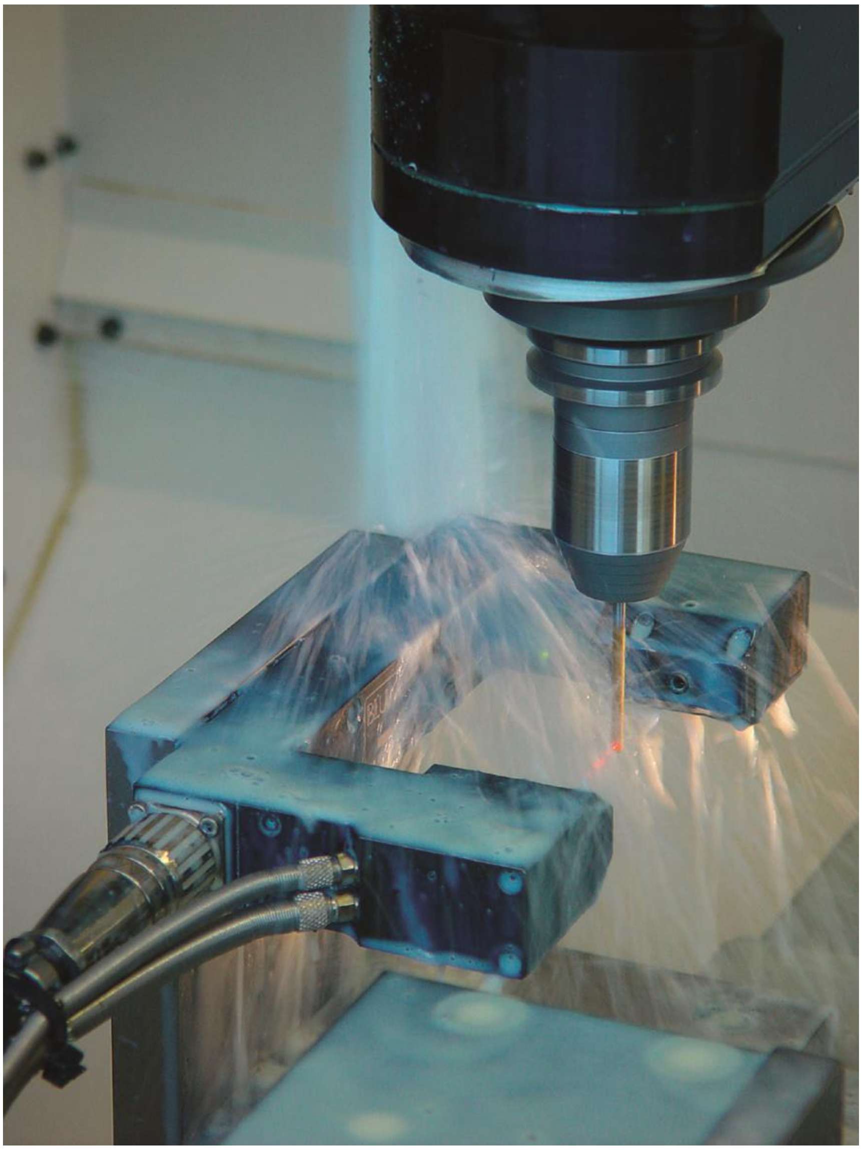


for Laser systems

- Tool Measurement
- NT Technology
- Tool Length Measurement
- Tool Radius Measurement
- Tool Breakage Detection
- Tool Form Monitoring
- Tool Form Measurement
- Single Cutting Edge Monitoring
- Axes Compensation

Programming instructions	**English**
Software	148844
Licence	
Version	V5F
Control	Okuma OSP-P200M, P300M

Copyright

All data has been carefully checked. Nevertheless, no guarantee can be given for completeness, correctness and errors.

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

Prior to the start up procedure 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 of used parameters and data can have unpredictable influence on the machine and could cause damage.
After finishing the installation, the used parameters and the performed changes to programme 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. Installing the measuring cycles, the mentioned warranty exclusions are accepted.
We reserve the right for technical modifications which improve the product. All suggestions for improvement are gladly accepted.
Decisive for the technical contents is the language version of the manufacturer (DE/EN).
Original operating instructions

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

Order number Software until 03-2018: P87.0634-030.315
Order number Software as from 04-2018: 148844

Content

1. Introduction

The BLUM Laser measuring system is a highly accurate optical measuring device for automated tool measurement and tool monitoring on a machining centre under operating conditions. Measuring the tool geometry makes it possible to recognise incorrectly loaded tools or inaccurately pre-set tools in time, as well as worn or damaged tools. This helps to avoid further damage to the workpiece or to the following tool.

1.1 Applications

The following measuring tasks can be solved with the BLUM standard cycles:

- Calibration of the laser system (by O9601)
- Tool length and diameter measurement (by O9602, O9603, O9606)
- Compensation of thermal drift (by O9604)
- Single cutting edge monitoring (cutting edge breakage) on straight cutting edges (by O9605)
- Single cutting edge monitoring (cutting edge breakage) on round cutting edges (by O9607)
- Tool breakage detection / Shaft breakage detection (by O9608)
- Tool identification (by O9602, O9603 with measuring mode: 2)

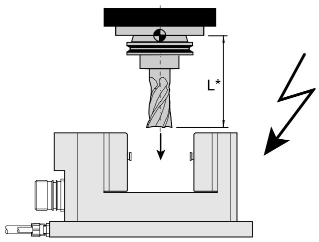
It is possible that some of the functions described herein have not been realized.

CAUTION!

Risk of collision due to false data possible.

During tool measurement using measuring mode NT and tool checking cycles, the tool tip is positioned directly into the beam with rapid traverse.

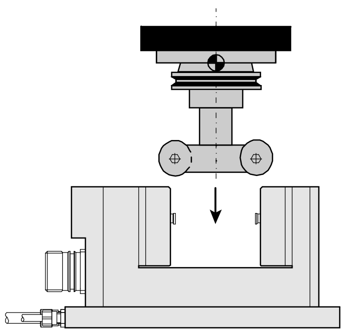
▶ Enter correct tool geometry data in the tool memory.



CAUTION!

Risk of collision due to false data of oversized tools possible.

▶ The tool radius of oversized tools always has to be entered in the tool memory. The indicated interfering contours will then be considered at positioning. Collisions with the laser system can thus be prevented.



1.2 Literature

The following manuals can be ordered from Blum-Novotest GmbH:

1. Laser measuring system
2. Pneumatic unit
3. Installation instructions

For further information about the machine, please refer to the machine documentation supplied by the machine manufacturer.

2. Description of the measuring cycles

2.1 General

Operating mode

When the laser beam is interrupted, the laser system generates a signal at the "OUTPUT DYNAMIC". This output is linked to the CNC measuring input. In the measuring block, the signal is detected by the CNC and the axis positions at the trigger point will be stored. These axis positions are read and processed in the measuring cycles. The axis positions (trigger points) at tool measurement are compensated with those of calibration. So, the tool dimensions are determined.

Cycle call

The measuring cycles can only be carried out in operating "AUTO“ and "MDI“. The measuring task will be defined by the called measuring cycle and the call parameters.
The input of the call parameters must always be in the measuring system of the machine otherwise the cycle is interrupted with an error message. For all examples, metric systems have been selected.
Some call parameters are optional, i.e. they must not be programmed compulsorily, e.g. CALL O9608 instead of CALL O9608 PM=1. PQ=0.5 PW=0. PX=0. Parameters, which have not been transmitted, are automatically set to standard values. In the following chapters they are marked as optional parameters. Essential parameters must be specified in any case, otherwise the error parameter (VC100=4) is set. We recommend that all parameters are specified to make sure that the programme is precisely defined.
Generally, most of the error messages were left aside during programming. Instead of that, the call parameters are internally “redefined” and the cycle is executed with these changed parameters.
Example: Laser measuring system standard (=no NT-function), call of PB=–1, –2 or –3: It is "switched back" on PB=1, 2 or 0 and the cycle is executed.

Spindle rpm

The tool must be in the spindle before the measuring cycle is called. Normally, the tools are measured in rotation. The measurement can be carried out depending on the tool, the spindle and the machine at any spindle rpm. Select the speed you use for machining to take the thermal rpm related spindle drift into account. The measuring time is extended with low spindle speed. If certain tools shall not be checked while rotating, they are usually indexed to a specific spindle position.
The single cutting edge monitoring is automatically performed at the correct checking speed. It depends on the time constant of the used system as well as on the number of cutting edges. Usually, the spindle speed S=3750 1/ min is divided by the number of cutting edges. If the calculated checking speed is higher than the actual speed, the error parameter is set and the cycle is interrupted.

Approach and retracting strategy

Before the measuring task, the machine moves to a retract position in the tool length axis. After that, positioning is made in the laser axis on the measuring position between transmitter and receiver. Then the tool is moved with the radius axis over the beam. Now, the measuring task is being executed. It ends with the approach of a retract position in the tool length axis.

Process error

When a cycle is called, the set parameters are evaluated. If any curious values are detected or if there is risk of collision, the cycle is interrupted with an error message. If any errors occur during the execution of the programme (i.e. no trigger point, tool broken), then the cycle will be repeated until the error has been eliminated or until the maximum number of attempts has been reached. If the error could not be eliminated, the error parameter is set. If the machine manufacturer has not programmed anything else, the execution of the programme ends with an error message or an endless loop. Machine or user related fault evaluation is usually performed in the sub-routine O9679 (USERENDPROG).

Tool data processing

At the cycle call the H-code of the tool in the spindle are passed to the programme. If no PH-Word is handed over the active H-Code (VHCOD) is used. If H0 was programmed, the active T-Code (VTLCN) is used. If active T-Code (VTLCN) is higher than 999 the HA-code is used (HA-code is not supported by controls older than P300M).
The same appends at the cycle call with the D-code of the tool in the spindle, which are passed to the programme. If no PD-Word is handed over, the active D-Code (VDCOD) is used. If D0 was programmed, the H-Code is used.
If H-Code and D-Code don't match or if H/D-Code and active tool number don't match the programme shows a caution message but with start button the programme could continue the execution.
Some collision checks are only working when the tool radius is set in the programme. Tool length and wear length resp. radius and wear radius are added for the positioning of the tool in the Laser beam. They are corrected by VC114..VC116, so that virtual tool data (zero tool or values generated by the post processor) may be processed.
Used tool length:

TOOLLEN = VC146 + VC148 + VC114 + VC116

Used tool radius:

TOOLRAD = VC147 + VC149 + VC115

In order to ensure correct operation of the tool checking cycles, an initial measurement has to be made with each tool (e.g. with O9602 or O9603 and measuring mode=0). If errors on the workpiece are corrected via the tool data (e.g. in the wear values), the tolerance limits must be extended at the tool control.

Tool measurement and tool setting cycles

During the measuring operation the tool, which is pre-set to ± 5 mm, is measured and the determined data are entered in the tool memory. Measuring speed, travel and retracting are progressively reduced whilst measurement is proceeding. Thus, short measuring times are realised.
Measuring mode –1,–2,–3 consists of positioning the tool tip at rapid traverse feed directly in the laser beam (→ risk of collision in case of incorrect data!). At measuring mode 0,1,2,3 the tool length within the tool limit data is searched. If a tool is too long, it is detected, there is no risk of collision.
Note: Tools with an unknown length can be measured at the tool leading edge with measuring mode = 0 or +3, if value 0 is entered for the length in the tool memory (→ low risk of collision).

Measuring mode	Description	Approach and measuring strategy	Tool data
-3	NT measurement	Positioning directly in the laser beam	• Length / radius must be pre-set roughly in the tool memory (accuracy ±5mm)
		Measurement pulling out of the beam	• Length L / radius R are updated • Wear data are set to 0 • Tool is not evaluated
-2	NT check	Positioning directly in the laser beam	• Length / radius must be measured
		Measurement pulling out of the beam	• Length L / radius R are not updated • Wear data are not updated • Tool is evaluated / locked in error case
-1	NT verification	Positioning directly in the laser beam	• Length / radius must be measured
		Measurement pulling out of the beam	• Length L / radius R have not changed • Wear data are updated • Tool is evaluated / locked in error case
±0	Measurement	Search function to determine the tool data	• Length / radius may be unknown
		Measurement pushing into the beam	• Length L / radius R are updated • Wear data are set to 0 • Tool is not evaluated

Measur-ing mode	Description	Approach and measuring strategy	Tool data
+1	Verification	Search function to de-termine the tool data	• Length / radius must be measured
		Measurement pushing into the beam	• Length L / radius R have not changed • Wear data are updated • Tool is evaluated / locked in error case
+2	Check	Search function to de-termine the tool data	• Length / radius must be measured
		Measurement pushing into the beam	• Length L / radius R are not updated • Wear data are not updated • Tool is evaluated / locked in case of error
+3	NT measure-ment	Search function to de-termine the tool data	• Length / radius may be unknown
		Measurement pushing and pulling	• Length L / radius R are updated • Wear data are set to 0 • Tool is not evaluated

Attention:

On cycle call with measuring mode NT (-3,-2,-1,+3) a minimum spindle speed is required. It must be set before the cycle call.

- S>= 3000 rpm (at Blum LaserControl NT Type Z1-1)
- S>= 600 rpm (at Blum LaserControl NT Type Z1-2)

If the spindle speed is too low, the measuring mode is automatically changed to search run (0, +1, +2).

Cycles for tool check

During the tool check, positioning in the laser beam is done at rapid traverse feed. The correct tool geometry data has to be entered into the tool memory before the checking cycle is called
(→ risk of collision in case of incorrect data!).
If a shaft or cutting edge breakage is detected on the checked tool, the error parameter (VC100) is set. The standard action (error message and programme stop) can be replaced by a machine related action (lock tool, call sister tool, etc.). It usually may be found in programme O9679 (USERENDPROG) and is described in detail in the documentation of the machine manufacturer.

Additional measuring modes for grinding tools

Measurement of grinding tools by the search run is not possible because of increased coolant load. At coarse-pored grinding tools, coolant may leak out of the abrasive wheel. The NT measuring modes (PB: -1, -2, -3) are measuring the biggest diameter (coolant mist or "peak" of the abrasive surface). By additional measuring modes (PB: -11, -12, -13), the smallest diameter is measured ("low point" of the abrasive surface).

Attention:
The additional measuring modes may NOT be used for grinding wheels with segment eruptions and for milling tools! The measured tool data may be too low and may lead to damage of the machine.

Measuring mode	Description	Approach and measuring strategy	Tool data
-13	Measure-ment	Positioning directly in the laser beam	• Length / radius must be pre-set roughly in the tool memory (accuracy ±5mm)
		Measurement pulling out of the beam	• Length L / radius R are updated • Wear data are set to 0 • Tool is not evaluated
-12	Check	Positioning directly in the laser beam	• Length / radius must be measured
		Measurement pulling out of the beam	• Length L / radius R are not updated • Wear data are not updated • Tool is evaluated / locked in error case
-11	Verification	Positioning directly in the laser beam	• Length / radius must be measured
		Measurement pulling out of the beam	• Length L / radius R are not changed • Wear data are updated • Tool is evaluated / locked in error case

Switching the unit of measurement mm/inch

The current measuring system of the machine (mm/inch) will be read at the beginning of the cycle. The basic settings of the variables in the auxiliary programme O967x (USERPARATABx) will be calculated automatically. Therefore, adjustment of these variables is not necessary.

CAUTION!

Malfunction

The call parameter has to be entered depending on the measuring system of the machine (values given in mm or inch). All programme examples are displayed in mm.

Proceeding after change of measuring system from mm to inch
*Executing of programme calibration with values in inch: e.g. CALL O9601 PA=3 PR=0.0004 PX=0 PZ=0
*Call of the requested cycles with values in inch: e.g. tool breakage detection: CALL O9608 PX=0 PM=1 PQ=0.04 PW=0

Proceeding after change of measuring system from inch to mm
*Executing off programme calibration with values in mm: e.g. CALL O9601 PA=3 PR=0.01 PX=0 PZ=0
*Call of the requested cycles with values in mm: e.g. tool breakage detection: CALL O9608 PX=0 PM=1 PQ=1 PW=0

Note example pictures:

Some pictures show examples of Fanuc controls.

2.2 Calibration of the laser measuring system

Before using the laser system it must be calibrated. Thereby, the exact machine co-ordinate positions of the trigger points have to be determined. For this purpose, a special reference tool must be used, its data (length, diameter and height) must be defined in tool memory (A) or in the subroutine O967x (USERPARATABx) (B). Ideally, it will be shrunk into a tool holder.

From the tool geometry of the reference tool and the trigger points, the cycle defines the exact position of the laser beam. The results will be written into calibration parameters. They are not allowed to be overwritten by any other programme. At later tool measurement, these parameters will be used for positioning in the laser beam and for calculation of tool length and tool radius.

Call: CALL O9601 PH.. PD.. PA.. PR.. PZ.. PX..

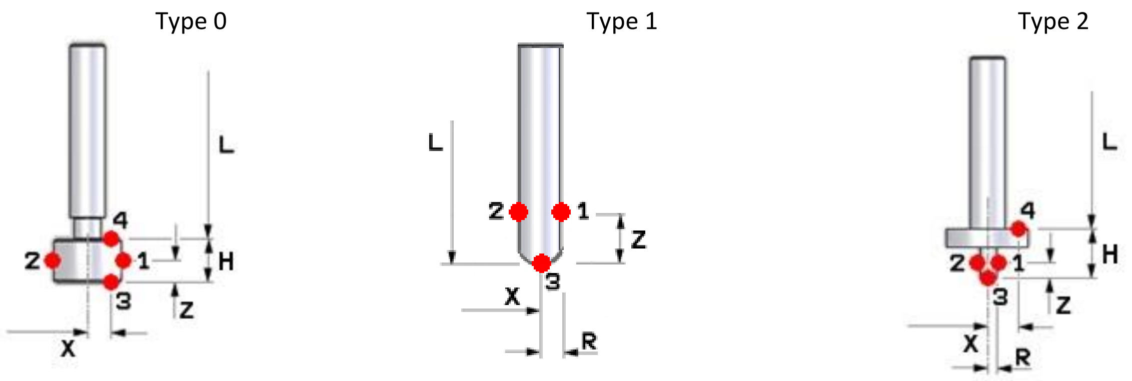
	Parameter	Description	Standard value	
0.	PH,PD	Offset number for tool length and radius		optional
1.	PA	Number of repeated measurements	3	optional
2.	PR	Maximum deviation of measured values	0.010 mm	optional
3.	PZ	Axial measuring position for radius measurement	0.000 mm	optional
4.	PX	Radial measuring position for length measurement	0.000 mm	optional

Return parameter:

The calibration data (VC[VC140+x]) are updated.

Notes:

1. Calibration has to be carried out at working temperature of the machine.
2. With input of PX=0, a suitable standard value is set automatically:
   at type 0 with VC127<8mm: PX = VC127-0.5mm
   at type 0 with VC127>8mm: PX = VC127-1.5mm
   at type 1: PX = 0.
3. With input of PZ=0 a suitable standard value is set automatically.
   at type 0: PZ = VC128/2



A. Entry in tool memory

Type 0	Type 1	Type 2
Length: L	Length: L	Length: L
Wear length: H	Wear length: 0	Wear length: H
Radius: R	Radius: R	Radius: R

B. Entry in O967x (USERPARATAB):

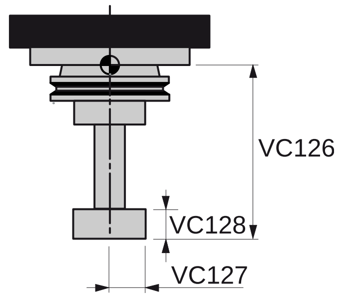
Type 0	Type 1	Type 2
VC126=L+H	VC126=L	VC126=L+H
VC127=R	VC127=R	VC127=R
VC128=H	VC128=0	VC128=H

Recommended call parameters:

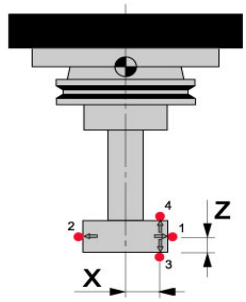
Type 0	Type 1	Type 2
PX = R – 0.5 mm	PX = 0	PX = 6.3 mm
PZ = H / 2	PZ >= R + 0.5 mm	PZ = 2.0 mm

Example for a calibration:

```
T1 M6                                        (Load reference tool)
M3 S3000                                     (Spindle rpm 3000 1/min)
CALL O9601 PH=1 PD=1 PA=3 PR=0.01 PZ=0 PX=0  (Calibration)
T0 M6                                        (Unload reference tool)
```



Dimensions of reference tool



Measuring position

Process:

Loading of reference tool (T1, M6), selection of an offset number (PH=1, PD=1) and spindle start (M3 S3000). The calibration is called by CALL O9601… . Standard values are used for positioning the reference tool in the laser beam (PZ=0., PX=0). The average is determined, as long as the deviation of the individual measured values does not exceed the tolerance (PA=3., PR=0.01). After that, the reference tool is unloaded (T0, M6).

2.3 Temperature compensation of the NC axes

When high precision is required, the laser system and the reference tool can compensate the thermal drift of the machine. The cycle "measures" the reference tool regarding to the reference measurements, determines the actual position of the laser beam and updates the calibration values. The actual position is compared to the position of the temperature compensation reference measurement (TC = temperature compensation). The determined difference can be calculated as follows:
zero point offset = actual offset – reference (e.g. as basic zero point offset).
The evaluation strategy is defined by the machine manufacturer in the sub-routine O9678 (USERSETOFFS). The part programme is then continued with the corrected values.
Temperature compensation with the laser system only makes sense if the measuring system is exposed to the same thermal influences as the workpiece. The best results are obtained with a laser system (MicroCompact or NANO), mounted near the workpiece.

Call: CALL O9604 PH=.. PD=.. PB=.. PA=.. PR=.. PZ=.. PX=.. PS=.. PT=..

	Parameter	Description	Standard value	
0.	PH,PD	Offset number for tool length and radius		optional
1.	PB	Measuring mode	no standard	compulsory
2.	PA	Number of repeated measurements	3	optional
3.	PR	Maximum deviation of measured values	0.010 mm	optional
4.	PZ	Axial measuring position for radius measurement	0.000 mm	optional
5.	PX	Radial measuring position for length measurement	0.000 mm	optional
6.	PS	Admissible temperature drift of length axis	0.050 mm	optional
7.	PT	Admissible temperature drift of radius axis	0.050 mm	optional

Return parameter:

1. The measured drift of the radius- and length axis is available in the programme O9678 (USERSETOFFS) or written into a zero point offset.
2. Error parameter VC100=11
   in case of comparative measurement and exceeding of axial tolerance PS or radial tolerance PT.

Notes:

1. Calibration has to be carried out with the machine at working temperature.
2. With input of PX=0, a suitable standard value is set automatically:
   at type 0 with VC127<8mm: PX = VC127-0.5mm
   at type 0 with VC127>8mm: PX = VC127-1.5mm
   at type 1: PX = 0.
3. With input of PZ=0 a suitable standard value is set automatically.
   at type 0: PZ = VC128/2
   at type 1: PZ = VC127.
4. At a low machine drift, the cycle time can be divided at the reference measurement. Here, PS=xxx.yyy or PT=xxx.yyy is entered. If the measured drift of the first side is <0.xxx, the second side is not measured.
   xxx = 1000 x requested tolerance (in mm) or xxx = 10000 x requested tolerance (inch)
   Recommendation: xxx = 0.2 ... 0.5 x yyy

The integration of temperature compensation cycle (TC) into a normal machining process should be performed as follows:

1. Calibration of the laser system with the reference tool.
2. Length and radius measurement of all tools to be used.
3. Test machining of the first part until all part data are in compliance with rated dimensions.
4. TC reference measurement (PB=0) to keep the actual machine status as a reference.
5. Machining of the next work piece.
6. TC comparative measurement (PB=1) to determine the relevant axes offset with respect to the reference values and processing by means of the additive reference point offset.
7. Depending on thermal machine drift and required precision, TC comparative measurement can be repeated before or after any desired number of machining tasks.

Example for a reference measurement:

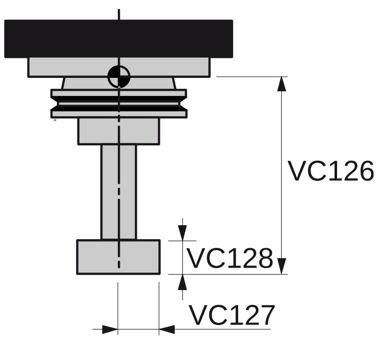
```
T1 M6                                  (Load reference tool)
M3 S3000                               (Spindle rpm 3000 1/min)
CALL O9604 PH=1. PD=1. PB=0. PA=3. PR=0.01 PZ=0. PX=0.
                                       (Reference temperature compensation)
T0 M6                                  (Unload reference tool)
```

Process:

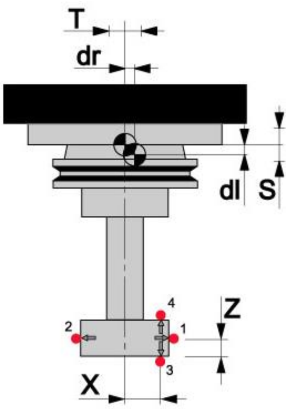
Reference tool is loaded (T1 M6), an offset number is chosen (PH=1 PD=1) and spindle is switched-on (M3 S3000). The temperature compensation is called by CALL O9604. A reference measurement (PB=0) is made. Standard values are used for positioning the reference tool in the laser beam (PZ=0, PX=0). The average is determined, as long as the deviation of the single measured values does not exceed the tolerance (PA=3, PR=0.01). After that, the reference tool is unloaded (T0, M6). Parameters PS and PT may be left out when calling.

Example for a comparative measurement:

```
                                       (Machining of the parts)
T1 M6                                  (Load reference tool)
M3 S3000                               (Spindle rpm 3000 1/min)
CALL O9604 PH=1 PD=1 PB=1 PA=3 PR=0.01 PZ=0.05 PX=0.05
                                       (Reference temperature compensation)
T0 M6                                  (Unload reference tool)
                                       (Continue machining of the parts)
```



Dimensions of reference tool



Measuring position

Process:

Loading of reference tool (T1, M6), selection of an offset number (PH=1, PD=1) and spindle start (M3 S3000). The temperature compensation is called by CALL O9604... . A comparative measuring (PB=1.) is made. Standard values are used to position the reference tool in the Laser beam (PZ=0., PX=0.). The average is determined, as long as the deviation of the individual measured values does not exceed the tolerance (PA=3., PR=0.01). Temperature drift in both axes must be within the authorized tolerances (PS=0.05, PT=0.05). Afterwards, the machining of the parts is continued.

2.4 Tool length measurement of centric tools

Solid core tools (drills, spot facer, ball end mills, etc.) are measured and checked by this cycle. The cycle starts from a tool with an unknown tool length. In order to re-write the measuring result in the tool memory, a tool number (T) and a valid offset no. (PH) have to be chosen.
The most accurate measuring results are obtained, when a calibration or a TC measurement has compensated the thermal drift of the machine.

Call: CALL O9602 PH=.. PB=.. PS=.. PW=.. PA=.. PR=..

	Parameter	Description	Standard value	
0.	PH	Offset number for tool length		optional
1.	PB	Measuring mode	0	optional
2.	PS	Length tolerance	0.050 mm	optional
3.	PW	Additive length offset	0.000 mm	optional
4.	PA	Number of repeated measurements	3	optional
5.	PR	Maximum deviation of measured values	0.010 mm	optional

Return parameter:

1. The measured tool data are available in the tool memory.
2. Error parameter VC100=16
   at PB=±1 or PB=±2 and exceeding of the length tolerance PS.
3. Error parameter VC100=19
   at PB=±1 or PB=±2 and exceeding of the double length tolerance PS.
4. The measured tool length is written into VC126.

Notes:

When exceeding PS or PR, the measurement will be repeated (depending on VC141). Single or incidental error sources (coolant drops or chips, etc.) that cause an incorrect measuring value are mostly excluded by this strategy. Under slightly changed conditions, e.g. blowing off the tool, measurement is carried out again and the value will be either corrected or confirmed.

Example for a tool length measurement:

```
T2 M6                                          (Loading of tool T2)
M3 S3000                                       (Spindle rpm 3000 1/min)
CALL O9602 PH=2. PB=0. PW=0. PA=3. PR=0.01     (Length measurement)
T0 M6                                          (Unload tool)
```

Process:

Loading of the tool (T2 M6), selection of an offset number (PH=2) and spindle start (M3 S3000). The tool length measurement is called by CALL O9602. The measuring cycle executes a search run to determine the tool length by (PB=0). The average is determined, as long as the deviation of the single measured values does not exceed the tolerance (PA=3, PR=0.01). The measured length is written into the tool memory as geometric tool length (PB=0), the additive length offset (PW=0) is entered as wear length. After this, the tool is unloaded (T0 M6). The parameter PS can be omitted on call.

Example for tool length verification (wear measurement):

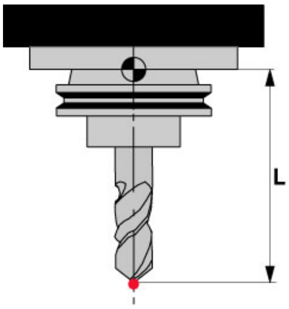
```
                                      (Tool change or machining of the parts)
CALL O9602 PH=2 PB=1 PS=0.05 PW=0 PA=3 PR=0.01
                                      (Length measurement)
                                      (Continue machining of the parts)
```


Process:

Before or after part machining (e.g. with tool T2, offset number PH=2 and spindle speed M3 S3000) the tool length measurement is called with CALL O9602. The measuring cycle executes a search run to determine the tool length by (PB=1). The average is determined, as long as the deviation of the single measured values does not exceed the tolerance (PA=3, PR=0.01). The tool length from the tool memory is deducted from the measured length. The length offset (PW=0) is added to this calculated difference. The result is entered as wear length in the tool memory (PB=1). The error parameter is set when the difference exceeds the length tolerance (PS=0.05). Then, the tool is unloaded (T0 M6) and machining of the parts is continued.

Example for a tool length check:

```
T2 M6                                  (Loading of tool T2)
M3 S3000                               (Spindle rpm 3000 1/min)
                                       (Tool change or machining of the parts)
CALL O9602 PH=2. PB=-2. PS=0.05 PA=3. PR=0.01
                                       (Length measurement)
                                       (Continue machining of the parts)
```



Process:

Loading of the tool (T2 M6), selection of an offset number (PH=2) and spindle start (M3 S3000). The tool length measurement is called by CALL O9602. The tool length entered in the tool memory is used for fast positioning of the tool tip in the laser beam (PB=–2). The average is determined, as long as the deviation of the single measured values does not exceed the tolerance (PA=3, PR=0.01). The tool length from the tool memory is deducted from the measured length. The length offset (PW=0) is added to this calculated difference. The tool memory is not modified (PB=–2). The error parameter is set when the difference exceeds the length tolerance (PS=0.05). Then, the tool is unloaded (T0 M6) and machining of the parts is continued.

2.5 Tool setting in length and radius with run-out monitoring

Tools without a solid core (e.g. end mills, face mills, slot mills, disk milling cutters, ...) are checked and measured with this cycle. The cycle starts from a tool with unknown tool data. The measuring position (PZ, PX) must be specified when calling. In order to re-write the measuring result in the tool memory a tool number (T) and a valid offset no. (PH, PD) have to be chosen.
The most accurate measuring results are obtained, when a calibration or a TC measurement has compensated the thermal drift of the machine.

Call: CALL O9603 PH=.. PD=.. PB=.. PE=.. PM=.. PZ=.. PX=.. PS=.. PT=.. PC=.. PQ=.. PW=.. PU=.. PA=.. PR=..

	Parameter	Description	Standard value	
0.	PH,PD	Offset number for tool length and radius		optional
1.	PB	Measuring mode	0	optional
2.	PE	Scope of measurement	0	optional
3.	PM	Special function	-1	optional
4.	PZ	Axial measuring position for radius measurement	± 0.5 mm	optional
5.	PX	Radial measuring position for length measurement	0	optional
6.	PS	Length tolerance	0.050 mm	optional
7.	PT	Radius tolerance	0.050 mm	optional
8.	PC	Number of cutting edges	0	optional
9.	PQ	Admissible run-out tolerance	0.050 mm	optional
10	PW	Additive length offset	0.000 mm	optional
11.	PU	Additive radius offset	0.000 mm	optional
12.	PA	Number of repeated measurements	3	optional
13.	PR	Maximum deviation of measured values	0.010 mm	optional

Return parameter:

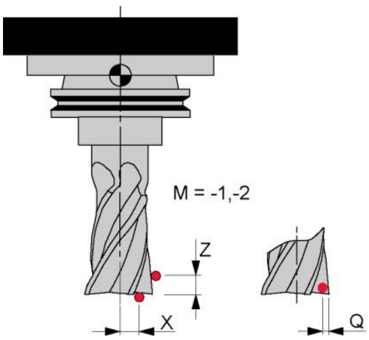
1. The tool data is available in the tool memory.
2. Error parameter VC100=16
   for PB=±1 or ±2 and exceeding length tolerance PS or radius tolerance PT.
3. Error parameter VC100=19
   for PB=±1 or ±2 and exceeding the double length tolerance PS or the double radius tolerance PT.
4. Error parameter VC100=15
   in case of exceeded concentricity tolerance PQ.
5. Error parameter VC100=13
   if the calculated control RPM exceeds the actual spindle RPM. Spindle over speed is prevented to protect tool and machine from damage.
6. The measured tool length is written into VC126, the measured tool radius into VC127. At radius measurement on both sides the measured tool radius of the second side is written into VC128.
7. At PC<-1 depending on the measuring scope, the shortest cutting edge is measured additionally and the minimum value is entered into VC185 (length) or VC186 (radius). The maximum values are available in the tool memory and in VC126..VC128.

Notes:

1. When exceeding PS, PT or PR the measurement will be repeated (depending on VC141). Single or incidental error sources (coolant drops or chips, etc.) that cause an incorrect measuring value are mostly excluded by this strategy. Under slightly changed conditions, e.g. blowing off the tool, measurement is carried out again and the value will be either corrected or confirmed.
2. With PM=+1/+2, the tool length has to be pre-set in the tool memory (±1.8 mm or VC123), otherwise there is risk of collision!
3. At PC=0 no single cutting edge control is carried out.
4. At PC=-1 a measuring block in tool axis direction is carried out instead of a concentricity check. Hereby the laser beam must continuously be interrupted. So, a concentricity check can be carried out e.g. on the tool shaft.
5. At PE=-1, -2, -3, the tool spindle has to be orientated.

Example for the measuring of an end mill:

```
T3 M6                                           (Load tool T3)
M3 S3000                                        (Spindle rpm 3000 1/min)
CALL O9603 PH=3. PD=3. PB=0. PE=0. PM=-1. PZ=0.5 PX=4.5 PC=4. PQ=0.05 PW=0.
PU=0. PA=3. PR=0.01                             (Tool setting)
T0 M6                                           (Unload tool)
```



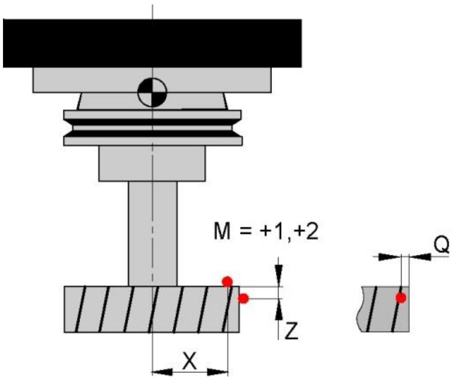
PM = Radius measurement, -1 = on one side or -2 = on both sides
PQ = Runout tolerance
PX = Radial measuring position for length measurement
PZ = Axial measuring position for radius measurement

Process:

Loading of the tool (T3 M6), selection of an offset number (PH=3 PD=3) and spindle start (M3 S3000). The tool length measurement is called by CALL O9603. The measuring cycle executes a search run to determine the tool length by (PB=0). Tool length and radius are measured (PE=0). The length is measured on the front edge, the radius measurement is made on one side (PM=–1). The measuring positions are defined with PX=4.5 and PZ=5. The average is determined, as long as the deviation of the single measured values does not exceed the tolerance (PA=3, PR=0.01). The cutting edges are checked to see if they are within run-out tolerance (PC=4, PQ=0.1). The measured values for length and radius are written as geometric tool data into the tool memory (PB=0). The additive length and radius offset (PW=0, PU=0) are entered in the wear parameters. After this, the tool is unloaded (T0 M6). Parameters PS and PT may be left out when calling.

Example for the measurement of a disk milling cutter:

```
T3 M6                                              (Load tool T3)
M3 S3000                                           (Spindle rpm 3000 1/min)
CALL O9603 PH=3. PD=3. PB=0. PE=0. PM=1. PZ=-0.5 PX=24.5 PC=4. PQ=0.05 PW=0.
PU=0. PA=3. PR=0.01                                (Tool setting)
T0 M6                                              (Unload tool)
```



PM = Radius measuring, +1 = on one side or +2 = on both sides
PQ = Runout tolerance
PX = Radial measuring position for length measuring
PZ = Axial measuring position for radius measuring, negative value!

Caution: With PM=+1/+2 the tool length has to be defined in the tool memory (±5mm), otherwise there is a risk of collision!

Process:

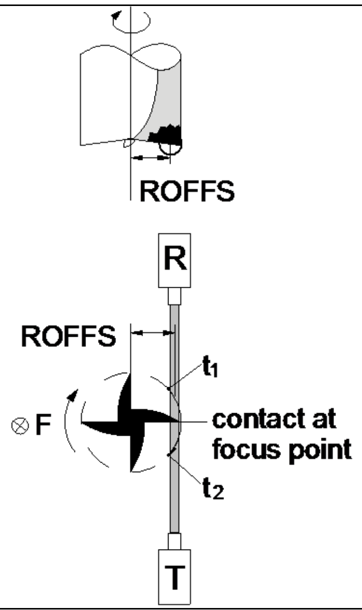
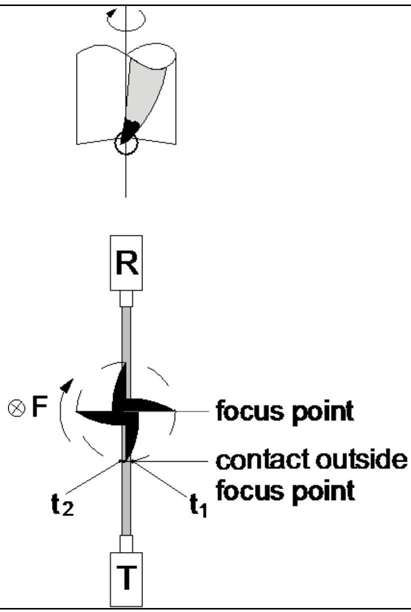
Loading of the tool (T3 M6), selection of an offset number (PH=3 PD=3) and spindle start (M3 S3000). The tool length measurement is called by CALL O9603. The measuring cycle executes a search run to determine the tool length by (PB=0). Tool length and radius are measured (PE=0). The length is measured on the rear edge, the radius measurement is made on one side (PM=1). The measuring positions are defined with PX=24.5 and PZ=-1.5. The average is determined, as long as the deviation of the single measured values does not exceed the tolerance (PA=3, PR=0.01). The cutting edges are checked to see if they are within run-out tolerance (PC=4, PQ=0.05). The measured values for length and radius are written as geometric tool data into the tool memory (PB=0). The additive length and radius offset (PW=0, PU=0) are entered in the wear parameters. After this, the tool is unloaded (T0 M6). Parameters PT and PS may be left out when calling.

CAUTION!

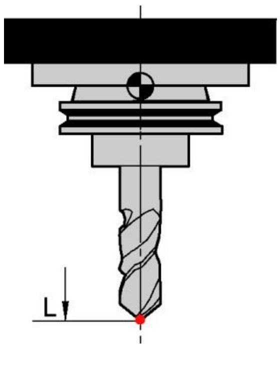
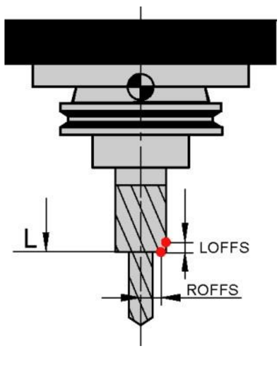
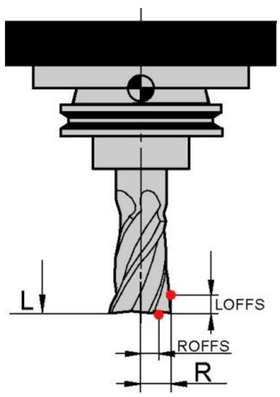
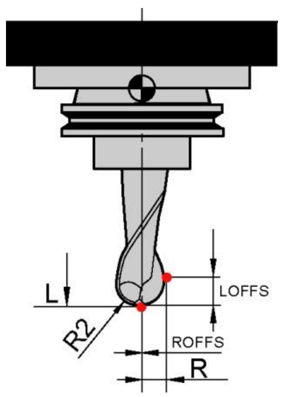
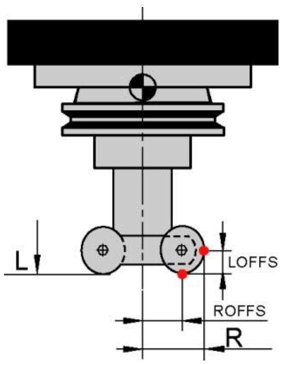
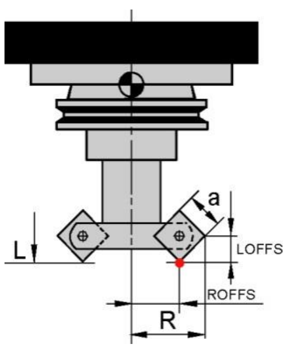
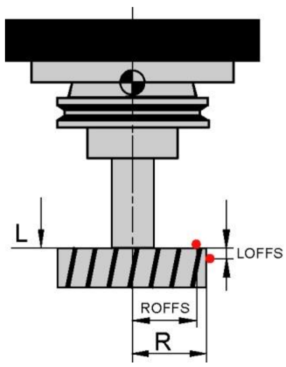
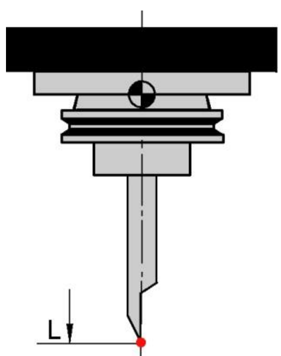
Risk of collision due to false data possible.

- With PM=+1/+2, the tool length has to be pre-set in the tool memory (±1.8 mm or VC123), otherwise there is risk of collision!

Reasons for eccentric measuring position during length measurement:

	
Correct! 1. Frontal shadowing profile of the cutting edge 2. Touch point of the cutting edge in the laser focus 3. Dwell time in the beam t2-t1 is extended, the electronics can safely follow the signal chart.	**Wrong!** 1. Shadowing profile of the side of the cutting edge 2. Touch point out of the laser focus 3. Dwell time in the beam t2-t1 is very short, the electronics can follow the signal chart conditionally.

More examples to define the measuring positions for different types of tools:

 PX=0	 1) PX=0 2) PX=R–0.5mm	 PX=R-R2–0.5mm PZ=R2+0.5mm	 PX=0 PZ=R2+0.5mm
 PX=R-R2–0.5mm PZ=R2	 PX=R–0.707a–0.5mm PZ=0.707a	 PX=R–R2–0.5mm PZ=–(R2+0.5mm)	 PX=0...0.1mm

2.6 Measurement of tool length, radius and corner radius

By this cycle, tools with a corner radius (e.g. ball end mill, torus mill, concave shaped mill, ...) can be measured and controlled. The cycle assumes a tool with approximately known tool data. If an unknown tool should be measured, the measuring position (PX) and the position of the circle centre (PZ, PQ) have to be set in the programme call. In order to re-write the measuring result in the tool memory a tool number (T) and a valid offset no. (PH, PD) have to be chosen.
The most accurate measuring results are obtained, when a calibration or a TC measurement has compensated the thermal drift of the machine.

Call: CALL O9606 PH=.. PD=.. PB=.. PI=.. PK=.. PC=.. PM=.. PJ=.. PX=.. PZ=.. PQ=.. PS=.. PT=.. PW=.. PU=.. PA=.. PR=..

	Parameter	Description	Standard value	
0.	PH,PD	Offset number for tool length and radius		optional
1.	PB	Measuring mode	0	optional
2.	PI	Starting angle to tool symmetric axis	5°	optional
3.	PK	Target angle to tool symmetric axis	90°	optional
4.	PC	Number of measuring points (3..20)	3	optional
5.	PM	Position of cutting edge	7	optional
6.	PJ	Tool corner radius	no standard	compulsory
7.	PX	Radial measuring position for length measurement	0	optional
8.	PZ	Length offset from centre of circle to tool length	0	optional
9.	PQ	Radial offset from centre of circle to tool axis	0	optional
10.	PS	Length tolerance	0.050 mm	optional
11.	PT	Radius tolerance	0.050 mm	optional
12	PW	Additive length offset	0.000 mm	optional
13.	PU	Additive radius offset	0.000 mm	optional
14.	PA	Number of repeated measurements	3	optional
15.	PR	Maximum deviation of measured values	0.010 mm	optional

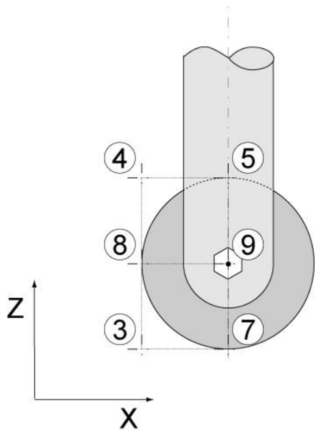
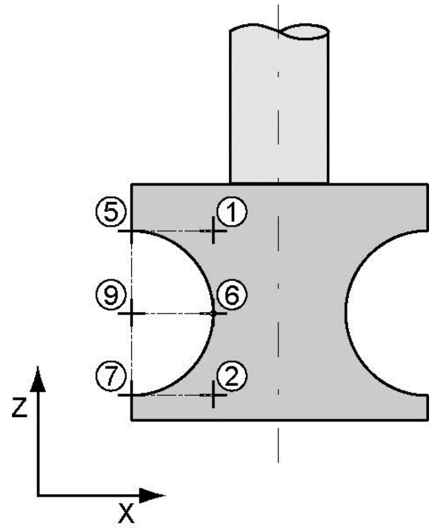
Return parameter:

1. The tool data is available in the tool memory.
2. Error parameter VC100=16
   for PB=±1 or ±2 and exceeding length tolerance PS or radius tolerance PT.
3. Error parameter VC100=19
   for PB=±1 or ±2 and exceeding the double length tolerance PS or the double radius tolerance PT.
4. The measured position of the circle point is written into VC126 (length axis) and VC127 (radius axis). The measured tool corner radius is written into VC128.

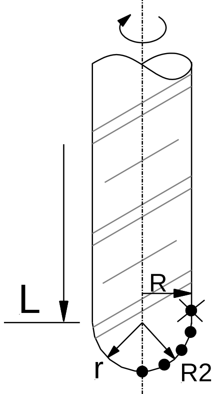
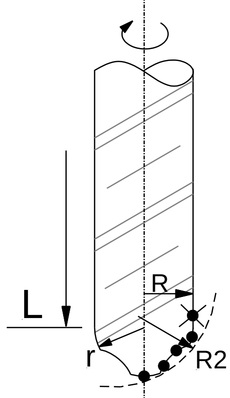
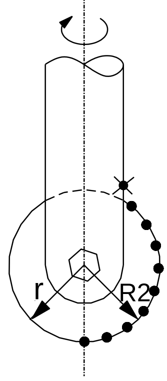
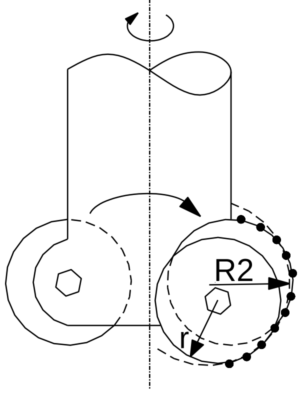
Notes:

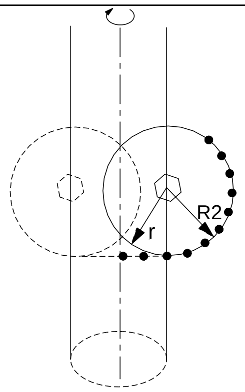
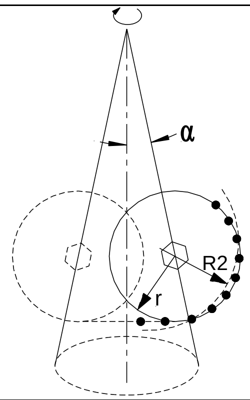
1. At least 3 measuring values are needed to calculate the circle parameter.
2. The measuring direction depends on the angle position of the measuring point. In the angle ranges 0°…45°, 135° … 225° and 315°…360° the measurements are carried out in the length axis, otherwise in the radius axis.
3. PI = 0 / PK = 0: From the parameter PM suitable values for PI and PK are set (90° segment).

Influences of the cutting edge geometry on the possible tool nose position (Call parameter PM):

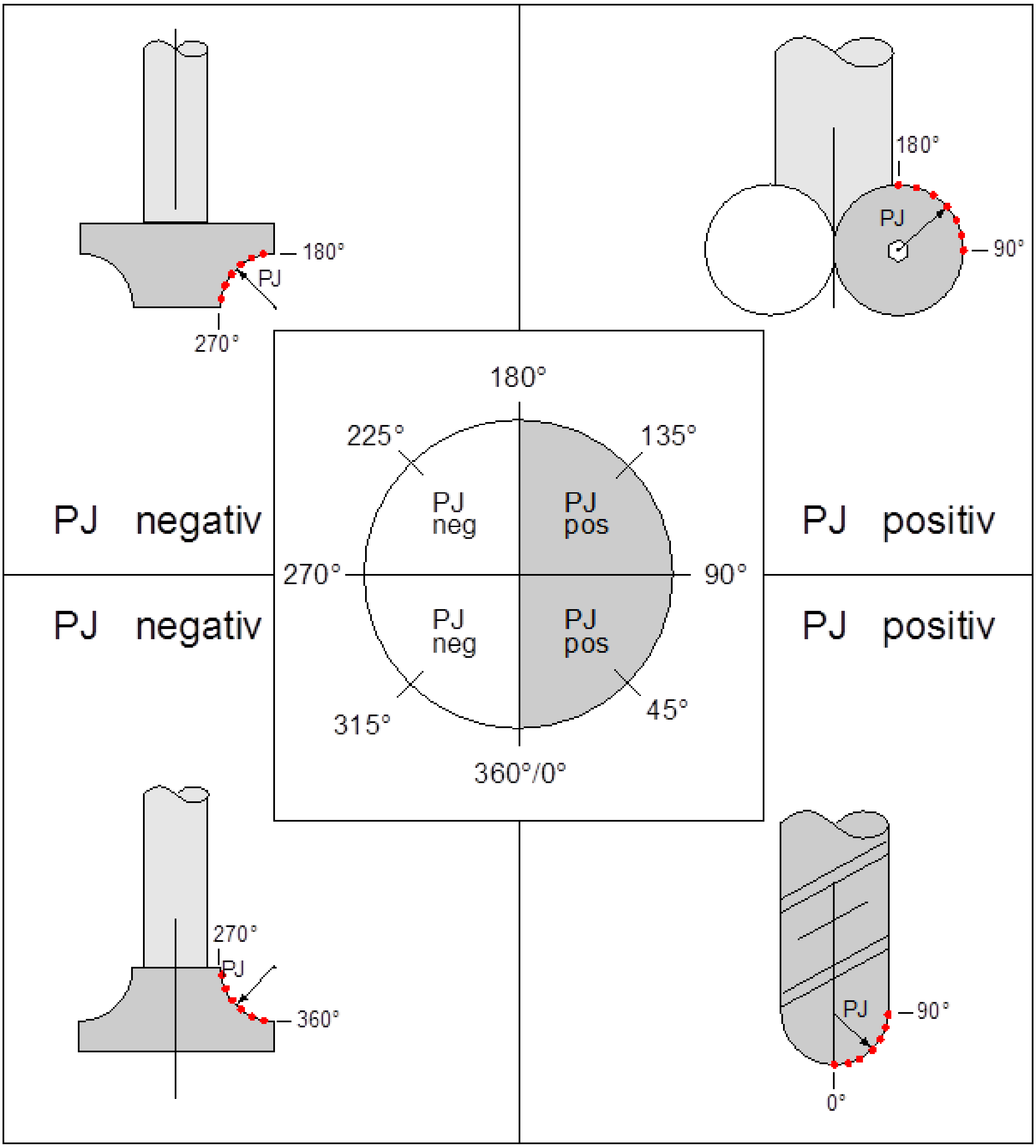
	
Tool nose position on convex radius shape: 0° <= PI, PK<= 180°, PJ > 0	Tool nose position on concave radius shape: 180° <= PI, PK <= 360°, PJ < 0

Influence of defective tools and wrong parameters on the measured corner radius (VC128)

			
PJ ≅ r	**PJ > r**	**PJ ≅ r**	**PJ > r**

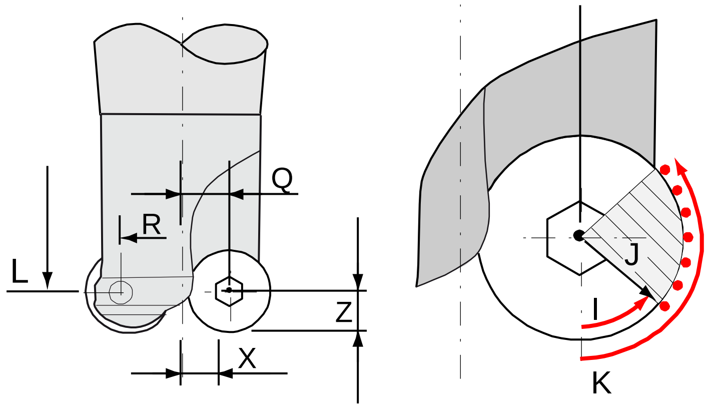
	
PJ ≅ r Eccentric tool clamping: Tool axis describes a cylinder	**PJ > r** Inclined tool clamping: Tool axis describes a cone

Influence of cutting edge geometry on the corner radius PJ, start angle PI, target angle PK



Example for the measuring of a torus mill:

```
T6 M6                                        (Load tool T6)
M3 S3000                                     (Spindle rpm 3000 1/min)
CALL O9606 PH=6. PD=6. PB=0. PI=0. PJ=6. PK=120. PC=7. PM=9 PX=8.5 PZ=6. PQ=9.
PS=0.05 PT=0.05 PW=0. PU=0. PA=3. PR=0.01 (Tool setting)
T0 M6                                        (Unload tool)
```

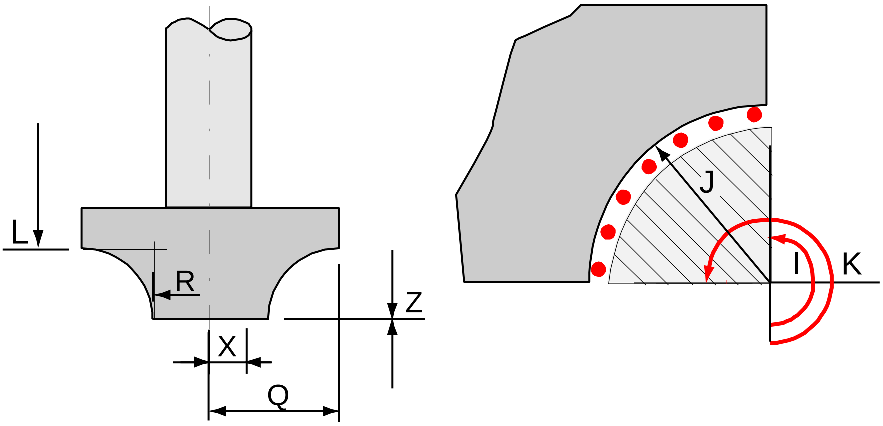


Process:

Loading of the tool (T6 M6), selection of an offset number (PH=6 PD=6) and spindle start (M3 S3000). The tool length measurement is called by CALL O9606. The measuring cycle executes a search run to determine the tool length by (PB=0). At first, the length is measured beside the tool axis (PX=8.5) and afterwards, the corner radius is determined. The circle centre is pre-defined with PZ=6 and PQ=9, the corner radius with PJ=6 (convex shape). The measuring positions are calculated by the cycle with the start angle (PI=0), target angle (PK=120) and the number of measuring points (PC=7). The first measuring point is determined by a measuring stroke in axial direction and is 9mm (PQ) away from the tool axis at start angle 0° (PI). With angle steps of 20°, the L/R-coordinates of seven (PC) measuring points are determined one after another on the circular tool shape. Every position is repeatedly measured and the average is determined as long as the tolerance (PA=3, PR=0.01) is not exceeded. The last measuring point is determined by a measuring stroke in radial direction and is above the maximum tool radius at target angle 120° (PK). The circle radius and centre are calculated by a circle regression using the measured values. The values of the circle centre (PM=9) are taken as length and radius and are written as geometric tool data (length and radius) in the tool memory (PB=0). The additive length and radius offset (PW=0, PU=0) are entered in the wear parameters. After this, the tool is unloaded (T0 M6). Parameters PS and PT may be left out when calling.

Example for checking a convex/concave shaped mill:

```
T6 M6                                                 (Load tool T6)
M3 S3000                                              (Spindle rpm 3000 1/min)
CALL O9606 PH=6. PD=6. PB=1. PI=180. PJ=-6. PK=270. PC=7. PM=1. PX=2.5 PZ=0.
PQ=9. PS=0.05 PT=0.05 PW=0 PU=0 PA=3. PR=0.01         (Tool setting)
T0 M6                                                 (Unload tool)
```



Process:

Loading of the tool (T6 M6), selection of an offset number (PH=6 PD=6) and spindle start (M3 S3000). The tool length measurement is called by CALL O9606. The measuring cycle executes a search run to determine the tool length by (PB=1). At first, the length is measured beside the tool axis (PX=2.5) and afterwards, the corner radius is determined. The corner radius (concave shape) is pre-defined with PJ=-6. The measuring positions are calculated by the cycle with the start angle (PI=180), target angle (PK=270) and the number of measuring points (PC=7). The first measuring point is determined by a measuring stroke in axial direction and is 9 mm away from the tool axis at start angle 180° (PI). With angle steps of 15° the L/R-coordinates of seven (PC) measuring points are determined one after another on the circular tool shape. Every position is repeatedly measured and the average is determined as long as the tolerance (PA=3, PR=0.01) is not exceeded. The last measuring point is determined with a measuring stroke in radial direction. The circle radius and centre are calculated by a circle regression using the measured values. The result of circle centre minus radius is the length and the radius is the result of the circle centre minus radius (M=1). The tool length from the tool memory is deducted from the measured length and the tool radius from the tool memory is deducted from the measured radius. These calculated differences will be added to the length or radius offset (PW=0, PU=0). The result is written into the tool memory as length or radius wear (PB=1). The error parameter is set if the difference exceeds the length e.g. or radius tolerance (PS=0.05, PT=0.05). After this, the tool is unloaded (T0 M6).

2.7 Single cutting edge monitoring on a straight edge

The edges or the wear of critical tools are checked before or after machining to guarantee a faultless part production. For control the rotating tool is driven through the laser beam to see if all the cutting edge diameters are within run-out tolerance.
This cycle is suitable to check end mills and taps, facing heads, etc. This cycle starts from a tool with known geometric tool data. In order to read the tool data, a valid tool number (T) and a valid offset no. (PH, PD) must be selected.

Call: CALL O9605 PH=.. PD=.. PC=.. PQ=.. PZ=.. PV=.. PA=.. PF=..

	Parameter	Description	Standard value	
0.	PH,PD	Offset number for tool length and radius		optional
1.	PC	Number of cutting edges	1	optional
2.	PQ	Admissible run-out tolerance	0.050 mm	optional
3.	PZ	Axial offset of starting position	0.500 mm	optional
4.	PV	Scan distance or thread pitch	0	optional
5.	PA	Number of measuring points for thread mills	0	optional
6.	PF	Positioning feed rate	100 mm/min	optional

Return parameter:

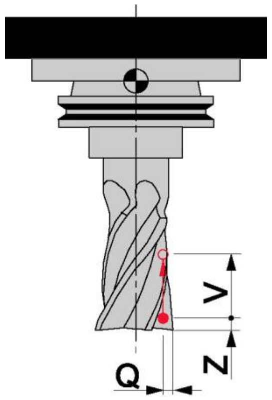
1. Error parameter VC100=15
   in case of exceeded concentricity tolerance PQ
2. Error parameter VC100=13
   if the calculated control RPM exceeds the actual spindle RPM. Spindle over speed is prevented to protect tool and machine from damage.

Notes:

1. PA saw blade can only be checked for segment cut-out. A measuring block is carried out, hereby the laser beam must continuously be interrupted. The parameters must be set as follows:
   PC=-1 → a measuring block is carried out instead of a single cutting edge monitoring.
   PQ → the selected value must be greater than the tooth depth.
   PZ → the start position for the measuring block must be within the saw blade.
   PV → the target position for the measuring block must be within the saw blade.

Example for single cutting edge monitoring of an end mill:

```
T5 M6                                  (Load tool T5)
M3 S8000                               (Max. spindle rpm 8000 1/min)
                                       (Machining of the parts)
CALL O9605 PH=5. PD=5. PC=3. PQ=0.05 PZ=0.5 PV=6. PA=0. PF=100.
                                       (Single cutting edge monitoring)
                                       (Continue machining of the parts)
```



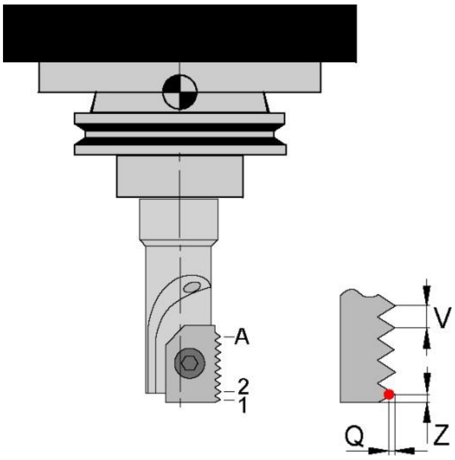
PQ = Admissible runout tolerance
PV = Checking stroke shaft/cone
PF = Measuring feed
PZ = Length offset for cutting edge control

Process:

Loading of the tool (T5 M6), selection of an offset number (PH=5 PD=5) and spindle start (M3 S8000). The single cutting edge monitoring is called by CALL O9605. The spindle rpm is adapted to the number of cutting edges of the tool (PC=3), but it cannot be higher than the actual spindle rpm. The geometry data are read in the tool memory and are used for positioning. The axial start position is located at (tool length – PZ), the radial position at (tool radius – PQ). The movement in the length axis is PV=6 with PF=100. Within this traverse path the cutting edges are checked to be within breakage tolerance (PQ=0.05). The error parameter is set when cutting edge breakage is detected. Afterwards the machining of the parts is continued.

Example for single cutting edge monitoring of a thread mill:

```
T5 M6                                      (Load tool T5)
M3 S8000                                   (Max. spindle rpm 8000 1/min)
                                           (Machining of the parts)
CALL O9605 PH=5. PD=5. PC=4. PQ=0.1 PZ=0.5 PV=1.5 PA=5. PF=500.
                                           (Single cutting edge monitoring)
                                           (Continue machining of the parts)
```



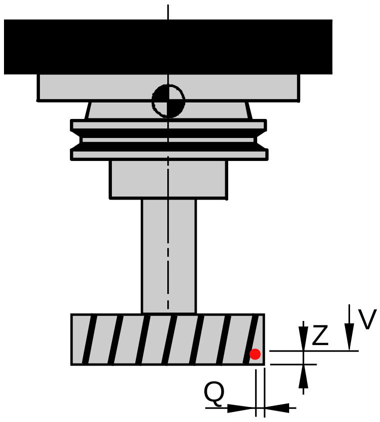
PQ = Admissible runout tolerance
PV = Distance of the teeth, pitch
PA = Number of spirals
PZ = Starting position for tooth control

Process:

Loading of the tool (T5 M6), selection of an offset number (PH=5 PD=5) and spindle start (M3 S8000). The single cutting edge monitoring is called by CALL O9605. Spindle rpm is adapted to the number of cutting edges (PC=4), but it cannot be higher than the actual spindle rpm. The geometry data are read in the tool memory and are used for positioning. The axial position of the front tooth row is located at (tool length –PZ), the radial position at (tool radius – PQ). The distance between the tooth rows is PV=1.5. Positioning between the tooth rows is made at feed (PF=500). In each tooth row (number PA=5) the cutting edges are checked to be within breakage tolerance (PQ=0.1). The error parameter is set when cutting edge breakage is detected. Afterwards, the machining of the parts is continued.

Example for the check of a saw blade:

```
T5 M6                                  (Load tool T5)
M3 S8000                               (Max. spindle rpm 8000 1/min)
                                       (Machining of the parts)
CALL O9605 PH=5 PD=5 PC=-1 PQ=0.1 PZ=0.5 PV=1 PA=0 PF=500
                                       (Single cutting edge monitoring)
                                       (Continue machining of the parts)
```



Process:

Loading of the tool (T5 M6), selection of an offset number (PH=5 PD=5) and spindle start (M3 S8000). The concentricity check is called by CALL O9605. The spindle speed is not changed (PC=-1). The geometry data are read in the tool memory and are used for positioning. The axial start position is located at (tool length – PZ), the radial position at (tool radius – PQ). In the length axis, a measuring block is carried out with PV=1 and feed PF=100. The error parameter is set when the target position is not reached, i.e. break of a saw blade is detected. Afterwards, the machining of the parts is continued.

2.8 Fast single cutting edge monitoring on a straight edge

The edges or the wear of critical tools are checked before or after machining to guarantee a faultless part production. For checking purposes, the rotating tool is driven through the laser beam to see if all the cutting edge diameters are within run-out tolerance.
This cycle is suitable to check end mills and taps, facing heads, etc. This cycle starts from a tool with known geometric tool data. In order to read the tool data, a valid tool number (T) and a valid offset no. (PH, PD) must be selected.

Call: CALL O9685 PH=.. PD=.. PC=.. PQ=.. PZ=.. PV=.. PA=.. PF=.. PB=..

	Parameter	Description	Standard value	
0.	PH,PD	Offset number for tool length and radius		optional
1.	PC	Number of cutting edges	1	optional
2.	PQ	Admissible run-out tolerance	0.050 mm	optional
3.	PZ	Axial offset of starting position	0.500 mm	optional
4.	PV	Scan distance or thread pitch	0	optional
5.	PA	Number of measuring points for thread mills	0	optional
6.	PF	Positioning feed rate	100 mm/min	optional
7.	PB	Interim position (tool tip in front of laser beam)	VC120	optional

Return parameter:

1. Error parameter VC100=15
   in case of exceeded concentricity tolerance PQ
2. Error parameter VC100=13
   if the calculated control RPM exceeds the actual spindle RPM. Spindle over speed is prevented to protect tool and machine from damage.

Notes:

1. The positioning of the tool to the interim position at the beginning of the cycle is executed with all axes at the same time (risk of collision!). Possibly, a safe positioning must be approached first.
2. The parameter PB gives the position of the tool tip in front/above the laser beam (interim position). If PB is not programmed, the tool is positioned on the retract position (VC120). If there are no interfering contours on the way from the actual position to the laser beam, the start position (PB= – Z-value) can be chosen as interim position.

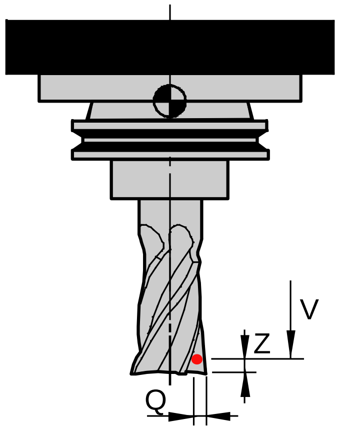
CAUTION!

Risk of collision due to false data possible.

Ensure programming of a suitable interim position at machines with multiple tool spindles. This interim position will be used when positioning the following spindle. This affects the laser systems Nano or MicroCompact in particular.

Example for fast cutting edge monitoring of an end mill:

```
                                        (Machining of the parts)
CALL O9685 PH=5. PD=5. PC=3. PQ=0.1 PZ=0.5 PV=0. PA=0. PF=100. PB=30.
                                        (Single cutting edge monitoring)
                                        (Continue machining of the parts)
```



Process:

The fast single cutting edge monitoring is called by CALL O9685.... The tool data is read from the tool memory and used for positioning. All axes move directly to the start position. At the interim position the tool is 30mm above the laser beam (PB=30). Afterwards, the machining of the parts is continued.

See chapter 3.4 for a detailed description of the other call parameters.

2.9 Single cutting edge monitoring on a round cutting edge geometry

The edges or the wear on critical tools are checked before or after machining to guarantee faultless part production. For control the rotating tool is driven through the laser beam to see if all the cutting edge diameters are within run-out tolerance. Due to the required precision, the contour check is only possible with axis-parallel laser beam adjustment and can only be performed on a focused short-distance laser system (support system MicroCompact/Nano). Moreover, the calibration with the reference tool must be made on four sides, since the exact laser beam centre is taken into account in both axes to calculate the path.
The cycle is suitable for contour checks on ball nose mills and torus mills, inserts with corner radius, etc. This cycle starts from a tool with known geometric tool data. In order to read the tool data, a valid tool number (T) and a valid offset no. (PH, PD) must be selected.

Call: CALL O9607 PH=.. PD=.. PC=.. PQ=.. PX=.. PI=.. PJ=.. PK=.. PV=.. PF=.. PM=..

	Parameter	Description	Standard value	
0.	PH,PD	Offset number for tool length and radius		optional
1.	PC	Number of cutting edges	1	optional
2.	PQ	Admissible run-out tolerance	0.050 mm	optional
3.	PX	Radial offset for the start of the corner radius	0.000 mm	optional
4.	PI	Starting angle towards tool symmetric axis	25°	optional
5.	PJ	Tool corner radius	no standard	compulsory
6.	PK	Target angle towards tool symmetric axis	90°	optional
7.	PV	Checking stroke on shaft/cone	0 mm	optional
8.	PF	Checking feed rate	100 mm/min	optional
9.	PM	Position of cutting edge	0	optional

Return parameter:

1. Error parameter VC100=15
   in case of exceeded length tolerance PQ
2. Error parameter VC100=13
   if the calculated control RPM exceeds the actual spindle RPM. Spindle over speed is prevented to protect tool and machine from damage.

Notes:

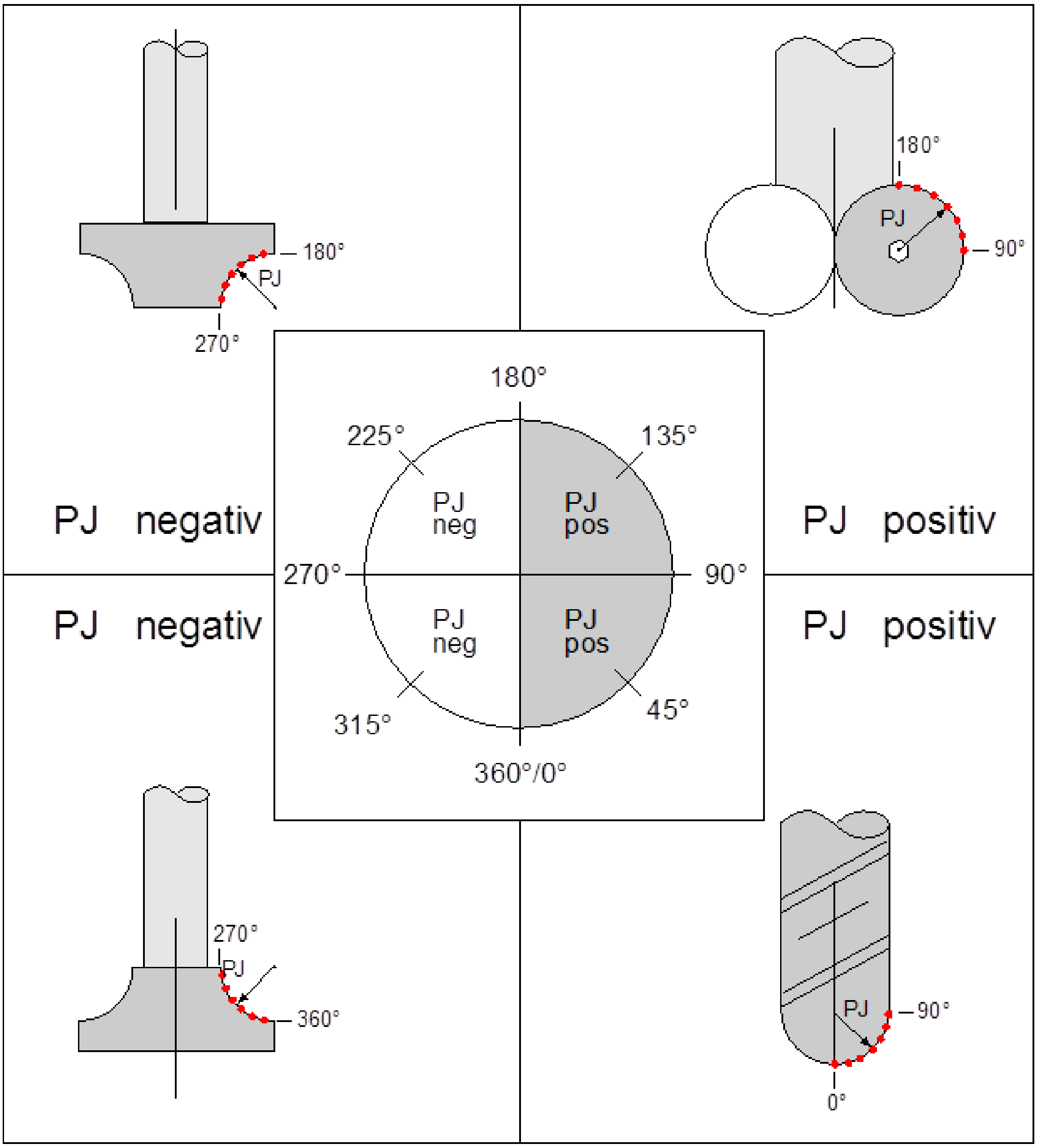
1. If parameter PM is left or set to zero when calling, the tool length up to the tip has to be entered in the tool memory. A radius value in the tool memory is not considered with PM=0.
2. The parameter PM / tool nose position is described in detail in chapter 2.6.

⚠ **CAUTION!**

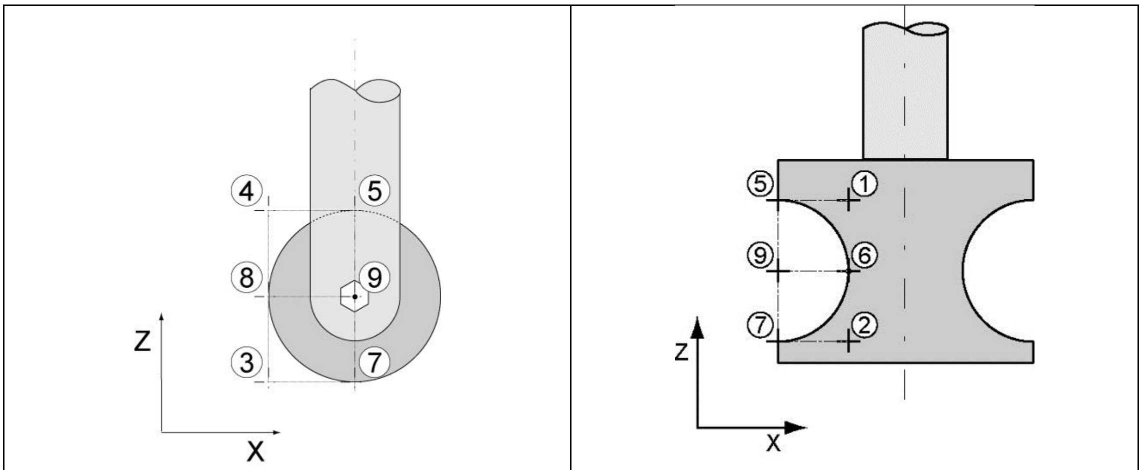
Error in case of angular installation of the laser.

Because of incompatibility reasons, the cycle O9607 can not be used with deviation to the perpendicular line between laser beam and length or radius axis. VC162 and VC163 must be 0. (Check installation instructions).

Influence of cutting edge geometry on the corner radius PJ, start angle PI, target angle PK



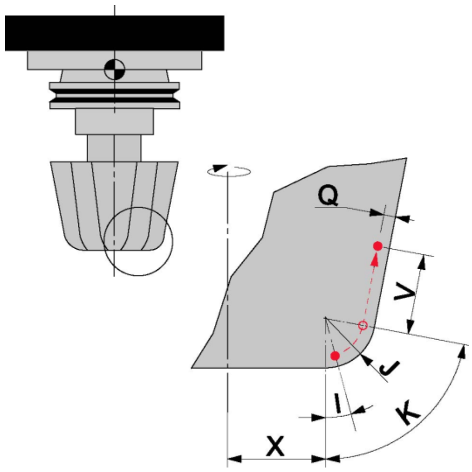
Influences of the cutting edge geometry on the possible tool nose position (Call parameter PM):



Tool nose position on convex radius shape: 0° <= PI, PK<= 180°, PJ > 0	Tool nose position on concave radius shape: 180° <= PI, PK <= 360°, PJ < 0

Example for contour monitoring of a ball end mill:

```
T7 M6                                   (Load tool T7)
M3 S8000                                (Max. spindle rpm 8000 1/min)
                                        (Machining of the parts)
CALL O9607 PH=7. PC=3. PQ=0.05 PX=0. PI=25. PJ=6. PK=90. PV=3. PF=100. M0.
                                        (Contour monitoring)
                                        (Continue machining of the parts)
```



PQ = Admissible runout tolerance
PV = Checking stroke on shaft/cone
PF = Measuring feed
PX = Radial offset for the start of the corner radius
PJ = Tool corner radius
PI = Starting angle to tool symmetric axis
PK = Target angle to tool symmetric axis
PM = Position of cutting edge

Process:

Loading of the tool (T7 M6), selection of an offset number (PH=7 PD=7) and spindle start (M3 S8000). The contour monitoring is called by CALL O9607. Spindle rpm is adapted to the number of cutting edges (PC), but it cannot be higher than the actual spindle rpm. The geometry data is read in the tool memory and is used for positioning. The starting position is calculated by means of the tool data and the call parameters (PQ, PX, PI, PJ and PM). The final position of the arc is calculated from call parameters (PQ, PK, PJ and PM), the final position of the straight line depends on the call parameters (PK, PQ, PV). The feed rate from the start to the target position is PF. Within this traverse path the cutting edges are checked to be within breakage tolerance (PQ). The error parameter is set when cutting edge breakage is detected. Afterwards, the machining of the parts is continued.

2.10 Tool breakage detection

Critical tools are subject to breakage detection before or after machining to guarantee faultless part production. For control the rotating tool is driven through the laser beam to see if all the cutting edge diameters are within run-out tolerance.
This cycle is suitable for breakage detection of drills, taps, gravers, etc. The cycle starts from a tool with a known tool length. In order to read the tool data, a valid tool number (T) and a valid offset no. (PH) must be selected.

Call: CALL O9608 PH=.. PX=.. PM=.. PQ=.. PW=..

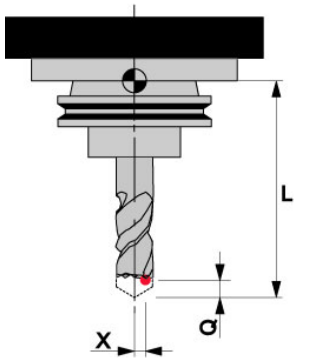
	Parameter	Description	Standard value	
0.	PH	Offset number for tool length		optional
1.	PX	Radial checking position	0.000 mm	optional
2.	PM	Measuring direction	1	optional
3.	PQ	Admissible length tolerance	1 mm	optional
4	PW	Additive length offset	0.000 mm	optional

Return parameter:

1. Error parameter VC100=19
   in case of exceeded length tolerance PQ
2. The measured tool length is written into VC126.

Example for tool breakage detection of a drill:

```
T8 M6                                       (Load tool T8)
M3 S3000                                    (Spindle rpm 3000 1/min)
                                            (Machining of the parts)
CALL O9608 PH=8. PX=0. PM=1 PQ=1. PW=0.     (Breakage detection)
                                            (Continue machining of the parts)
```



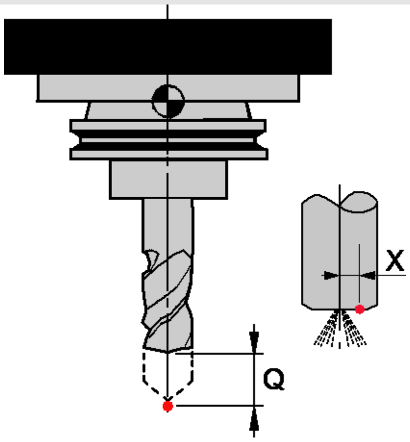
PX = Radial checking position
PQ = Admissible length tolerance
PF = Measuring feed

Process:

Loading of the tool (T8 M6), selection of an offset number (PH=8) and spindle start (M3 S3000). The tool breakage detection is called by CALL O9608. The tool length is read in the tool memory and used for positioning. At the starting position the tool tip stands axially 5*PQ in the laser beam with a radial offset of PX=0 from the centre. Measurement in pull-operation, i.e. dark → light (PM=1). The measured tool length including the additive length offset (PW=0) must be within breakage tolerance (PQ=1). The error parameter is set when shaft breakage has been detected. Afterwards, the machining of the parts is continued.

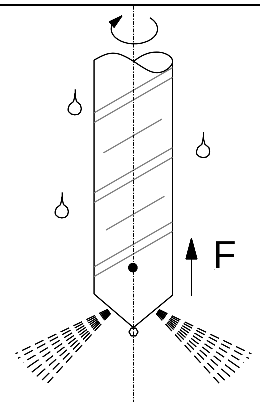
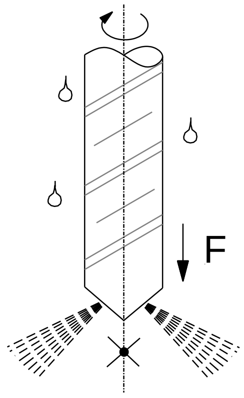
Example for tool breakage detection of an engraving mill:

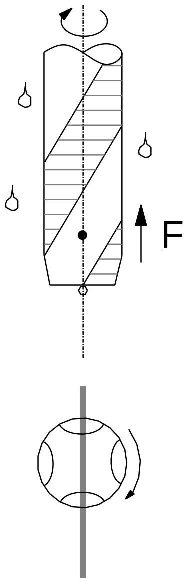
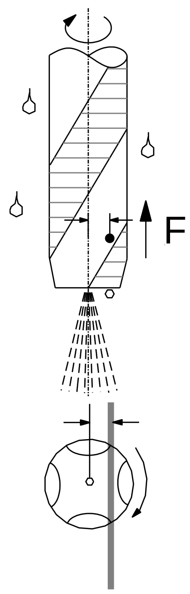
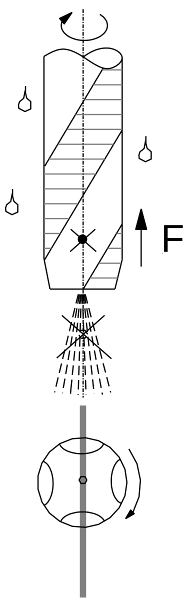
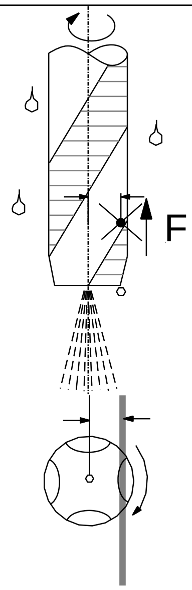
```
T8 M6                                       (Load tool T8)
M3 S3000                                    (Spindle rpm 3000 1/min)
                                            (Machining of the parts)
CALL O9608 PH=8. PX=0. PM=-1. PQ=0.1 PW=0.(Breakage detection)
                                            (Continue machining of the parts)
```

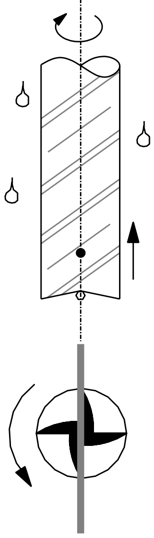
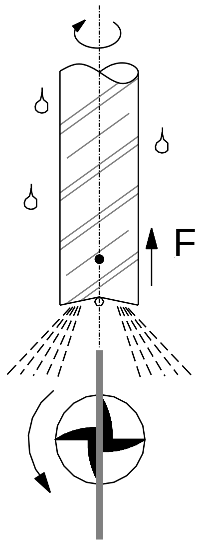
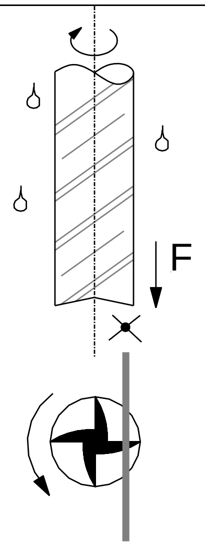
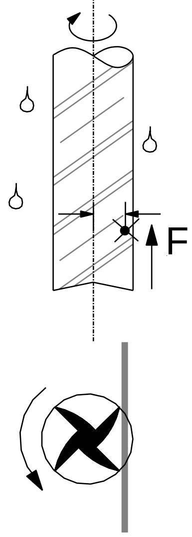


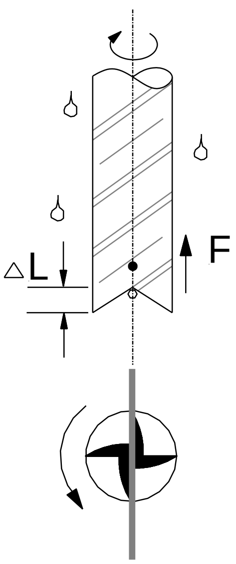
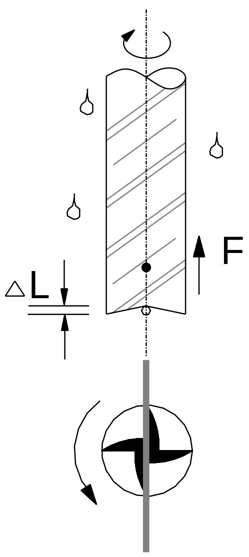
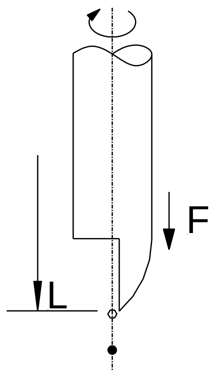
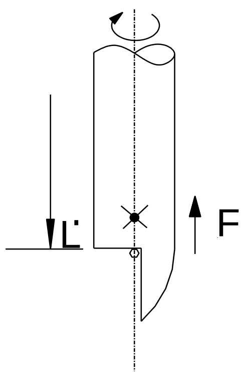
Process:

Loading of the tool (T8 M6), selection of an offset number (PH=8) and start (M3 S3000). The tool breakage detection is called by CALL O9608. The tool length is read in the tool memory and used for positioning. At the starting position the tool tip stands axially 5*PQ in the laser beam, with a radial offset of PX=0 from the centre. Measurement in push-operation, i.e. light → dark (PM=–1). The measured tool length including the additive length offset (PW=0) must be within breakage tolerance (PQ=0.1). The error parameter is set when a shaft breakage has been detected. Afterwards, the machining of the parts is continued.

Tool breakage detection on a drill	
 	
Correct!	**Wrong!**
... PM=**+1** ; MEASURING DIRECTION PX=0 ; RADIAL OFFSET PW=0 ; ADD. LENGTH CORRECTION ...	... PM=**-1** ; MEASURING DIRECTION PX=0 ; RADIAL OFFSET PW=0 ; ADD. LENGTH CORRECTION ... **Wrong measuring direction PM**

Tool breakage detection on a tap			
 	 	 	
Correct!	**Correct!**	**Wrong!**	**Wrong!**
... PM=**+1** PX=**0** PW=0 ...	... PM=**+1** PX≥**0** PW=0 ...	... PM=+1 PX=**0** PW=0 ... **Wrong measuring position PX**	... PM=+1 PX>>**0** PW=0 ... **Wrong measuring position PX**

<table>
<tr><th colspan="4">Tool breakage detection on an end mill</th></tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr><td>Correct!</td><td>Correct!</td><td>Wrong!</td><td>Wrong!</td></tr>
<tr>
<td>...
PM=+1
PX=0
PW≤0
...</td>
<td>...
PM=+1
PX=0
PW≤0
...</td>
<td>...
PM=-1
PX≥0
PW=0
...
Wrong measuring direction PM</td>
<td>...
PM=+1
PX>>0
PW=0
...
Wrong measuring position PX</td>
</tr>
</table>

<table>
<tr><th colspan="2">Tool breakage detection on an end mill</th><th colspan="2">Tool breakage detection on an engraving tool</th></tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr><td>Correct!</td><td>Correct!</td><td>Correct!</td><td>Wrong!</td></tr>
<tr>
<td>...
PM=+1
PX=0
PW=-ΔL
...</td>
<td>...
PM=+1
PX=0
PW=-ΔL
...</td>
<td>...
PM=-1
PX=0
PW=0
...</td>
<td>...
PM=+1
PX=0
PW=0
...
Wrong measuring direction PM</td>
</tr>
</table>

2.11 Fast tool breakage detection

Critical tools are subject to breakage detection before or after machining to guarantee faultless part production. For control the rotating tool is driven through the laser beam to see if all the cutting edge diameters are within run-out tolerance.
This cycle is suitable for breakage detection of drills, taps, gravers, etc. The cycle starts from a tool with a known tool length. In order to read the tool data, a valid tool number (T) and a valid offset no. (PH) must be selected.

This cycle contains only the main sequence of the breakage detection. It must be optimised before it is used by the machine manufacturer or the end user, especially concerning axis exchange, approach and retracting strategy. The difference to other cycles is that no safety checks are executed and no collision warnings are included.

Call: CALL O9688 PH=.. PX=.. PM=.. PQ=.. PW=.. PB=..

	Parameter	Description	Standard value	
0.	PH	Offset number for tool length		optional
1.	PX	Radial checking position	0.000 mm	optional
2.	PM	Measuring direction	1	optional
3.	PQ	Admissible length tolerance	1 mm	optional
4.	PW	Additive length offset	0.000 mm	optional
5.	PB	Interim position (tool tip in front of laser beam)	maximum	optional

Return parameter:

1. Error parameter VC100=19
   in case of exceeded length tolerance PQ
2. The measured tool length is written into VC126.

Notes:

1. The positioning of the tool to the interim position at the beginning of the cycle is executed with all axes at the same time (risk of collision!). Possibly, a safe positioning must be approached first.
2. The parameter PB gives the position of the tool tip in front/above the laser beam (interim position). If PB is not programmed, the tool is positioned on the retract position (VC120). If there is no obstacle on the way from the actual position to the laser beam, the start position (PB= – 5*PQ-value) can be chosen as interim position.
3. The measured tool length is written into VC126.
4. Because of compatibility reasons, call parameter PZ can be used instead of PB.

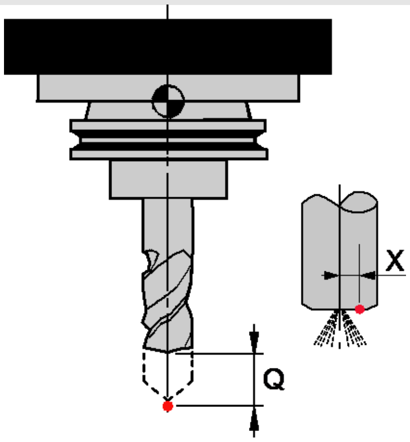
⚠ **CAUTION!**

Risk of collision due to false data possible.

Ensure programming of a suitable interim position at machines with multiple tool spindles. This interim position will be used when positioning the following spindle. This affects the laser systems Nano or MicroCompact in particular.

Example for fast tool breakage detection of a drill:

```
                                                (Machining of the parts)
CALL O9688 PH=8. PX=0. PM=1. PQ=1. PW=0. PB=30. (Breakage detection)
                                          (Continue machining of the parts)
```



Process:

The fast tool breakage detection is called by CALL O9688. The tool length is read in the tool memory and used for positioning. All axes move directly to the interim position. The interim position (PB) is chosen so that the tool tip axial 5*PQ is in the laser beam (time optimised). There is no radial offset of the tool axis. Measurement in pull-operation, i.e. dark → light (PM=1). The measured tool length including the additive length offset (PW=0) must be within breakage tolerance (tool length-PQ<measured length<tool length+2*PQ). Afterwards, the machining of the parts is continued.

3. Tables

3.1 Programme overview

Main programmes

These programmes shall demonstrate as an example which way the cycle related parameters are defined and how the measuring cycles are called and started. The parameter definitions and the cycle call can directly be taken over into a part programme. Alternatively, depending on the control, the call can be made via GUI.

O601	Main programme 2-axes calibration (length and radius)
O602	Main programme length measurement of centric tools (drills)
O603	Main programme length and radius measurement of non centric tools (mills)
O604	Main programme thermal 2-axes compensation (length and radius)
O605	Main programme single cutting edge monitoring on straight cutting edges
O606	Main programme length-, radius and cutting edge radius measurement
O607	Main programme contour monitoring on round cutting edges (ball nose mills)
O608	Main programme tool breakage detection (shaft breakage)
O685	Main programme fast single cutting edge monitoring
O688	Main programme fast breakage detection

Sub-routines

These programmes contain the actual measuring cycles. Any modification of the execution of the programme should only be made after having consulted the manufacturer.

O9601	Sub-routine 2-axis calibration (length and radius)
O9602	Sub-routine length measurement of centred tools (drills)
O9603	Sub-routine length and radius measurement of non centred tools (mills)
O9604	Sub-routine thermal 2-axis compensation (length and radius)
O9605	Sub-routine single cutting edge monitoring on straight cutting edges
O9606	Sub-routine length-, radius and cutting edge radius measurement
O9607	Sub-routine contour checking on round cutting edges (ball nose mills)
O9608	Sub-routine tool breakage detection (shaft breakage)
O9685	Sub-routine fast single cutting edge monitoring on straight cutting edges
O9688	Sub-routine fast tool breakage detection (shaft breakage)

General auxiliary programmes

These auxiliary programmes are called by the different measuring cycles. Following programmes are necessary for the execution:
O9630, O9632, O9633, O9634, O9636, O9637, O9638, O9639, O9640, O9641, O9642, O9643, O9644, O9646, O9650..O9669 (depending on the language).

Machine-specific auxiliary programmes

These auxiliary programmes are called by the different measuring cycles. During the start-up of the machine, the manufacturer has made machine- and user-specific adaptations.

O9679	Final actions, retracting strategy, fault management, tool change
O9671... O9677	Machine related definitions, axis attribution, basic settings, configuration for calibrating programme
O9678	Setting of zero point offset
O9670	Preliminary actions, approach strategy, axis limit switch skip

Note: Other programmes, not described herein, may be required to guarantee proper operation of the measuring cycles. Please refer to the documentation supplied by the machine manufacturer.

3.2 Error messages

At disturbances during the process, an error message is given. The macro variable VC100 contains an error status.
Keys:

User reserve code = Error number

= Check the following points

= Error description

= Remedy / Workaround

1	**“Laser not ready”**
	The intensity of the laser beam at the receiver is too low; measurement is not possible.
	(1) Are the shutters open? (2) Are the lenses clean? (3) Are the laser beam and the “Laser ON” LED red? (4) Does PLC receive the signal ”Laser OK”?
	(1) Turn on PLC output for "shutter". Control air pipes. Adjust the barrier air pressure 0.2 MPa below the shutter pressure. (2) Remove the shutters and clean the lenses. (3) Turn on PLC output for ENABLE0. (4) Turn on PLC output for ENABLE0 and ENABLE1 on and deactivate ENABLE2. Control the connection.

2	**“Reserved”**
	not used

3	**"Incorrect table parameter"**
	The measuring cycle cannot be executed with the parameters set in O967x (USERPARATAB).
	Are the parameters in O967 (USERPARATAB) set correctly?
	Set all parameters correctly according to chapter 3.2 in set-up manual. Run the calibration cycle to get more detailed information.

4	"Incorrect call parameter"
	The measuring cycle cannot be executed with the set parameters.
	Are the parameters in the call up line correct? e.g. CALL O9608 PH=.. PD=.. PQ=.. PM=..
	Input correct values according to chapter 3.4 in user manual.

5	"Incorrect tool parameter"
	The tool data stored in the tool memory is faulty.
	(1) Is the tool length entered correctly if the length should be measured on the tool surface (PM =1 or =2) or if only the radius (PE=2) should be measured? (2) Are VC150 .. VC153 set correctly?
	(1) Enter correct tool length data. (2) Enter correct values for software limit switch according to chapter 3.2 of set-up manual.

6	"Collision risk with Laser"
	There is risk of collision with the set call parameters or tool data.
	(1) Is the measuring point too far away from the tool tip? (2) Is the tool radius larger than the interference distance (VC107 and VC108)? (3) Are the parameters in the call up line correct?
	(1) The measuring point must be within VC109. (2) Measure tools with a smaller radius as the interference distances! Choose larger system and adapt the values of VC107 and VC108 in O967x (USERPARATAB). (3) Set correct values.

7	"Tool data out of limit"
	Tool length or radius exceed the admissible limit.
	(1) Are VC110 .. VC113 set correctly? (2) Is the tool data exceeding the limit values (VC110] .. (VC113)?
	(1) Set all parameters according to chapter 3.2 of set-up manual. (2) The tool cannot be measured.

8	"Error at start measuring block"
	The skip signal has a wrong level. The measuring block cannot be executed.
	(1) Are the lenses clean? (2) Is NC Optional parameter 61 (rising or falling edge) set correctly? (3) Is the dynamic signal (VC177) assigned correctly? (4) Is the laser beam interrupted by coolant drops? (5) Is the correct tool data used for NT measurement?
	(1) Remove the pollution protectors and clean the lenses (glasses cleaning cloth). (2) Set NC Optional parameter 61 correctly. (3) Check assignment of signals in O967x(USERPARATAB). (4) Execute NT measurement (PB=3 set in the call). (5) Enter correct tool data in tool memory.

9	**"Measurement without trigger signal"**
	No skip signal has been detected during the measuring block (G31).
	(1) Are the lenses clean? (2) Is NC Optional parameter 61 (rising or falling edge) set correctly? (3) Is the dynamic signal (VC177) assigned correctly? (4) Is coolant or oil on the tool? (5) Is the laser beam interrupted by coolant drops? (6) Are VC104..VC106 in O967 (USERPARATAB) set correctly? (7) Are the calibration values VC[VC140+0] to VC[VC140+9] correct? (8) Is the tool shorter than the minimum tool length? (9) Is a measuring feed rate too fast? (10) Is the retracting stroke too short after the measuring approach?
	(1) Remove the pollution protectors and clean the lenses (glasses cleaning cloth). (2) Set NC Optional parameter 61 correctly. (3) Check assignment of signals in O967x (USERPARATAB). (4) Clean the tool. (5) Execute NT measurement (PB=3 set in the call). (6) Set parameter VC104..VC106 according to chapter 3.2 in set-up manual. (7) In VC[VC140+0] to VC[VC140+9] manual zero setting and execution of calibration cycle. (8) Set minimum tool length VC111 in O967x (USERPARATAB) correctly. (9) Reduce measuring feed VC119 in O967x (USERPARATB). (10) Enlarge the values of VC122..VC124 in O967x (USERPARATAB) step by step (x1.5, x2, x3, …).

10	**"Deviation of meas. values > limit"**
	The dispersion of the single measuring values exceeds the maximum value.
	(1) Are the lenses clean? (2) Is the laser rigidly fixed? (3) Is coolant or oil on the tool? (4) Is the laser beam interrupted by coolant drops? (5) Is call parameter R too small?
	(1) Remove the pollution protectors and clean the lenses (glasses cleaning cloth). (2) Mount the laser rigidly. (3) Clean the tool. (4) Execute NT measurement (PB=3 set in the call). (5) Enlarge R step by step (R=0.005, R=0.01, R=0.015, …).

11	**"Thermal drift > limit"**
	The measured temperature drift of an axis is out of tolerance.
	(1) Are parameters S or T in the programme too small? (2) Is the machine temperature drift unusual? (3) Has a reference measurement (PB=0) been executed before?
	(1) Enlarge values for S and T. (2) Fix the mechanical error (check levelling of the machine, etc.). (3) Execute reference measurement (PB=0).

12	**"Error during evaluation"**
	A fatal error has been detected while evaluating the measuring data, e.g. division through zero.
	Are there any differing values in O967x (USERPARATAB)?
	Set all parameters correctly according to chapter 3.2 in set-up manual.

13	**"Incorrect spindle RPM"**
	Spindle rpm for cutting edge control is wrong.
	(1) Is spindle override at 100%? (2) Is the programmed spindle speed too low?
	(1) Set spindle override to 100%. (2) The spindle speed must be larger than S=3750 / PC (number of cutting edges).

14	**"Incorrect calibration-/TC-parameter"**
	The reference tool data is wrong or the calibration / TC values differ too much from the measuring position in the calibration cycle. There is risk of collision.
	(1) Are the calibration values VC[VC140+0] to VC[VC140+9] changed or replaced? (2) Is the difference between VC[VC140+0], VC[VC140+1] or VC[VC140+4] and VC106 smaller than 2mm? Is the difference between VC[VC140+2], VC[VC140+3] or VC[VC140+5] and VC104 smaller than 2mm? (3) Is the reference data VC126 .. VC128 correct?
	(1) In VC[VC140+0] to VC[VC140+9] manual zero setting of macro variables and execution of calibration cycle. (2) Set VC104 and VC106 in O967x (USERPARATB) correctly according to chapter 3.2 in set-up manual. (3) Set correct reference tool data according to chapter 3.2 in set-up manual.

15	**"Cutting edge broken"**
	A broken cutting edge was detected at the tool.
	(1) Is the cutting edge broken? (2) Is call parameter PC set correctly? (3) Is spindle override at 100%? (4) Are call parameters PQ and PZ set correctly? (5) Is there an unusual run-out? (6) Is coolant or oil on the tool? (7) Are VC132, VC133..VC136 correct?
	(1) Replace tool or insert. (2) Set the correct cutting edge number. (3) Set spindle override to 100%. (4) Enlarge the call parameter PQ and set PZ correctly. (5) Remove chips from the tool holder / clean contact surfaces. (6) Clean the tool. (7) Set parameter VC132, VC133..VC136 according to chapter 3.2 in set-up manual.

16	**"Out of tolerance"**
	The measured tolerance exceeds the limit.
	(1) Is the tool worn or broken? Was the tool drawn out of the holder? (2) Are chips sticking at the cutting edge?
	(1) Replace the tool or mount it correctly. (2) Remove the chips.

17	**"VC129 invalid"**
	The value of the variable is out of the valid range (1 to 7).
	Check the value of VC129 in USERSTARTPROG.
	Set VC129 according to chapter 3.2 in set-up manual. Default value: VC129=1.

18	**"Incorrect scale unit G20/G21"**
	The cycle was not called in the machine measuring system.
	Call up line/check the programme.
	Use the correct unit.

19	**"Tool broken"**
	The cycle was not called in the machine measuring system.
	(1) Is the tool broken? (2) Is H number correct? (3) Are call parameters PQ, PS or PT too small?
	(1) Replace tool. (2) Set correct H number. (3) Enlarge the values. For tool breakage cycle PQ should be 0.5mm or larger.

20	**"Function not possible"**
	The hardware does not match to the option.
	(1) Is Laser NT function required? (2) Are option bits VC154..VC157 set correctly? (3) Is call parameter PB negative without Laser NT installed?
	(1) Install Laser NT hardware. (2) Set VC154..VC157 according to chapter 3.2 in set-up manual. (3) Set call parameter PB to 0, 1 or 2.

24	**"Incorrect error number"**
	The programme for error messages has been called with an invalid error number.
	Internal use only.

-66	**"Incompatibility issue"**
	Control error in case of angular installation of the laser.
	Internal use only. Check page 31 of this manual.

-99	**"Skip function deactivated"**
	The Skip function of the control is deactivated.
	Internal use only.

RESERVED	**“Reserved”**
	not used

3.3 List of macro variables

Macro variables, which have to be defined in O9670 (USERSTARTPROG):

VC129	Choose between O9671 (USERPARATAB) (=1) ... O9677 (USERPARATAB7) (=7)

Common variables, which have to be defined in O967x (USERPARATABx):

VC101, 1 digit	Axis number for tool radius measurement
VC101, 10 digits	Axis number of laser beam axis
VC101, 100 digits	Axis number for tool length measurement
VC102	Side of approach in the length axis (+1 or -1)
VC103	Side of approach in the radius axis (+2,+1 or –1, –2)
VC104	Measuring position of the axis for tool radius measurement
VC105	Measuring position of the laser axis (position between transmitter and receiver)
VC106	Measuring position of the axis for tool length measurement
VC107	Distance between laser beam and interfering contour in tool radius axis
VC108	Distance between laser beam and interfering contours in laser beam axis
VC109	Distance between laser beam and interference contour in tool length axis
VC110	Maximum tool length
VC111	Minimum tool length
VC112	Maximum tool radius
VC113	reserved
VC114	Tool length offset (option for post processor applications)
VC115	Tool radius offset (option for post processor applications)
VC116	Real length of “zero tool”
VC117	Rapid speed
VC118	Positioning speed
VC119	Maximum measuring speed
VC120	Retraction level in the tool length axis prior measuring cycle
VC121	Retraction level in the tool length axis after measuring cycle
VC122	Distance before tool edge
VC123	Retracting travel after first measuring block
VC124	Retracting travel after second measuring block
VC125	Retracting travel after the following measuring blocks
VC126	Length of reference tool (L+H) or measured length for diagnosis
VC127	Radius of reference tool (R) or measured radius for diagnosis
VC128	Height of reference tool (H) or measured 2nd radius for diagnosis
VC130	Measuring speed1
VC131	Measuring speed2
VC132	Pulse time of “Output DYNAMIC” of the laser system
VC133	Basic speed of laser system for MODE0
VC134	Basic speed of laser system for MODE1
VC135	Basic speed of laser system for MODE2
VC136	Basic speed of laser system for MODE3
VC138	Type of reference tool (type 0, type 1, type 2), performing radial measurements of the reference tool one-sided/both-sided or measuring mode for diagnosis
VC139	Measuring direction for calibration and temperature compensation measurement

Common variables, which have to be defined in O967x (USERPARATABx):

VC140	Basic address for calibration parameter block
VC141	Number of trials (2, .., 10)
VC142	Feed rate at the end of cycle
VC150	Positive software limit switch of the tool length axis
VC151	Negative software limit switch of the tool length axis
VC152	Positive software limit switch of the tool radius axis
VC153	Negative software limit switch of the tool radius axis
VC154	Option bits for laser system
VC155	Option bits for measuring cycles
VC156	Option bits for temperature compensation, allocation of tool correction
VC157	Option bits (reserved)
VC162	Angle deviation laser beam to tool length axis
VC163	Angle deviation laser beam to tool radius axis
VC170	Number of PLC output for signal "ENABLE0" or M-Code
VC171	Number of PLC output for signal "ENABLE1" or M-Code
VC172	Number of PLC output for signal "Valve Shutter" or M-Code
VC173	Number of PLC output for signal "Valve barrier air" or M-Code
VC174	Number of PLC output for signal "ENABLE2" or M-Code
VC175	Number of PLC output for signal "Valve tool cleaning nozzle" or M-Code
VC176	Number of PLC input for signal "LASER OK"
VC177	Number of PLC input for signal "DYNAMIC"
VC178	Number of PLC input for signal "STATIC"
VC180	Language code for NC error messages
VC181	Definition of mode of nozzle usage
VC182	Distance between laser beam and interfering contour in front of laser beam in tool radius axis
VC200	SSU unit connector used for "DYNAMIC" input signal

Common variables, which are automatically set by the program:

VC77	Parameter to enable the sensor
VC78	Parameter to check the sensor status
VC100	Error text number, 0=no error
VC137	Current programme number (1..29,81..89.99)
VC[VC140+0] VC[VC140+1] VC[VC140+2] VC[VC140+3] VC[VC140+4] VC[VC140+5] VC[VC140+6] VC[VC140+7] VC[VC140+8] VC[VC140+9]	1st calibrating value of tool length axis (front edge) 2nd calibrating value of tool length axis (rear edge) 1st calibrating value of tool radius axis (standard side) 2nd calibrating value of tool radius axis (option side) Mean value of calibrating values in tool length axis Mean value of calibrating values in tool radius axis TC-reference value in tool length axis TC-reference value in tool radius axis Angle ≙ VC162 Angle ≙ VC163 / position laser axis (≙VC105)
VC143	Spindle rpm and rotation direction at programme call
VC144	Scaling factor, depending on unit system of the machine 1=mm, 25.4=inch
VC145	Tool offset code=H*1000+D*1
VC146 VC147 VC148 VC149 VC158 VC159	Length (drilling or milling tool) or length L1 (turning tool) Length wear (drilling / milling tool) or wear L1 (turning tool) Radius Wear radius Tool length L2 (turning tool) Wear L2 (turning tool)
VC161	Tool cutting edge position
VC164 VC165 VC166	Radius position in tool coordinates Laser position in tool coordinates Length position in tool coordinates
VC168	Trigger-point of measuring axis
VC169	Active tool length for internal calculation of position
VC185..VC192	Internal variables (e.g. for circle calculation)
VC193	Counter for temperature compensation values (option)
VC194 VC195 VC196 VC197 VC198 VC199	1st temperature compensation value of tool length axis (Option) 1st temperature compensation value of tool radius axis (Option) 2nd temperature compensation value of tool length axis (Option) 2nd temperature compensation value of tool radius axis (Option) 3rd temperature compensation value of tool length axis (Option) 3rd temperature compensation value of tool radius axis (Option)

Macrovariables, which have to be defined in O967x (USERPARATAB)
(only at measurement with angle head not equal to 0°)

VC70	Distance in radius direction from laser beam to positioning of the swivel axis
VC71	Transformation angle
VC72	Angle of swivel head after measurement
VC73	Angle of swivel head to measurement
VC74	Head kinematics radius axis
VC75	Head kinematics laser axis

VC76	Head kinematics length axis
VC79	Axis number of swivel head

3.4 List of call parameters

Parameter	Description
PA	**Cycle O9601, O9602, O9603, O9604, O9606: Number of repeated measurements** Repeated measurements are made to establish an average. Value range: 1 <= PA <= 10, standard value: PA = 3 **Cycle O9605, O9685: Number of measuring points for thread mills** The specified numbers of tooth rows on taps are checked for broken cutting edges. Value range: 0 < PA, default value: PA = 0
PB	**Cycle O9602, O9603, O9606: Measuring mode** Defines, whether a tool shall be measured, whether a measured tool shall undergo a wear check or whether it has just to be evaluated. The approach to the laser beam is made with a search run (values 0, 1, 2, 3) or at rapid traverse feed (values –1, –2, –3, –11, –12, –13). –3=NT measurement –2=NT checking if tool is within tolerance, tool data are not modified –1=NT verification, measured geometric difference is written into wear 0=Measurement (=default value) +1=verification, measured geometric difference is written into wear +2= checking if tool is within length tolerance, tool data are not modified +3=NT measurement Additionally for grinding tools -13=NT measurement -12=NT checking if tool is within tolerance, tool data are not modified -11=NT verification, measured geometric difference is written into wear For diagnostic purposes PB is stored in VC138. **Cycle O9604: Measuring mode** Calculating mode for the result of thermal compensation 0=Reference measurement (standard value) 1=Comparative measurement **Cycle O9685, O9688: Position before the cycle call** Parameter PB is the position of the tool tip in front/above (positive value) or behind/below (negative value) the laser beam (interim position). If PB is not programmed, the tool is positioned on the retract position (VC120). If there are no interfering contours between the actual position and the laser beam, the start position (PB= -PZ-value for O9685 or PB= –5*PQ-value for O9688) can be chosen as interim position.

Parameter	Description
PC	**Cycle O9603, O9605, O9607, O9685: Number of cutting edges** Using the number of cutting edges PC and the basic speed VC[133+x] (x=0 / 2 with Laser Control NT) or the time constant VC132 (laser system standard) checking speed S = VC[133+x] / PC is determined and set, if it is below the actual rpm. PC = 0 → no cutting edge monitoring PC = 1 → checking of the outer contour (single broken cutting edges are not detected!) PC > 1 → monitoring of all cutting edges to be within tolerance Value range: 0 ≤ PC Exceptions: Only O9603 (concentricity check on tool shaft), O9605 (monitoring of a saw blade): PC = -1 → no signal change in the traversing area (start position ... target position) O9603 only (measurement of the shortest cutting edge) PC < -1: Measurement (in mode NT) with reduced spindle speed Attention: **Set spindle rpm potentiometer to 100%, otherwise faulty measurement!** **Cycle O9606: Number of measuring points** The calculation of circle radius and position of the centre needs at least 3 measuring values. The influence of a single value can be lowered by increasing PC. Although the measuring time is extended then. Value range: 3 <= PC <= 20, standard value: PC=3
PD	**All cycles: Offset number for tool radius** Normally PD-Code and PH-Code are the same, and PD can be omitted in the cycle call. In exceptional cases PD-Code and PH-Code could be different. In these cases a caution stop (M0) is set. See chapter 3.2.3 (USERSTARTPROG) in Installation instruction manual. Value range: 1 <= PD <= 999 and -3 <= PD <= -1 Type1: D1…D999 -> PD=1…PD=999 Type2: DA -> PD=-1 (Only in P300M) DB -> PD=-2 DC -> PD=-3 **Attention:** The type of PH-Code and PD-Code has to be the same. i.e. HA+DC(type2) or H2+D7(type1). Never mix HA+D5(type2+type1). Otherwise the control shows an error.
PE	**Cycle O9603: Scope of measurement** 0= length- and radius measurement (=default value) 1= length measurement only 2= radius measurement only -1=only length measurement inside contour (option) -2=only radius measurement inside contour (option) -3= length- and radius measurement inside contour (option)
PF	**Cycle O9605, O9607, O9685: Checking feed rate** During the single cutting edge monitoring the axes are moved at the specified feed rate. The feed rate determines the pitch width for the cutting edge monitoring. Example: Feed rate=100 mm/min, check rpm (PC=3, VC133=3750) =1250 min^{-1} → pitch width = 100 / 1250 mm= 0.08 mm, i.e. each cutting edge is checked at a resolution of 0.08 mm. Value range: 1 mm/min ≤ PF ≤ 1000 mm/min, standard value: PF = 100 mm/min

Parameter	Description
PH	**All cycles: Offset number for tool length** Normally PH-Code and T-Code are the same. If T-Code and type1 of H-Code matches, programming of PH can be omitted in the cycle call. In exceptional cases T-Code and PH-Code could be different. In these cases a caution stop (M0) is set. See chapter 3.2.3 (USERSTARTPROG) in Installation instruction manual. Value range: 1 <= PH <= 999 and -3 <= PH <= -1 Type1: H1…H999 -> PH=1…PH=999 Type2: HA -> PH=-1 (Only in P300M) HB -> PH=-2 HC -> PH=-3 **Attention:** The type of PH-Code and PD-Code has to be the same. i.e. HA+DC(type2) or H2+D7(type1). Never mix HA+D5 (type2+type1). Otherwise the control shows an error.
PI	**Cycle O9606: Starting angle to tool symmetric axis** To prevent imprecise circle data and to be sure to reach a measuring point in the circle shape a starting angle is always needed. Using mills with a convex shape (ball end mill, torus mill) and with extended run-out, the angle must be greater than 0°. Milling with concave shaped tools (e.g. quarter shaped mill) and with extended run-out, the angle must be greater than 180°. Value range: 0° ≤ PI ≤ 360° Default value: PI = 0° **Cycle O9607: Starting angle to tool symmetric axis** In the case of ball nose mills, there is usually only one cutting edge outside the tool centre, the other entire cutting edges end before the tool centre. The cutting edge monitoring can only start from the starting angle PI from which all cutting edges are captured while they rotate. The starting angle PI is the initial point for the circular path section. PI = 0° → e.g. cutting edge monitoring on facing mills with corner radius, torus mills PI > 0° → e.g. cutting edge monitoring on ball nose mills (PI < 90°) PI = 90° → e.g. cutting edge monitoring on tool shaft only, not on cutting edge radius. Value range: 0° ≤ PI ≤ 360° Default value: PI = 25°
PJ	**Cycle O9606, O9607: Tool corner radius** In order to ensure a useful tool check, the tool corner radius should be PJ>=2 mm. In order to calculate the scan distance, the tool corner radius must be entered with the highest possible precision. Value range: -tool radius ≤ PJ ≤ tool radius
PK	**Cycle O9606: Target angle to tool symmetric axis** To prevent imprecise circle data and to be sure to reach a measuring point in the circle shape, a target angle is always needed. Milling with concave shaped tools (e.g. quarter shaped mill) and with extended run out, the angle should be lower than 360°. Value range: PI ≤ PK ≤ 360°, default value: PK = 0° **Cycle O9607: Target angle to tool symmetric axis** The cutting edge monitoring on a corner radius is made on the arc of a circle from the starting angle PI to the target PK. Target PK determines if a full circle is to be checked or a partial half or quarter circle. Target angle PK is the last point of the circular path section and also the initial point for the following tangential path section. PK = PI → e.g. for cutting edge monitoring on tool shaft only, not on corner radius PK = 60° → e.g. for cutting edge monitoring on ball nose mills with cone shaft 30° PK = 90° → e.g. for cutting edge monitoring on ball nose mills with cylinder shaft PK = 180° → e.g. for cutting edge monitoring on torus mills Value range: PI ≤ PK ≤ 360°, default value: PK = 90°

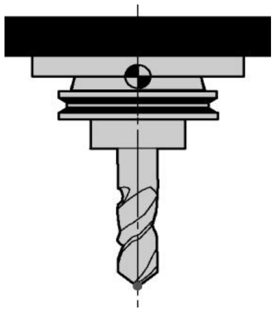
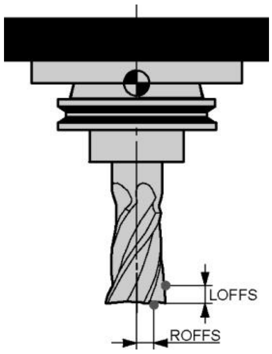
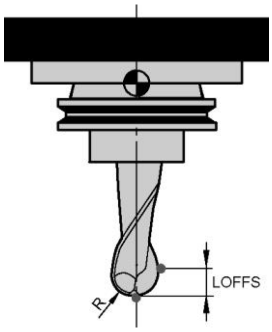
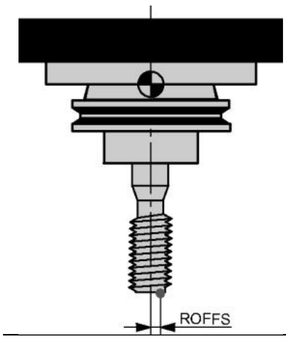
Parameter	Description
PM	**Cycle O9603: Special function** This parameter permits to take influence on the measuring process, i.e. radius measurement on one or two sides. Various measuring tasks are possible, i.e. length measurement at the front side of the tool (drill, end mill, etc.) or at the rear (disk milling cutter, slot mills, etc.). – 2= length measurement at the front tool edge and radius measurement on both sides – 1= length measurement at the front and radius measurement on one side (=standard) +1= length measurement at the rear tool edge and radius measurement on one side +2= length measurement at the rear tool edge and radius measurement on both sides Note: Radius measurement on both sides is only possible at set option of the manufacturer (defined by VC103=+/-2 in O967x (USERPARATABx)) **Cycle O9606, O9607: Position of cutting edge** The definition of the tool nose position follows the same principle as turning tools do, view picture in chapter 2.6. Tool length and radius in the tool memory are depending on the position of the cutting edge. Exception: In cycle O9607, PM=0 can be used because of compatibility reasons. Value range: 0 ≤ PM ≤ 9 **Cycle O9608, O9688: Measuring direction** The tool length measurement in tool breakage detection may be executed by pulling the tool out of the beam or by pushing it into the beam. In case of large quantities of coolant or for tools with a core diameter >∅ 1mm pulling measurement is recommended. Gravers or permanent error messages require the pulling measuring method. PM = +1 → measuring block from dark → light (=default value) PM = –1 → measuring block from light –> dark Only if option is set: PM = 0 → check if laser beam is interrupted by a tool, no measuring block. PM =+2 → appropriate only for very small tools (Ø < 0.5mm): in case of error at first radial search of the tool axis and then repeated breakage detection. Default value. PM = +1
PQ	**Cycle O9603, O9605, O9607, O9685: Admissible run-out tolerance** The cutting edge monitoring consists of a verification if the diameter of all cutting edges is within the admissible run-out tolerance with respect to the largest measured cutting edge. If the tolerance is exceeded, an error parameter is set. Value range: 0.000 mm < PQ, standard value: PQ = 0.050 mm **Cycle O9606: Radial offset from centre of circle to tool axis** If an unknown tool is measured, the position of the circle centre up from the tool axis has to be given. Using PB=-1,-2,-3 this parameter is not needed. **Cycle O9608, O9688: Admissible length tolerance** Tool breakage detection executes a rough tool length measurement and compares the value with the value from the tool memory. The tool is declared as "broken" if the length difference Delta = measured value – reference value is out of the permitted range: –PQ ≤ Delta ≤ 2* PQ Value range: 0.000 mm ≤ PQ ≤ 3000 mm, default value: PQ = 1000 mm

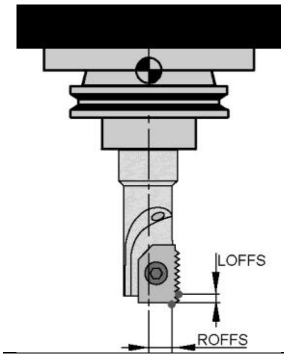
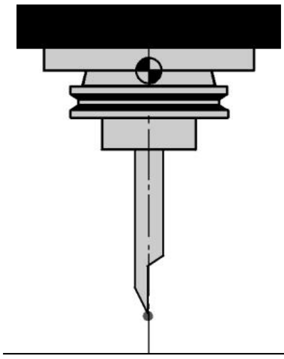
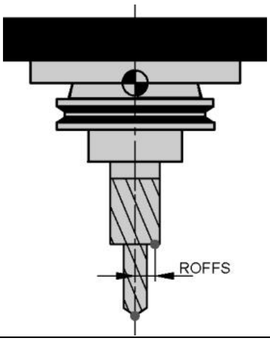
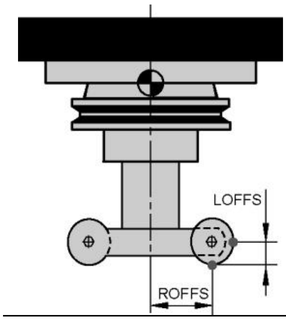
Parameter	Description
PR	**Cycle O9601, O9602, O9603, O9604, O9606: Maximum deviation of measured values** For the specified repeated measurement (PA>1) the difference between the biggest and the smallest measuring value is determined and compared to the deviation tolerance. If a value is out of this tolerance, a message is edited and the cycle is interrupted. Value range: 0.001 mm ≤ PR ≤ 0.100 mm, default value: PR = 0.010 mm
PS	**Cycle O9604: Length tolerance** At PB=1 the temperature drift will be determined as difference between the current and saved calibration value. When exceeding PS, error "E11 temperature drift > Limit" is given. Value range: 0.000 mm ≤ PS, default value: PS = 0.050 mm **Cycle O9602, O9603, O9606: Length tolerance** At PB=±1, PB=±2 and PB=±12 the tool wear will be determined as a difference between the measured length value and the given nominal value. If the value is out of the length tolerance, the tool is declared "out of tolerance". Value range: 0.000 mm ≤ PS, standard value: PS = 0.050 mm
PT	**Cycle O9604: Radius tolerance** At PB=1 the temperature drift will be determined as difference between the current and saved calibration value. When exceeding PT, an error "E11 temperature drift > Limit" is given. Value range: 0.000 mm ≤ PT, default value: PT = 0.050 mm **Cycle O9603, O9606: Radius tolerance** At PB=±1 and PB=±12 the tool wear will be determined as a difference between the measured radius value and the given nominal value. If the radius tolerance is exceeded, the tool is declared "out of tolerance". Value range: 0.000 mm ≤ PT, default value: PT = 0.050 mm
PU	**Cycle O9603, O9606: Additive radius offset** The additive radius offset value is added to the determined tool radius wear. This parameter can be used, i.e. when high-end surfaces are milled with an allowance. Moreover, radius variations can be compensated due to different cutting edge geometries. 2*PU must be smaller than VC122, otherwise error E4 is displayed. Default value: PU = 0.000 mm

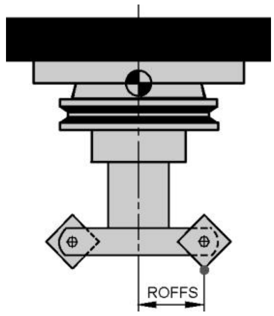
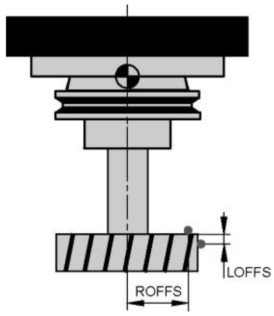
Parameter	Description
PV	**Cycle O9605, O9685: Scan distance or thread pitch** PV=0: The cutting edge monitoring starts at the PZ distance behind the tool tip. With PC>0 without any travel movement in position PZ, the check is executed whether all cutting edges interrupt the laser beam. At PC=-1 the tool must interrupt the beam continuously. PV≠0, PA=0 and PC>0: continued checking of a cutting edge section (i.e. end mill). In order to check the radius tolerance of all cutting edges, the tool length axis moves continuously into the laser beam by the traverse path (PV). PV≠0, PA=0 and PC=-1: Continuous check of an area (e.g. shaft or saw blade). The beam must continuously be interrupted in the whole area. PV≠0, PA>0: Step by step checking of a cutting edge section (i.e. taps). The radius tolerance is checked at the starting position (i.e. first tooth row on the tap). Depending on the number of measuring points (PA) all the following tooth rows are checked for broken cutting edges, offset by the pitch width (PV). Value range: 0.000 mm ≤ PV, default value: PV = 0.000 mm **Cycle O9607: Checking stroke on shaft/cone** The cutting edge monitoring starts at the tool tip and ends at the tool shaft. In the first section a circular path is described on the corner radius, in the second section a following straight tangential movement on the cylinder or cone shaft. The traverse path (PV) determines the length of the second section. PV = 0 → cutting edge monitoring on cutting edge radius only, not on the tool shaft PV > 0 → cutting edge monitoring on cutting edge radius and on the tool shaft Value range: 0.000 mm ≤ PV, default value: PV = 0.000 mm
PW	**Cycle O9602, O9603, O9606: Additive length offset** The additive length offset value is added to the determined tool length wear. This parameter can be used, i.e. when high-end surfaces are milled with an allowance. Moreover, length variations can be compensated due to different cutting edge geometries. 2*PW must be smaller than VC122, otherwise error E4 is displayed. Default value: PW = 0.000 mm **Cycle O9608, O9688: Additive length offset** The additive length offset value is added to the measured tool length. This parameter should be used if tools are generally measured “wrong” and declared “broken” (e.g. with milling cutters which are "free" in their core). Default value: PW = 0.000 mm

Parameter	Description
PX	**Cycle O9601, O9604: Radial measuring position for length measurement** At PX = 0 (recommended), a suitable default value will be used depending on the type of the reference tool. Further notes: See chapter calibration. Value range: 0.000 mm ≤ PX < tool radius, standard value: PX = 0.000 mm **Cycle O9603, O9606: Radial measuring position for length measurement** On tools like end mills, facing heads, etc. the tool length is measured outside the centre on the cutting edges. The position for length measurement is the radial offset from the measuring position to the tool axis: Default value: PX ≈ tool radius – corner radius – 0.5 mm **Cycle O9607: Radial offset for the start of the corner radius** Distance between tool axis and start at the corner radius. Value range: 0.000 mm ≤ PX, default value: PX = 0.000 mm PX = 0 → corner radius starts in tool axis (e.g. ball nose mill) PX > 0 → corner radius starts outside the tool axis (e.g. torus mill) **Cycle O9608, O9688: Radial checking position** On tools with inside-tool-coolant supply, i.e. taps with inner cooling, the tool breakage detection is performed off the source of disturbing influences. The checking position is located within the tool core and it is the lateral offset to the tool axis. Value range: 0.000 mm ≤ PX < tool radius, standard value: PX = 0.000 mm
PZ	**Cycle O9601, O9604: Axial measuring position for radius measurement** At PZ = 0 (recommended) a suitable default value will be used depending on the type of the reference tool. Further notes: See chapter calibration. Default value: PZ = 0.000 mm **Cycle O9603: Axial measuring position for radius measurement** The position for radius measurement is the length offset of the measuring position at the tool cutting edge to the tool tip. In order to make sure that the laser beam is darkened, it is essential that the tool radius of tools like end mills, facing heads, etc. is measured behind the tool tip. If, on a disk milling cutter the rear tool edge is measured, a negative value has to be entered for PZ. Default value: PZ ≈ tool corner radius + 0.5 mm **Cycle O9605, O9685: Axial starting position of the control** Make sure that the laser beam is covered up in start position. Default value: PZ ≈ tool corner radius + 0.5 mm **Cycle O9606: Length offset from centre of circle to tool length** If an unknown tool is measured, the position of the circle centre up from the tool length has to be given. Using PB=-1,-2,-3 this parameter is not needed.

3.5 Measurement and check of different type of tools

Tool type	Measurement	Check
Drill 	**CALL O9602 PB=0** Result (in tool memory): ⇒ tool length	**CALL O9608 PM=1** Result (in VC100): ⇒ Tool OK (VC100=0) ⇒ Tool broken (VC100=19)
End mill 	**CALL O9603 PB=0 PE=0 PM=–1 PZ=.. PX=..** PX= (tool radius – tool corner radius – 0.5 mm) PZ=Tool corner radius + 0.5 mm Result (in tool memory): ⇒ tool length, tool radius	**CALL O9605 PC=.. PQ=..** PC=1 ⇒ PQ ≈ 0.01 mm PC=n ⇒ PQ ≈ 0.1 mm (tool radius is read) Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)
Ball nose mill 	**CALL O9603 PB=0 PE=0 PM=–1 PZ=.. PX=..** PX=0 (possible problems if the hole is in the centre) PZ=tool corner radius + 0.5 mm Result (in tool memory): ⇒ tool length, tool radius	**CALL O9607 PC=.. PQ=.. PX=0** PC=1 ⇒ PQ ≈ 0.01 mm PC=n ⇒ PQ ≈ 0.1 mm Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)
Tap 	**CALL O9602 PB=0** Result (in tool memory): ⇒ tool length	**CALL O9608 PM=1** Result (in VC100): ⇒ Tool OK (VC100=0) ⇒ Tool broken (VC100=19)

Tool type	Measurement	Check
Thread mill  	**CALL O9603 PB=0 PE=0 PM=–1 PZ=.. PX=..** Result (in tool memory): ⇒ tool length, tool radius	**CALL O9605 PC=.. PQ=.. PA=..** PC=1 ⇒ PQ ≈ 0.01 mm PC=n ⇒ PQ ≈ 0.1 mm (tool radius is read) Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)
Engraving mill 	**CALL O9602 PB=0** Result (in tool memory): ⇒ tool length	**CALL O9605 PC=1 PQ=0 PZ=..** PZ ≈ 0.2 ... 0.5 mm (Enter tool radius ≈ 0.2 mm!) Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)
Step drill / Step mill  	**CALL O9603 PB=0 PE=1 PM=–1 PZ=0 PX=..** PX=(Tool radius – 0.5 mm) Result (in tool memory): ⇒ tool length of 2nd cutting edge	**CALL O9605 PC=.. PQ=..** PC=1 ⇒ PQ ≈ 0.01 mm PC=n ⇒ PQ ≈ 0.1 mm (enter tool radius!) Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)
Insert tooth mill  	**CALL O9603 PB=0 PE=1 PM=–1 PZ=.. PX=..** PX= (Tool radius – corner radius – 0.5 mm) PZ=(tool corner radius) Result (in tool memory): ⇒ tool length, tool radius	**CALL O9607 PC=.. PQ=..** PC=1 ⇒ PQ ≈ 0.01 mm PC=n ⇒ PQ ≈ 0.1 mm Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)

Tool type	Measurement	Check
Insert tooth mill 	**CALL O9603 PB=0 PE=1 PM=–1 PZ=0 PX=..** PX= (Cutting edge position – 0.5 mm) Result (in tool memory): ⇒ tool length (Tool radius =PX enter manually in tool memory!).	**CALL O9605 PC=.. PV=0 PZ=..** PZ ≈ 0.2 ... 0.5 mm (Enter tool radius!) Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)
Disk milling cutter 	**CALL O9603 PB=0 PE=0 PM=1 PZ=.. PX=..** PX= (Tool radius – corner radius – 0.5 mm) PZ= - (tool corner radius) Result (in tool memory): ⇒ tool length, tool radius	**CALL O9605 PC=.. PQ=..** PC=1 ⇒ PQ ≈ 0.01 mm PC=n ⇒ PQ ≈ 0.1 mm (enter tool radius!) Attention: Spindle rpm is calculated automatically (S=VC133/PC) Result (in VC100): ⇒ Cutting edges ok (VC100=0) ⇒ Cutting edge broken (VC100=15)

4. Service

focus on productivity

Headquarters:

Blum-Novotest GmbH
Kaufstr. 14
88287 Grünkraut, 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: