

Q para	Definition / Meaning
Q351	**Measuring scope**

Definition which tool geometry is to be measured (part or complete measurement).

Q351 = +0	Tool length and radius measurement
Q351 = +1	Length measurement only
Q351 = +2	Radius measurement only (condition: LMIN < (L + DL) < LMAX)
Value range:	+0 ≤ Q351 ≤ +2
Default value:	Q351 = +0

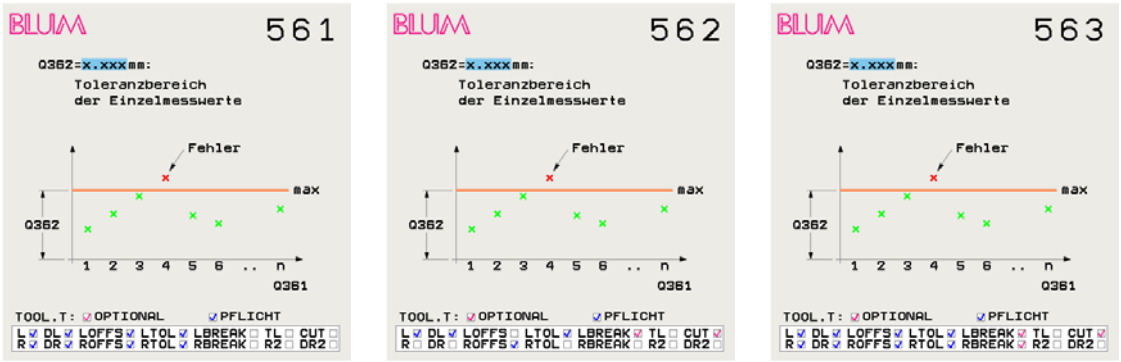
Q352 **Manual pre-positioning**

Q352 = +0	The measuring position for calibration of the probe system is read from parameter table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 43, 44, 45. The reference tool is automatically pre-positioned on the centre of the measuring surface.
Q352 = +1	The reference tool was manually pre-positioned on the centre of the probe element, the tool tip must not contact the measuring surface. The current tool position is adopted for parameter table PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 43, 44, 45 when the calibration cycle starts. If parameter P7=1 is set, Z-3D probe systems also measure the centre of the probe surface by means of multiple approaches from different directions and the positions determined in this way entered in line 43, 44, 45.
Default value:	Q352 = +0

Q361 **Number of repeat measurements for determining the average**

Value range	1 ≤ Q361 ≤ 10
Default value:	Q361 = 2

Q362 **Permitted scattering tolerance of the individual measured values for repeat measurement**



For the specified repeated measurement (Q361>1) the difference between the maximum and minimum measuring value is determined and compared with the scattering tolerance. If the scattering tolerance is exceeded, an error message is output and the cycle is aborted.

Value range	+0.001 mm ≤ Q362 ≤ +0.100 mm
Default value:	Q362 = 0.010 mm

3.5.2 Description of parameters in TOOL.T

Para	Description
L	**Tool length** If Q350 = 0, the tool length can be set with L = 0 mm (unknown tool length). If Q350 = 1/ 2, the tool length L must be set with a valid value (known tool length). Value range: -999.999 mm ≤ L ≤ +999.999 mm
DL	**Tool length compensation** The parameters L + DL must/should be set with valid values (known tool length). Value range: -99.999 mm ≤ DL ≤ +99.999 mm
R	**Tool radius** The parameters R + DR must be set with valid values (known tool radius). When Q350 = 0, the tool radius can be set with R = 0 mm (unknown tool radius). When Q350 = 1/ 2, the tool radius R must be set with a valid value (known tool radius). Value range: -999.999 mm ≤ R ≤ +999.999 mm
DR	**Tool radius compensation** The parameters R + DR must be set with valid values (known tool radius). Value range: -99.999 mm ≤ DR ≤ +99.999 mm
CUT	**Number of tool cutting edges** Value range: +0 ≤ CUT ≤ +20
LTOL	**Wear tolerance: Length** With Q350 = 1, the tool wear is calculated as the difference between the measured length value and the specified set value and entered in parameter DL. If the length tolerance LTOL is exceeded, Q199=1 is set and the tool is locked in tool memory TOOL.T. If LTOL = 0, the wear tolerance value, universally valid for all tools in PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 56 is used automatically. Value range: +0.000 mm ≤ LTOL ≤ +0.9999 mm Default value: LTOL = +0.030 mm
RTOL	**Wear tolerance: Radius** When Q350 = 1, the tool wear is calculated as the difference between the measured radius value and the specified set value and entered in parameter DR. If the radius tolerance RTOL is exceeded, Q199=1 is set and the tool is locked in tool memory TOOL.T. If RTOL=0, the wear tolerance value, universally valid for all tools in PLC:\BLUM\ Z_PROBE_*.TAB [*.ZPR] line 57 is used automatically. Value range: +0.000 mm ≤ RTOL ≤ +0.9999 mm Default value: RTOL = +0.030 mm
LOFFS	**Axial measuring position for radius measurement** For tools such as ball nose mills, torus milling cutters, etc., the tool radius must not be measured at the height of the tool tip, but at the highest point of the corner radius. The position for the radius measurement is the relative, axial distance between the measuring position on the tool cutting edge and the tool tip. Value range: +0.000 mm ≤ LOFFS ≤ +999 mm Default value: LOFFS ≈ R2
ROFFS	**Radial measuring position for length measurement** In the case of acentric tools, such as end mills, facing mills, etc., the length must not be measured in the tool axis, but off-centre on the tool cutting edges. The position for length measurement is the relative radial distance from the measuring position on the tool cutting edge up to the tool axis. Value range: +0.000 mm ≤ ROFFS ≤ +999 mm (character 'R' not allowed!) Default value: ROFFS ≈ R – R2

LBREAK	**Breakage tolerance: Length** With Q350 = 2 the tool length is measured and compared with the preset value from the tool memory. If the breakage tolerance LBREAK is exceeded, Q199 = 2 is set and the tool locked in tool table TOOL.T. If LBREAK = 0, the breakage tolerance value, universally valid for all tools, in PLC:\BLUM\Z_PROBE_*.TAB [*.ZPR] line 58 is used automatically. Value range: +0.000 mm ≤ LBREAK ≤ +0.9999 mm Default value: LBREAK = +0.500 mm...+0.9999 mm
RBREAK	**Breakage tolerance: Radius** When Q350 = 2, the tool radius is measured and compared with the preset value from the tool memory. If the breakage tolerance RBREAK is exceeded, Q199 = 2 is set and the tool locked in tool table TOOL.T. If RBREAK = 0, the breakage tolerance value, universally valid for all tools, in PLC:\BLUM\Z_PROBE_*.TAB [*.ZPR] line 59 is used automatically. Value range: +0.000 mm ≤ RBREAK ≤ +0.9999 mm Default value: RBREAK = +0.500 mm...+0.9999 mm

4. Process and error messages

4.1 Process messages

The Heidenhain control provides the option of displaying running program operations as process messages in a pop-up window on the screen.

4.2 Error messages

If an error occurs that cannot be rectified, the detected error status is displayed as an error message on the screen. The cycle stops in an endless loop and the program must be restarted.
In some cases, reference in the error description is made to the parameters P0-P99 in parameter table Z_PROBE_*.TAB or Z_PROBE_*.ZPR for TNC 620/640 in directory PLC:\BLUM\.., where the *=0-7 table number may vary.
When displaying one of the following BLUM error texts, parameter Q190 can be used to render a more precise error diagnosis. The value before the decimal point identifies the indicated error text number, the three-digit value after the decimal point specifies the exact cause of the error.

Example: Q190=16.061

16.xxx → Error text: 'Error parameter table'
xx.061 → Error in parameter P61 in line 61

Legend:

1	= Error message		= Check the following points
	= Error description		= Remedy

E0	**Probe system not ready**
	The infrared/remote probe system failed to be activated.
	→ Check electrical connection → Check probe system type and measuring input P20 in Z_PROBE_*.TAB → Check selection mode probe system P21 in Z_PROBE_*.TAB → Check activation mode probe system P23 in Z_PROBE_*.TAB → Check identification TC/DUO on probe system IC5x/IF59 → Check alignment of the infrared receiver with the probe system → Check battery capacity

E1	**Probe system deflected**
	When the measurement process is started, the probe system is not in the rest position but deflected already.
	→ Check rest position of the measuring surface → Check ease of the deflection movement of the measuring surface → Check chip protection for contamination → Check battery capacity

E2	**Battery low**
	The battery capacity is no longer sufficient for precise and reliable measurement.
	→ Check battery capacity
	→ Replace the battery in the probe system as soon as possible → Maintain programming TC/DUO in the probe system

E3	Reserved
	Not used

E4	Incorrect approach side value
	The approach side radius axis and/or tool axis parameters in table PLC:\BLUM\ Z_PROBE_*.TAB are faulty.
	Parameter approach side length axis P50=-1/+1 Parameter approach side radius axis P51=-1/+1 Parameter approach side minor axis P52=-1/+1 P50,P51,P52=0 is not permitted

E5	Reserved
	Not used

E6	Measuring block without trigger point
	The tool could not be measured as the measuring surface had not been reached and deflected.
	→ Check tool geometry data L,R in TOOL.T → Check measuring positions ROFFS,LOFFS in TOOL.T → Check the following parameters in PLC:\BLUM\ Z_PROBE_0.TAB: → Check measuring positions P43,P44,P45 → Check calibration values P90,P91,P92 → Check parameters min/max tool length P60,P61 → Check parameters min/max tool radius P62,P63
	→ Repeat measurement with changed parameters

E7	No tool loaded T#=0
	The measuring cycle was started without a tool in the spindle. **Attention – risk of collision!**
	→ Load valid tool in spindle via TOOL CALL # → Repeat measurement

E8	Tool dimension exceeded
	The entered/measured tool geometry L,R in TOOL.T exceeds the permissible limit values in table PLC:\BLUM\ Z_PROBE_*.TAB.
	→ Check parameters min/max tool length P60,P61 in Z_PROBE_*.TAB → Check parameters min/max tool radius P62,P63 in Z_PROBE_*.TAB

E9	Measured value scatter too big
	The permissible scattering tolerance Q362 of the single measuring values for Q361 repeated measurements for average calculation is exceeded.
	→ Clean tool and measuring surface from dirt and chips → Increase scattering tolerance Q362 if necessary → Repeat measurement

E10	Tolerance value exceeded
	The wear tolerance LTOL,RTOL or breakage tolerance LBREAK,RBREAK in TOOL.T is exceeded. The tool is faulty and locked for further machining.
	→ Check tolerances LTOL,RTOL and/or LBREAK,RBREAK in TOOL.T → Check tool wear or tool breakage → Remove contamination from the tool and the measuring surface → Unlock tool in TOOL.T and repeat measurement

E11	Attention – risk of collision
	With the defined call parameters or tool data there is a risk of collision with the measuring system.
	→ Check tool parameters L,R,ROFFS,LOFFS in TOOL.T → Check radius measuring surfaces P46,P47,P48 in Z_PROBE_*.TAB
	→ Check tool parameters L,R,ROFFS,LOFFS in TOOL.T → Check radius measuring surfaces P46,P47,P48 in Z_PROBE_*.TAB → Repeat measurement with changed parameters → Soft limit switch L: SE L- <(P43+P50*(P60-P12)) < SE L+ → Soft limit switch L: SE L- <(P43+P50*(P61+P10))< SE L+ → Soft limit switch R: SE R- <(P44+P51*(P62+P47-P12)) < SE R+ → Soft limit switch R: SE R- <(P44+P51*(P63+P47+P10))< SE R+ → Soft limit switch N: SE N- <(P45+P52*(P62+P48-P12)) < SE N+ → Soft limit switch N: SE N- <(P45+P52*(P63+P48+P10))< SE N+

E12	Incorrect tool axis
	The current tool axis (X,Y,Z) from the last TOOL CALL is not matching with axis number tool axis P40 in Z_PROBE_*.TAB.
	→ Define tool axis with TOOL CALL command → Check axis number tool axis P40 in Z_PROBE_*.TAB

E13	Reserved
	not used

E14	Incorrect tool parameters
	A measuring parameter (LTOL, RTOL, LOFFS, ROFFS, CUT LBREAK, RBREAK, TL, etc.) in the tool table TOOL.T has been assigned an invalid value.
	→ Correct measuring parameters → Repeat measurement with changed parameters

E15	Incorrect call parameter
	One or more call parameter (Q350...Q3xx) are set with a critical or invalid value.
	→ Check all call parameters for valid values.
	→ Correct call parameters 0<=Q350<=2, 0<=Q351<=3, 0<=Q352<=1 1<=Q361<=9, 0.0001<=Q362<=0.9999 → - Repeat measurement with changed parameters

E16	Error parameter table
	Faulty parameter in PLC:\BLUM\ Z_PROBE_*.TAB line Pxx = (Q190-16)*1000
	Check the following parameters in PLC:\BLUM\ Z_PROBE_*.TAB:

Software version	P0
Program stop for tool breakage	0 ≤ P4 ≤ 2
Safety distance	5 ≤ P10 ≤ 50
Measuring surface overtravel	1 ≤ P12 ≤ 10
Retraction at standstill after probing	3 ≤ P13 ≤ 50
Retraction rotating after probing	0.1 ≤ P14 ≤ 2
Spindle indexing	-1 ≤ P18 ≤ 360
M-code spindle indexing	-1 ≤ P19 ≤ 999
Probe system type (cable,IR,X12,X13)	0 ≤ P20 ≤ 3
Positioning feedrate	P30 ≥ P31
Measuring speed	P32 ≥ P33
Measuring processes	0 ≤ P34 ≤ 2
Measuring surface radius	3 ≤ P46, P47, P48 ≤ 25
Measuring position calibration value L	P43-2 < P90 < P43+2
Measuring position calibration value R	P44-2 < P91 < P44+2
Soft limit switch L	SE L- <(P43+P50*(P60-P12)) < SE L+
Soft limit switch L	SE L- <(P43+P50*(P61+P10)) < SE L+
Soft limit switch R	SE R- <(P44+P51*(P62+P47-P12)) < SE R+
Soft limit switch R	SE R- <(P44+P51*(P63+P47+P10)) < SE R+
Soft limit switch N	SE N- <(P45+P52*(P62+P48-P12)) < SE N+
Soft limit switch N	SE N- <(P45+P52*(P63+P48+P10)) < SE N+

E17 / E18	Reserved
	Not used

E19	Error trigger point position
	The difference between the current axis position and the trigger point position signalled back by the control after a measuring block is greater than 20 mm. **Attention – risk of collision!**
	→ To delete the control error, the control must be completely switched off and on again. → Make a note of the TNC software version and inform the machine manufacturer. → Do not repeat measurement

E20	Reserved
	Not used

E21	SQL READ ERROR Z_PROBE_*.ZPR
	Read access on parameter file Z_PROBE_*.ZPR is not possible.
	→ Check the following file for existence and formatting: → Check parameter file PLC:\BLUM\ Z_PROBE_*.ZPR → Check configuration file ..\CONFIG\ Z_PROBE.CFG → Close parameter file Z_PROBE_*.ZPR in the text editor → Check parameter P1='TableSelection' in Z_PROBE_*.ZPR

E22	SQL WRITE ERROR Z_PROBE_*.ZPR
	Write access on parameter file Z_PROBE_*.ZPR is not possible.
	→ Check the following file for existence and formatting: → Check parameter file PLC:\BLUM\ Z_PROBE_*.ZPR → Check configuration file ..\CONFIG\ Z_PROBE.CFG → Close parameter file Z_PROBE_*.ZPR in the text editor → Check parameter P1='TableSelection' in Z_PROBE_*.ZPR

E23	SQL READ ERROR TOOL.T
	Read access on tool table TOOL.T is not possible.
	→ Close tool table TOOL.T in the text editor → Check the definitions of the active tool number in TOOL.T → The following parameters must be available in tool table TOOL.T: → L,R,DL,DR,ROFFS,LOFFS,LTOL,RTOL,LBREAK,RBREAK,CUT,TL,DIRECT

E24	SQL WRITE ERROR TOOL.T
	Write access on tool table TOOL.T is not possible.
	→ Close tool table TOOL.T in the text editor → Check the definitions of the active tool number in TOOL.T → The following parameters must be available in tool table TOOL.T: → L,R,DL,DR,ROFFS,LOFFS,LTOL,RTOL,LBREAK,RBREAK,CUT,TL,DIRECT

5. Service

BLUM
focus on productivity

Headquarters:

Blum-Novotest GmbH
Kaufstr. 14
88287 Gruenkraut, Germany
Tel.:+49 751 6008-0
Fax:+49 751 6008-156
www.blum-novotest.com
Vertrieb/Sales: sales@blum-novotest.com
Service: service@blum-novotest.com

Hotline:
Vertrieb/Sales: Tel.:+49 751 6008-200
Service: Tel.:+49 751 6008-202

Worldwide: