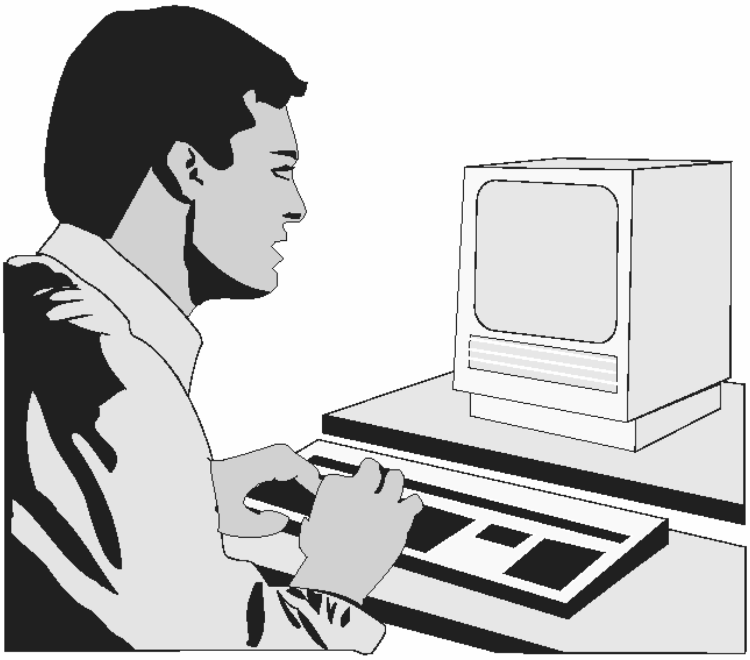
USE



INDEX OF CONTENTS:

END

INTRODUCTION

The JOBS-INSTRUCTION MANUAL describes the running and programming of the tool machine for what concerns the aspects settled by JOBS.
More precisely it integrates and completes the concepts concerning the use and programming provided in the CNC-manufacturer's User Manual.
For a correct use of the machine it is then necessary to take into consideration both the above mentioned documents.

- **CONTENTS OF THE JOBS- INSTRUCTION MANUAL**

  In this instruction manual you will find:

  - Description of the operator panel and of the other machine-push-button panels, with particular reference to the control drives introduced by JOBS in order to carry out the different running procedures.
  - The specific working procedures for the tool machine.
  - The "M" functions settled by JOBS for the programming.
  - Description of the operator-machine interface and MESSAGES.

  These concepts have been gathered in different sections :

  - MACHINE SECTION which concerns the machine's linear axis and the spindle, the hydraulic and electrical systems , the units and optional devices wihch the machine has been equipped.
  - TOOL CHANGE SECTION specific for the tool change system combined with the machine

  In order to facilitate the research of information concerning the auxiliary M-Functions and the alarms and messages, two other different sections have been scheduled .

- **SYMBOLOGY OF THE MANUAL**

  In this instruction manual some symbols and underlinings have been used in order to draw the attention of the reader to some particularly important or useful information.
  Herein these symbols are explained:

WARNING

It indicates a warning, an obligation, or a particularly important prohibition in order to avoid:

- Potential danger for people's safety;
- Damages to the machine.

NOTE

It draws the attention to a concept or a particularly interesting information.

INFORMATION

It points out the references to some sections of this manual or to other technical documentation.

AUTHORIZED PERSONNEL AND INTENDED USE

In this paragraph you will find some particularly important instructions concerning the INTENDED USE of the installation: the reader is therefore kindly requested to take note and to apply them carefully.

ATTENTION

The control of the installation during its production stages is to be carried out only by authorized skilled personnel , with an adequante technical experience and properly trained.

For thid reason it is nnecessary to know:

- **The contents of this instruction manual, along with the use and programming manual of the CNC's manufacturer.**
- **The contents of the other instruction manuals , with particular reference to those points concerning the safety standards.**

Non-skilled personnel must not enter the work area when the machine is working.

NOTE

It is important to remember that in all working conditions, the machine has been planned for just one operator .

Considering the dimensions of the installation and the manufacturing time, JOBS s.p.a. gives the possibility of two skilled operator who work together at close range and at the same time.

For safety reason, the interaction of three or more operators is forbidden .

NOTE

The working of the tool machine depends on the interaction of a great deal of subsystems (CNC, operator panel, drives and actuators, ect.): it is therefore useless and hard to describe all the procedures which cannot be carried out. The procedures or combination of working procedures which are not clearly marked as "SCHEDULED" either in this manual or in the technical documentation of the machine, should be considered as "NOT SCHEDULED" and so they have not to be tried .

ATTENTION

The imprudent or incorrect movement of the axes carried out by the operator, with following crash of the group ram-spindle against installaations or infrastructures positioned in the work-area can cause serious damages.

In particular, the damaging of the head-spindle assembly can lead to several downtime-days for the correct re-adjustment of the mechanical geometries.

END

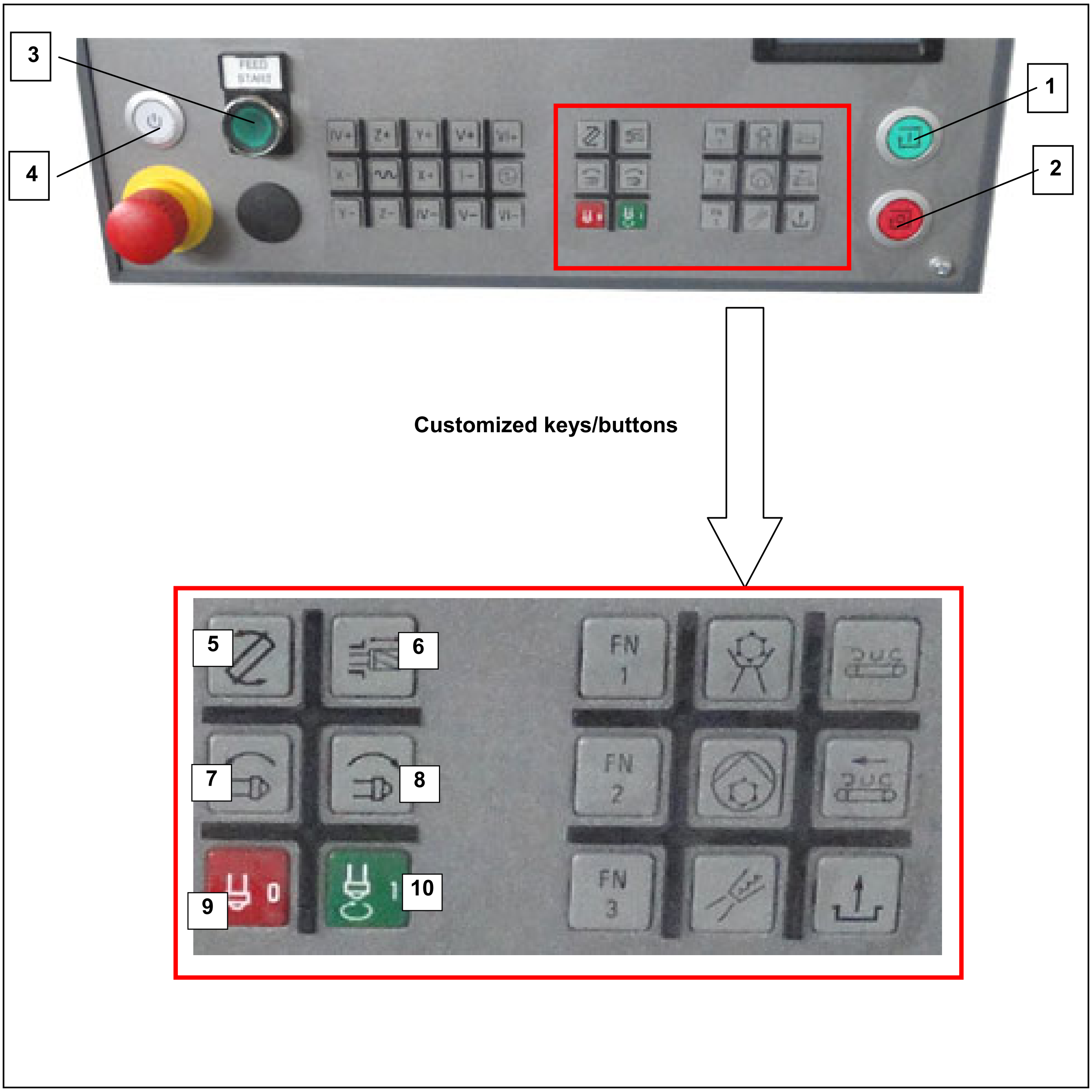
MACHINE PART

CONTROL UNIT (OPERATOR PANEL)

The control unit (IP65 protection level) allows the operator to control the machine operation during all machining stages.
It is composed of:

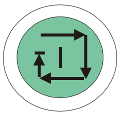
1. control panel with flat screen, 8 horizontal and 6 vertical softkeys
2. HEIDENHAIN machine keyboard
3. JOBS machine panel

The following picture shows the customized keys and buttons of the HEIDENHAIN machine keyboard

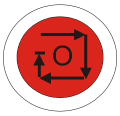


NOTE

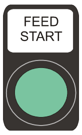
The detailed description of the operator panel and of the machine keyboard, as for the section settled by the CNC Manufacturer, can be found in the "OPERATION MANUAL" provided by HEIDENHAIN.

-  **START NC push-button (1)**

  This push-button allows the operator to start a program or a block of programs. The lamp light is ON when the program is running.

-  **STOP NC push-button (2)**

  This push-button allows the operator to stop a program or a block of programs.

- 

  FEED START push-button (3)

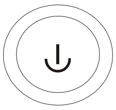
  This push-button allows the operator to enable the feeding of machine axes.

  Light off: machine in emergency.

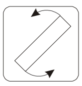
  Light steady on: Feed Start enabled.

  Light normally flashing: machine in hold axes.

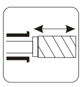
  Light quickly flashing: machine in error status.

-  **POWER ON push-button (4)**

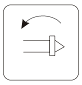
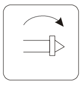
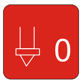
  This impulsive push-button allows the operator to switch on the machine. The white lamp light is steady ON when the operation is completed, it flashes when the machine is OFF.

-  **TOOL RELEASE ENABLING push button (5)**

  Press this button, to request the release of the tool from spindle. The spindle must be off (M5).
  Once enabled, the physical release of the tool will be carry out by pressing the “TOOL RELEASE”-button.
  After having replaced the tool, you have to disable the release condition by pressing the button again. Otherwise the spindle rotation is inhibited.

-  **TOOL RELEASE light push button (6)**

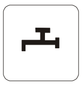
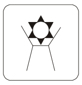
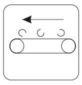
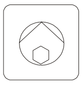
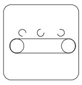
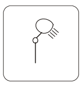
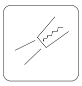
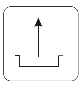
  Press this button, to actually release of the tool from spindle. If you press again this button, the tool is re-clamped. The LED is On when the tool is clamped, otherwise it is OFF.

-  **COUNTERCLOCKWISE SPINDLE ROTATION push button (7)**
-  **CLOCKWISE SPINDLE ROTATION push button (8)**
- 

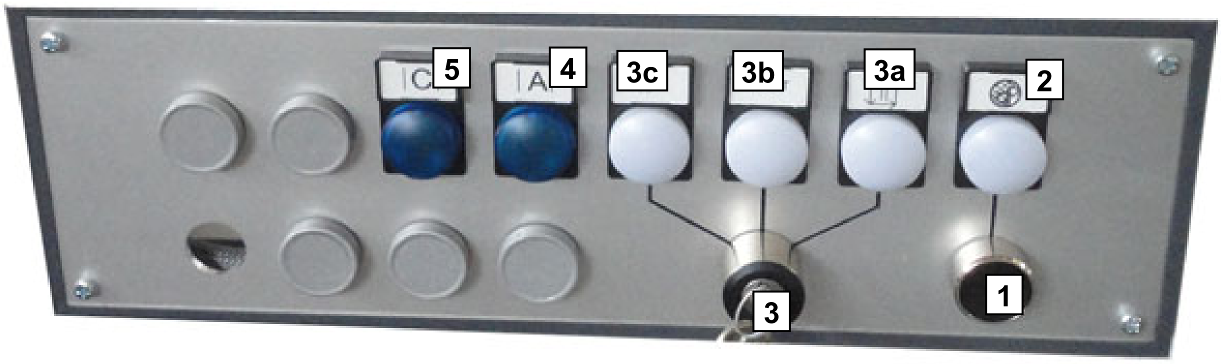
SPINDLE STOP push button (9)
- 

SPINDLE START push button (10)

The arrangement of the following buttons varies according to the machine configuration:

-  **COOLANT ACTIVATION push-button**
-  **AXES LUBRICATION push-button**
  This impulsive push-button allows the operator to lubricate the machine axes with intervals different from the lubrication automatic cycle.
-  **CHIP CONVEYOR BACKWARD activation push button**
-  **LUBRICATION activation push button**
-  **CHIPS CONVEYOR FORWARD ENABLING push-button**
-  **Workarea LIGHTING push-button**
-  **SPRAY MIST activation push-button**
-  **SUCTION activation push-button**

The picture below shows the JOBS-machine panel: the push buttons shown are those relevant to the MACHINE PART.



JOBS machine panel

INFORMATION

For the other customized push buttons, please refer to the specific sections (TOOL CHANGE, MACHINE OPTION, etc.) of this "JOBS OPERATING MANUAL".

- **1 – GUARD OPENING REQUEST push-button**

  This button is effective only when pressed with the Modal Selector Switch (3) set to Manual or Test position, and it allows you to unlock the interlocking device of the guard, thus allowing you to enter the work area and to operate in “limited performance” mode with DEAD MAN button pressed on portable keyboard.

  The lamp (2) lighting signals the door opening enabling.

- **3 – Operating mode selector switch**

  See next paragraph for a detailed description.

NOTE

For safety reasons, the mode selectors key must be entrusted only to authorized skilled staff.

- **4 – A AXIS LOCKED warning light**

  This lamp lights up when the A axis is locked.

- **5 – C AXIS LOCKED warning light**

  This lamp lights up when the C axis is locked.

OPERATING MODES: USING THE MODAL SELECTOR

The mode selector switch (with key that can be removed in any position) conforms to EN12417 safety standards for “Machining centers”. These standards establish a subdivision of machine operating modes to permit a certain degree of operational flexibility while simultaneously ensuring the operator safety. The machine can be operated in two different conditions:

- **Machine guards closed**: the operator cannot directly access the machine work area (i.e., enter, carry out measurements or checks, manually replace the tool on the spindle, etc.);
- **Machine guards open**: the operator may access the machine work area, making sure to observe particular operational procedures.

The mode selector is always combined with PNOZ-type electrical safety devices (which ensures a redundant hardware and software safety conditions).
In addition to the mode selector, there is also a safety push button (DEAD MAN button) which operates when the machine guards are open. If the operator does not keep this button continuously pressed, when it works with guards open, an emergency stop is activated.

INFORMATION

Please refer to the SAFETY section of the manual for further information about the DEAD MAN button operation.

The three possible operating modes are:

MODE 1. (MACHINE OPERATION WITH GUARDS CLOSED) (3a)

In this mode, the guards cannot be opened. In fact, with the guards open, the CNC forces the AXES HOLD and SPINDLE HOLD status and sets the machine to MANUAL mode, irrespective of the selector setting.
The DEAD MAN button is not effective.
All CNC functions can be utilised (AUTOMATIC, MDI, JOG, etc.).
This is the normal machine operating mode.

MODE 2. (MANUAL MACHINE OPERATION WITH GUARDS OPEN) (3c)

All machine movements must be carried out with the DEAD MAN button pressed, otherwise the spindle and axes are stopped cutting off the power to all drives and actuators (CNC in HOLD).
After releasing the DEAD MAN button, wait 1 to 2 seconds before restoring the machine cycle, due to the electrical delays of the safety circuit.
The CNC cannot be used in AUTOMATIC and MDI modes.
Since the MDI mode is forbidden, to rotate the spindle, follow the procedure below:

1. Close all doors and guards
2. Set the selector to MODE 1 (AUTOMATIC)
3. Select the CNC MDI mode and program M03 (M04) Sxxxx
4. Press the DEAD MAN button
5. Turn the selector to MODE 2 (MANUAL)
6. Open the mobile guard to access the work area, if necessary
7. Press FEED START and SPINDLE START

If the DEAD MAN button is released during this stage, the entire sequence must be repeated.
All other CNC operations are allowed, with the following restrictions:

- machine axes speed < 2 m/1';
- spindle speed < 2000 RPM.

TEST MODE 3. (PART-PROGRAM TEST WITH GUARDS OPEN) (3b)

All machine movements must be carried out with the DEAD MAN button pressed, otherwise the spindle and axes are stopped cutting off the power to all drives and actuators (CNC in HOLD).
After releasing the DEAD MAN button, wait 1 to 2 seconds before restoring the machine cycle, due to the electrical delays of the safety circuit.
The CNC cannot be used in AUTOMATIC mode.
All other CNC operations are allowed, within the following limitations:

- machine axes speed < 2 m/1';
- machine axes shift increments < 10 mm;
- spindle speed < 2000.rpm.
- SINGLE BLOCK forced;
- T functions issued and stored, but not carried out;

- M06 function inhibited at the beginning of the corresponding Macro.

To facilitate Program Test functions, the guards can be closed; this allows executing T and M06 functions.

Switching from one operating mode to another causes an immediate axes and spindle stop. Each operating mode is indicated by the corresponding lamp.

MODE SELECTOR AND GUARDS

The working of the modal selector and of the guards connected to the selector takes place in the way described below.

If the operator wants to open the guards in order to get into the work area and to use the machine in “limited performance” mode, he has to turn the modal selector to MANUAL position and press the push-button (1) “Door opening request”.

Under these conditions the PLC stops the axes and spindle in controlled mode; it releases the movable guard and lights the lamp (2) “Door opening enable”.

In MANUAL-mode and with guards open, in order to move the machine the operator shall keep the Dead Man-button pressed on the portable panel.

In this modes, the speed values established by the Harmonized Norm EN 12417 (2000 mm/min for axes and 2000 rpm for the spindle) must not be exceeded; otherwise, the DINA axes/spindle speed control device trips, and it sets the machine to emergency status.

AXES OVERRIDE

The axes feed override is managed by two selectors, one for the working feed and the other one for the rapid feed. (FMAX or G00). The two selectors work separately.

MACHINE PORTABLE KEYBOARD HR 410

On this keyboard you can find a number of remote working controls that allow the operator to keep control over the machine by visually monitoring the machining in progress.

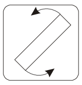
INFORMATION

For a description of the buttons and their use, please refer to the original documentation supplied by HEIDENHAIN

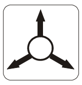
Mobile keyboard HR 410

Some of the buttons available have been customized by JOBS and repeat some of the drives located on the personalized pad of the main machine's keyboard.

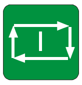
Below find a description of the customized JOBS-buttons.

-  **TOOL RELEASE ENABLING push button**

  Press this button, to request the release of the tool from spindle. The spindle must be off (M5).
  Once enabled, the physical release of the tool will be carry out by pressing the “TOOL RELEASE”-button.
  After having replaced the tool, you have to disable the release condition by pressing the button again. Otherwise the spindle rotation is inhibited.

-  **TOOL RELEASE push button**

  Press this button, to actually release of the tool from spindle. If you press again this button, the tool is re-clamped. The LED is On when the tool is clamped, otherwise it is OFF.

-  **NC start push button**

The keyboard is enabled through the button  located on the Heidenhain-push button-panel

The EMERGENCY-and DEAD MAN-buttons allow to work under complete safety.

WARNING AND SIGNALLING DEVICE

The machine is equipped with a 3-colour light device with siren that visually and acoustically warns the operator and any others in the vicinity of the work area (dangerous zone) of the machine operating status.

This device signals and their meaning are as follows:

SIGNAL	MEANING
STEADY GREEN LIGHT	Machine in regular automatic operating mode
FLASHING GREEN LIGHT	Machine in automatic (AUTO) or semiautomatic (MDA) operating mode (AUTO) but not in working cycle
STEADY YELLOW LIGHT	Running operation stopped, control in STOP
FLASHING YELLOW LIGHT	Running operation stopped, control in FEED HOLD
STEADY RED LIGHT	Standstill Machine in emergency, waiting for intervention
FLASHING RED LIGHT	Alarm and/or message active
SIREN WORKING	Machine logic message active and/or program start-up in automatic

OPERATING PROCEDURES

Before turning on the machine and preparing to operate, it is important to know the correct procedures for

- turning on the machine
- axis zero reset procedure
- axis re-entry following overrun
- machine emergency stop and power off
- door opening/closing
- spindle manual tool change
- spindle rotation
- restart of ka087 relay (three-phase supply line monitoring)
- polar axis locking.
- spindle preheating

TURNING ON THE MACHINE

The machine can only be turned on through the Control unit.

> **WARNING**
>
> **It is really important that the operator, who turns on the machine, makes sure that no person or object is within the machine's high-risk areas**

The correct procedure for turning on the machine is as follows:

- Check that all the machine auxiliary systems are connected correctly.
- Check that the head is not in contact with by the workpiece or worktable.
- Check that the upstream main switch is set to ON position.
- Check that all the machine mushroom-shaped emergency buttons are restored.
- Turn the main switch to ON position. The control unit screen should turn on.
- Close all doors and guards.
- Turn the POWER ON key located on the JOBS control panel and then release it.

> **NOTE**
>
> **The POWER ON key selector will not be immediately functional when the main switch is set to ON. In fact, it is necessary to wait the time required for the CNC to become completely operational.**

At this point the machine is ACTIVE, which means:

- electrical, hydraulic and pneumatic actuators are live;
- safety devices are active;
- access allowed only under safety conditions.

AXES ZERO RESET PROCEDURE

When the machine is turned off, the axis reference points are lost. If the axis zero reset procedure is not performed, the machine cannot measure shifts as required for automatic operation.

The ZERO SEARCH procedure, which must be performed each time the machine is turned on, permits automatic reset.

WARNING

Even if the axis zero reset procedure is not performed, it is still possible to move the axes in manual mode (using the dial or in JOG).

Please note that, since the axes travels' software limits are not active, if you move the axes in a wrong way, these latter may overrun their travel.

Select the JOG operating mode.

Remove the tool on the spindle following the instructions given in the section entitled "MANUAL TOOL CHANGE".

Using the JOG mode, place the axes in a position from which the zero reset point can be reached without collision's risks.

Axes' zero reset

Move to "Ratings Surpass' mode" (At the start of the machine, this is default-performed). Then a mask appears showing, in reverse and in decreasing order, the machine's axes for which the reference points' research is needed.

Pressing the "START CYCLE" button on the machine's keyboard, you carry out the zero of the first axis configurated. At the end, every time you press the "START CYCLE"-button, you carry out the zero of the next axis and so on , till the order of all the axes is finished.

Use the "FEED OVERRIDE" selector to make sure the axes move at low speed.

WARNING

In any case, the operator has to check the progress of the operation and make sure that it may be carried out without colliding with the workpiece or worktable.

Once the axes have been zero reset, the values displayed by the CN are the effective ones, the software travel limits are activated and the machine is ready to carry out programs in automatic mode.

AXIS RE-ENTRY FOLLOWING OVERRUN

The axis reset procedure activates both the positive and negative software limits for travel overrun in relation to the axis zero point. When the axes are moved in JOG mode, or when a part program is run, the axis limit stops cannot be exceeded (the machine displays an error message if an attempt is made to do so). However, this is clearly not the case if axis reset is not performed: in this case, the pre-set software limits can be exceeded if the machine is operated in JOG mode.

These software limits can also be exceeded using the electronic cranks which can manually override all the other controls.

When the software limits are surpassed it is called “axis overrun” and is considered an operating fault. The machine is protected from axis overrun through a second electrical protection device. Reversing direction does not suffice to resolve an overrun situation since a safety device is triggered and the machine enters EMERGENCY status.

Axis overrun emergency status is signalled by a message displayed on the screen, which identifies which axis is in positive or negative overrun emergency status.

To restore machine operation, the “axis overrun re-entry procedure” must be performed. This procedure requires two operators, one to maintain the emergency circuit in by-pass, and the other to perform the JOG manoeuvre to return the axis to its normal operating limits.

1 (1°Operator) Turn the by-pass key of the emergencies (SL100) control circuit and keep it turned. The by-pass key is located inside the electrical cabinet
2 Set the “FEED OVERRIDE” so that the axes move at low speed (a value of 10-20%)
3 (2°Operator) Turn the “POWER ON” key on the main control panel and, keeping it turned, move the relevant axis in JOG mode so that it comes back within the work travel limits.

WARNING

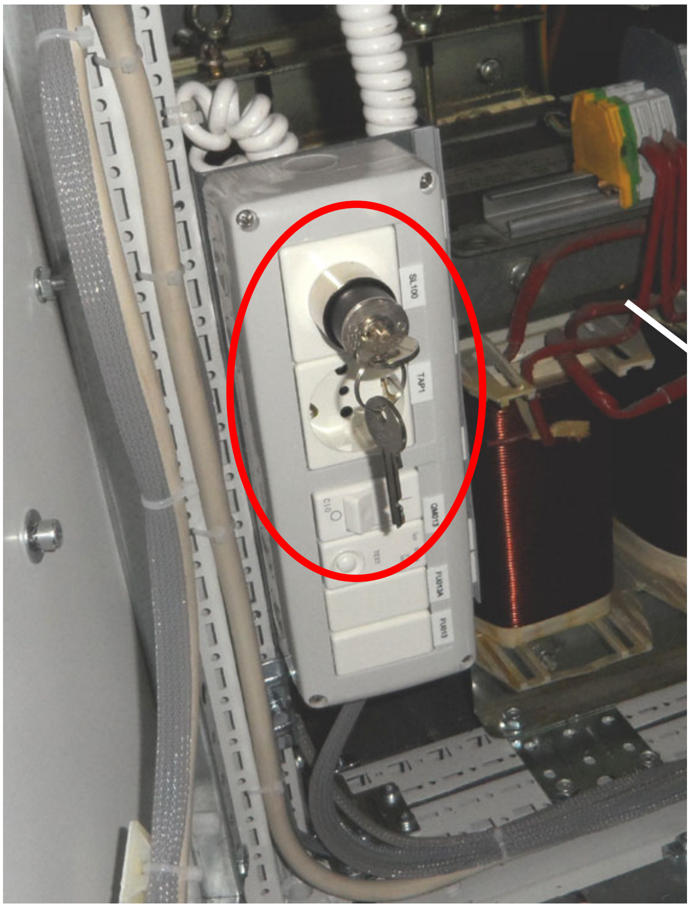
Take care during these operations because additional protection devices are not in operation and the axis could strain against the mechanical limit stop buffers.

The operator must also ascertain that the movements required for axis overrun re-entry may be performed without danger of collision

4. Once the axis has returned within its normal travel limits, the power on and PNOZ by-pass keys may be released; the POWER ON light on the control panel must remain lit. If the light goes off, repeat the above steps.
5. For machines equipped with the TCP option: if an overrun occurs when the TCP mode is selected, the latter must be disactivated before carrying out the steps described above

WARNING

In case of anomalous behavior, immediately release the SL100 selector, in order to set the machine to emergency status



Emergency by-pass key

MACHINE EMERGENCY STOP AND POWER OFF

The MACHINE EMERGENCY status is triggered when one of the EMERGENCY buttons on the machine is pressed.
This must be done:

- in the presence of a real or potentially dangerous situation (due to machine malfunction or operating error on the part of the operator or other factor)
- whenever the machine is not in use for short periods of time (to ensure maximum safety conditions for anyone coming near the machine).

At the end of a working shift or whenever the machine is not to be used for a fairly long period of time, it must be turned off and powered off using the main switch.

To correctly shut off the machine:

- the NC must not have programs running or on hold;
- the axes must be stopped and there must not be any ISO block in MDI mode;
- the spindle must be stationary.

Alternatively, before turning off the machine, one of the EMERGENCY buttons may be pressed to automatically activate shut-off conditions.
Only in this case the machine can be shut off by turning the handle of the main switch to “OFF”.

NOTE

If the machine does not shut off, interrupt power coming from the factory three-phase line.

OPENING/CLOSING OF THE MACHINE DOORS/GATES

The machine is equipped with a number of mobile protections (doors, gates, etc.) which allow potentially hazardous areas to be accessed safely.
There is a single condition applicable to all these protective measures: when the machine is in EMERGENCY status, the mobile protections can be opened and closed freely.

INFORMATION

This section examines the work area and operator station.

For positions relative to tool L/U and pallet, please refer to the chapter entitled "TOOL CHANGE PART".

Movable guards used:

- front door with opening controlled by interlock device;
- rear door with opening controlled by interlock device.

Interlock device management

As previously stated, the interlock devices are not active when the machine is in EMERGENCY status. Apart from this condition, the interlock devices operate as follows:

- Release

  To release, an authorising condition or a command is required.
  The authorising condition is when the machine is in either in MANUAL or TEST mode (please refer to the description found in the MODE SELECTOR AND GUARDS section on this chapter). If the machine is not in one of these two modes, the doors are not released and the machine displays the message that the request is pending "13: DOOR OPENING REQUEST".
  The command is given using the special key selectors mounted:
  - on the control panel (see the section entitled "CONTROL UNIT – JOBS MACHINE KEYBOARD")
  - on the push-button panel on the rear door (refer to safety guard opening push-button panel).

- Locking

  The locking is active when the machine works in AUTOMATIC mode.

Safety guard opening push-button panel (rear door)

- **1 – EMERGENCY STOP push button**

  A red mushroom-shaped button with yellow crown which, when pushed, puts the machine in emergency status by cutting the power.
  To deactivate this button, release it by turning it in a clockwise direction as indicated by the arrows on the button itself.

- **2 – DOOR OPENING REQUEST key selector switch**

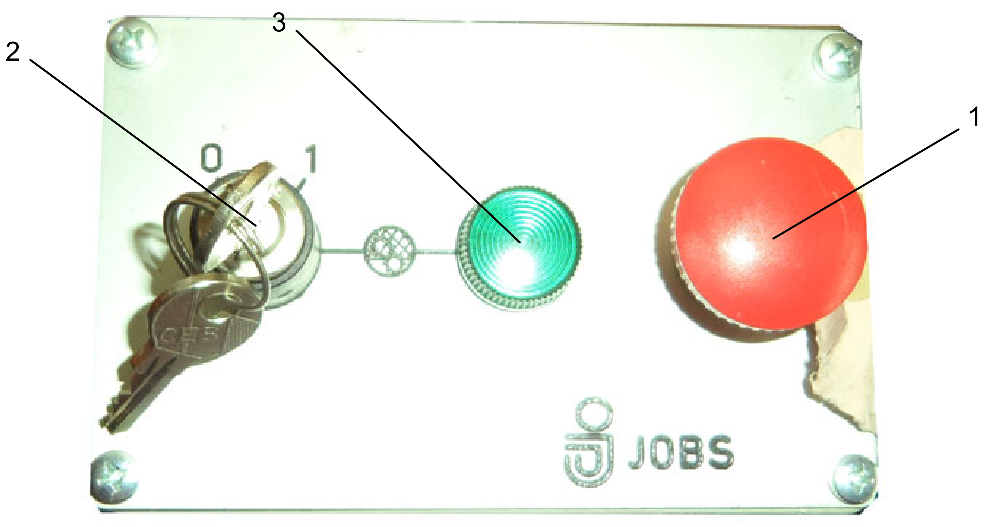
  Extractable-key selector switch: it executes the same function of the similar push-button on JOBS machine panel.

NOTE

For safety reasons, the keys must only be entrusted to personnel trained and authorised to work on the machine.

- **3 – DOOR OPENING REQUEST signal light**

  Lights up when the door opening is enabled (same function of lamp 2 on JOBS machine panel).



Safety guard opening push-button panel

MANUAL LOAD/UNLOAD OF TOOLS ON SPINDLE

> **WARNING**
>
> **The operator must scrupulously follow the information given in this section and be aware of the "residual risks" relevant to the above mentioned operation.**

To correctly load/unload the tools to/from the spindle, operate as follows:

1. If the spindle is rotating, program the M05 function (spindle stop) and wait for complete spindle stop.
2. In JOG mode, move the axes near the operator door so that the spindle can be accessed easily. If necessary, rotate axis A as required.
3. Turn the modal selector to the MANUAL MODE position
4. Open the operator's door by pressing the **"OPENING REQUEST"-** push button located on the JOBS machine keypad.
5. Grip the tool firmly with one hand and, with the other, press the **"TOOL RELEASE"** button on the HEIDENHAIN machine's keyboard.
6. The tool is released.

> **WARNING**
>
> **Be careful because, once the tool has been released, it is no longer held by the coolet and will fall if not held firmly by the operator.**
>
> **It is always recommended to wear gloves when handling the tool to avoid cuts and burns to the hand.**

7. Unload the old tool and firmly grasp the new one.

> **WARNING**
>
> **The tool must be held in such a way as to avoid the hand being crushed by the spindle jaws during the locking.**

8. Put the tool taper in its seat.

9. Press the “TOOL RELEASE” button: the tool will be locked. Press the TOOL UNCLAMPING ENABLING push button again.

WARNING

Manually check that the tool has been firmly locked on the spindle.

10. Close the door/s and put the machine in the normal working conditions.

NOTE

Be careful during the manual change of the tools on a machine with a tools’ magazine. In this case you have to update the “Tool Changer Information Table” (see paragraph “Displaying Data”, otherwise, at the next automatic tool change, an error is generated.

For the display and managing of the tools’ table on the CN, please refer to the HEIDENHAIN-“USER Manual”.

SPINDLE ROTATION

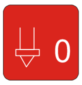
The spindle can rotate in both direction (either clockwise or counter-clockwise). The appropriate rotation speed must be selected depending on the machining type, material, tool and technical characteristics of the spindle.
The spindle direction and speed are programmed using the corresponding functions in MDI or inside the Part-Program.
The operator must, in any case, enable the operation using the “SPINDLE START” button.

Spindle rotation direction and spindle speed must be programmed using M03 (clockwise direction) or M04 (counter-clockwise direction) and “Sxxxx”, where xxxx is the desired rotation speed.

Programming the M19-function, you will obtain a spindle orientation to the automatic tool change position.

WARNING

Operating the spindle without a tool inserted is STRICTLY FORBIDDEN

- 

SPINDLE STOP button

Stop the spindle by pressing the “SPINDLE STOP” button or the red mushroom-shaped emergency button. Make sure that the spindle and the tool are completely stationary before touching them.

AXES LOCKING

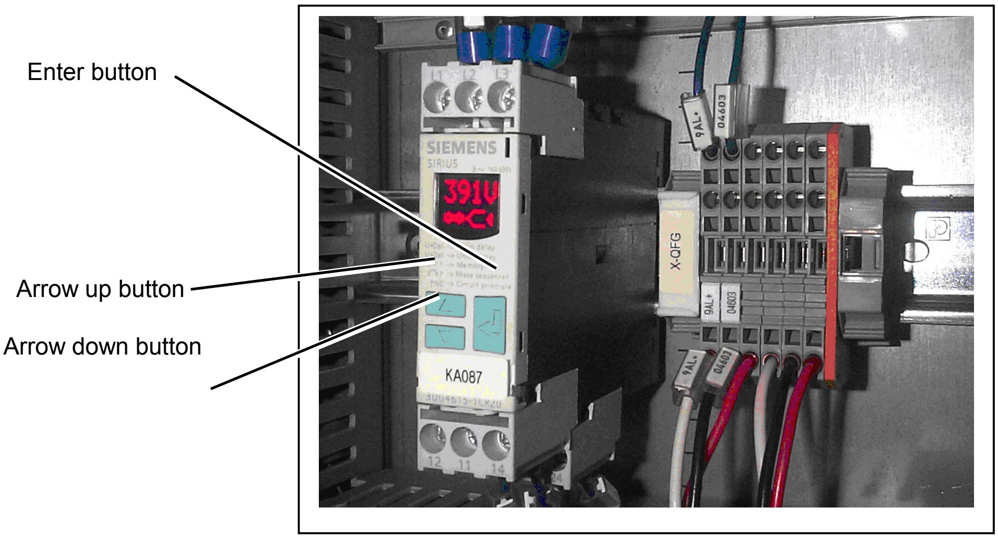
Polar axes A and C are equipped with a hydraulic movement locking system that enhances milling results by increasing the stiffness.
The activation/deactivation of the locking device can be controlled within the Part-Program using the appropriate M functions:

M61 : A axis lock
M62 : A axis unlock
M63 : C axis lock
M64 : C axis unlock

When function M61 or M63 is used, the corresponding axis is locked but all automatic and JOG movements remain operational: the axis is automatically released and re-locked at the end of the requested movement.

RESTART OF KA087 RELAY (THREE-PHASE SUPPLY LINE MONITORING)

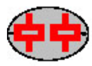
Inside the electrical cabinet of your Machining Center you may easily locate the KA087 component which controls the three-phase supply line of the machine.



This component sets the machine to emergency status when an error occurs on phase sequence or when the supply voltage raises or drops beyond the set value (standard voltage ± 10%).

To restart the component and power-on again the machine, open the electrical cabinet and carry out the following operation on KA087 component:

- Press simultaneously the “Arrow up” and “Arrow down” buttons
- Confirm that the component display shows the red indication RES? with a progressive bar.
- Keep the buttons pressed for 2-3 additional seconds, then release them.

If the power line is now stabilized, then the component will return to Run status and the symbol  will appear on the bottom part of component display, together with the actual power voltage.

SPINDLE PREHEATING

The spindle preheating cycle is used to ensure a perfect lubrication of spindle bearing; this operation must be carried out:

- at the machine power on
- after the electrospindle chartridge change (only for machines with JIMS system),
- after a spindle stop exceeding 1 hour.

The preheating cycle is requested by the machine with a specific message “87: CARRY OUT SPINDLE PRE-HEATING”, and it is automatically started up.

The preheating cycle consists of the following steps:

1. Preheating with spindle rotating at 10% of the maximum speed for 1 min
2. Preheating with spindle rotating at 50% of the maximum speed for 2 min.
3. Preheating with spindle rotating at 75% of the maximum speed for 2 min.
4. Preheating with spindle rotating at 90% of the maximum speed for 2 min.
5. Preheating with spindle rotating at 100% of the maximum speed for 2 min.
6. At the end of the heating time, the spindle can be used at 100% of the maximum speed.

INFORMATION

For further information about the spindle, please refer to the specific manual (see section ENCLOSURES).

In particular refer to the following chapters:

FIRST START UP

START UP AND TURN OFF

DAILY STARTUP (eventual preheating’s cycle)

MACHINE DYNAMIC CUSTOMIZATION

In order to optimize the machine performance according to the different working conditions (materials, quality levels, machining times), 7 different CNC-user subprograms expressly created by JOBS are available for the operator.
The execution of one of these subprograms allows overwriting some default-parameters of the CNC, thus permitting to get a different setting of the dynamics (acceleration and Jerk) and of the machine's Look-Ahead.

ATTENTION

The writing of these parameters is reserved to JOBS technical staff machine calibration.

Any other modification carried out by not-authorized staff can seriously compromise the machine operation.

The files are so called:

M350 : Mechanical machining
M451 : Fast roughing
M351 : Normal roughing
M452 : Finishing
M352 : Super- finishing
M453 : Fast semi-finishing
M353 : Normal semi-finishing

At the turning ON of the machine the M350 function is active by default.

MACHINE OPTIONS

COOLING UNIT

This option consists of:

- Coolant delivery from spindle nozzles
- coolant delivery through the tool

The cooling unit is composed of:

- Collection tank where the coolant is delivered by the chip conveyor.

  The tank is supplied with maximum and minimum electric level.
- Main tank .

- **PROGRAMMING**

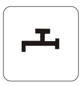
The cooling unit services are programmed through the following M functions:

M08 : tool external coolant

M18 : tool internal coolant

M09 : De-activation M08, M18

These functions can be programmed in MDI or within a Part-Program: in both cases, the function must be enabled or disabled by pressing the COOLANT ENABLING push-button

-  **COOLANT ENABLING push button**

  Press once to enable the cooling functions (programmed through M08, M18). Press a second time to disable it.

CHIP CONVEYOR

The machine can be supplied with an automatic conveyor system for chips produced during machining.

- **PROGRAMMING**

The conveyor programming and operation is carried out by means of the following M functions:

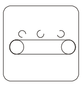
M54: Chip conveyor activation

M55 : Chip conveyor activation on reverse

M56 : Chip conveyor stop

These functions can be programmed in MDI or inside a Part-Program.

Programming the M54 function the conveyor starts. The M55 function causes the conveyor reverse motion and it can be useful to release a possible conveyor belt clogging.
The M56 function stops the conveyor motion.

-  **TRASPORTATORE TRUCIOLI activation push-button**

  Press once to enable the chip conveyor M functions.
  Press again to disable.

COOLANT (SPRAY MIST)

The machine can be supplied with a mist cooling system on the cutting edge (SPRAY MIST).

- **PROGRAMMING**

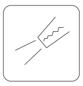
The SPRAY-MIST programming is carried out by means of the following M functions:

M07 : SPRAY-MIST Coolant

M09 : Reset M07

These functions can be programmed in MDI or inside a Part-Program.

The SPRAY MIST (INTERNAL and/or EXTERNAL) is delivered only if the SPRAY MIST enabling push-button was pressed before:

-  **SPRAY MIST INTERNAL / EXTERNAL enabling push-button**

Press once to enable the function.

Press again to disable.

COOLANT (AIR BLOW)

The machine can be supplied with an air blow cooling system.

- **PROGRAMMING**

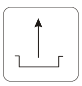
The AIR BLOW programming is carried out by means of the following M functions:

M77 : AIR BLOW ENABLING

M79 : Reset M77

These functions can be programmed in MDI or inside a Part-Program.

The AIR BLOW(INTERNAL and/or EXTERNAL) is delivered only if the AIR BLOW enabling push-button was pressed before:

-  **AIR BLOW INTERNAL / EXTERNAL enabling push-button**

Press once to enable the function.

Press again to disable.

WORK PIECE PROBE RMP60

INFORMATION

For a detailed description of the work piece probe RMP60, including the operation and maintenance procedure, please refer to the relevant RENISHAW manuals.

The Renishaw probe is used to carry out checks on work piece dimensions, zero set and tool centering cycles, and to determine the actual work piece position with respect to the theoretical one.

It is equipped with a tool coupling cone so that it can be mounted on the machine spindle in place of the tool.
It is not large in size and can be loaded into the machine tool magazine. As with any other tool, the machine can load the probe on the spindle automatically, by picking it up from the magazine (if present) and replacing it at the end of the measurement cycle (in this case, please refer to the chapter entitled “TOOL CHANGE PART”).

An interface receives and decodes the signals transmitted by the tracer pin. Once mounted on the spindle, its presence is automatically recognized by the machine, enabling it to function as previously established.

- **PROGRAMMING**

INFORMATION

For information on tracer pin operation, offset compensation, performing measurement cycles and programming functions, please refer to:

- **the “MEASUREMENT CYCLES” manual included with the HEIDENHAIN documentation**

WARNING

When the probe is installed, take great care during manual movement of the axes since even the smallest hit could damage the probe.

M47 : MI18 control deactivation

This function disables the monitoring of the MI18 ERROR input.

This control is used, at the end of a probing cycle and subsequent deposit of the peobw (still on) in the magazine, to avoid the generation of unwanted error signals.

M48 : Reset M49

M49 : Activation of Renishaw MI18micro monitoring

This function activates the monitoring of the logic signal generated by the RMP60 probe when it comes in contact with the workpiece.

It is used to protect the tracer pin in the event of accidental contact during the approach to the workpiece during automatic cycles, MDI or JOG mode.

BLUM TOOL PROBE

On the machine it is possible to assemble a TOOL PROBE, whose function is to check (at programmer choice) if one or more tools are broken, and to check that the tool placed on the spindle corresponds to the one recognized by the control unit.

INFORMATION

For a detailed description of the tool probe device BLUM, including its operation and maintenance procedure, see the instructions manuals of the BLUM-manufacturer.

The probing position is stored and compared with the one expected (presetting position). In case of positive reply, the machining goes on. On the contrary, if the reply is negative the control unit displays a warning message displayed.

In this case, the choice depends on the work shift you are in, if an operator is present or not. In case of presence of the operator, you should carry out an analysis of the situation and take the most suitable steps; in case of non-presence of the operator, we suggest to “reject” the pallet upon which some machinings are definitely missing.

Anyway, we would like to point out that, in these situations, the operator has got various possibilities of choice: to go on with the machining by storing the problem or to load a spare tool; to machine available workpieces which do not require the broken tool or to stop the plant in extreme cases.

The elements which influence the choice are as numerous as the possibilities (times of machining, kind of tool concerned, ratio between the lost times in one case or in the other one etc.).

In any case, the probing of the tool should be carried out at the end of each working cycle, but the frequency of the control can be increased till you repeat it at each operation (each hole).

The tools upon which you have to carry out the control are generally the most fragile, like drills or taps, but there are no limits to the use of the device.

- **BLUM probe sliding way**

  A laser BLUM probe support sliding way is mounted inside the tool change.

  It is automatically moved by all calibration and measure cycles supplied by the Manufacturer. The tool change automatic door is open before the motion and it is closed once the sliding way is backward at the end of operations.

  The motion can be carried out by means of the HELP mode screen pages.

- **HELP Mode**

  Pressing the MOD button and the HELP softkey, a screen page appears, whose pages contain all the available controls for the tool change part motion.

  To activate the controls, it is necessary to scroll the cursor on the desired row and, once highlighted, press the NC START button to execute the operation.

MONITORING THE VIBRATIONS OF THE SPINDLE BEARINGS

As already reported in the section about the GENERAL DESCRIPTION, on machines fitted with Jobs-Fischer spindle, it is possible to carry out the monitoring of vibrations occurring on bearings which support the spindle shaft.

The vibrations heavily affect the spindle life and therefore the duration of spindle warranty.

The spindle axial and radial vibration detection is carried out with the IBIS unit (SIVIB Record) inside the machine electrical cabinet.

Furthermore the use of IBIS control unit allows implementing two different sensorization levels, by means of special dedicated software functions.

First level sensorization

It allows you to view in real time the following values:

- Spindle rotation speed	in Rpm
- Absorbed power	in % with respect to max power
- Front bearing temperature	in °C
- Rear bearing temperature	in °C
- Vibrations on bearings	in mm/s

Second level sensorization

- Spindle working time (expressed in seconds), subdivided into “vibration bands”
- Vibration influence on spindle life (i.e. on spindle warranty time)
  (The data are saved to an internal system file)

INFORMATION

For more information about spindle, please consult the relevant manual. (See ENCLOSURES section).

In particular, read the following chapters:

FIRST-TIME OPERATION

STARTING UP AND RUNNING DOWN THE SPINDLE

DAILY START-UP PROCEDURE (with preheating cycle)

END

TOOL CHANGE SECTION

The TOOL CHANGE SECTION consists of a motor-driven chain magazine with a capacity of 20 tool places - HSK-A-63 (u axis) – protected on the front side by a movable shutter.

The chain is supported by a movable saddle, whose forward and backward movements are provided by a hydraulic cylinder.

The chain can rotate in both directions and its movement combined with the saddle stroke gives as a result the automatic tool change on spindle.

INFORMATION

For further information concerning the tool change unit see the relevant DESCRIPTION SHEET.

AUTOMATIC TOOL CHANGE CYCLE

The tool load/unload operations from spindle to magazine are made up of movement sequences of both machine and tool axes, which are programmed within an automatic cycle.
Depending on the programmed tool and on the tool actually mounted on spindle the cycle carries out the following operations:

- Tool deposit/pick up
- Tool pick up only
- Tool deposit only.

TOOL DEPOSIT/PICK UP AUTOMATIC CYCLE

The automatic sequence includes the following steps:

- De-activation of 1st software limit switch (work zone limit) and activation of 2nd software limit switch (tool change zone limit) for Y axis.
- Spindle stop
- Z-axis positioning to head orientation position and C/A-axes positioning to tool change position.

- Z-axis positioning to safety distance.
- X-axis positining to the tool change point.
- Forward movement of tool magazine saddle.
- Chain rotation to the assigned tool place
- Spindle orientation for tool change operation
- Y-axis positioning to safety distance
- Check for selected tool place free (photocell)
- Z-axis positioning to tool change point
- Y-axis positioning to first deposit point
- Slow Y-axis positioning to actual tool change point
- Tool unclamping and transfer from spindle to magazine
- Slow Z-axis positioning to first pick up point
- Chain rotation to the assigned tool place
- Check for tool presence on selected tool place
- Slow Z-axis positioning to the actual tool change point
- Tool clamping on spindle
- Slow Y-axis positioning to first deposit point
- Z-axis positioning to safety distance
- Backward movement of tool magazine saddle
- Y-axis positioning to safety distance
- De-activation of 2° software limit switch (work zone limit) and activation of 1° software limit switch (tool change zone limit).

End of cycle

TOOL PICK UP AUTOMATIC CYCLE

The automatic sequence includes the following steps:

- De-activation of 1st software limit switch (work zone limit) and activation of 2nd software limit switch (tool change zone limit) for Y axis.
- Spindle stop
- Z-axis positioning to head orientation position, of Y-axis to safety distance and of C/A-axes positioning to tool change positions
- Forward movement of tool magazine saddle.
- Chain rotation to the assigned tool place
- Z-axis positioning to safety distance
- Spindle orientation for tool change operation
- Check for tool presence on selected tool place
- Y-axis positioning to actual tool change point
- Z-axis positioning to first pick up point
- Tool collet opeing
- Slow Z-axis positioning to the actual tool change point
- Tool clamping on spindle
- Slow Y-axis positioning to first deposit point
- Z-axis positioning to safety distance
- Backward movement of tool magazine saddle
- Y-axis positioning to safety distance
- De-activation of 2° software limit switch (work zone limit) and activation of 1° software limit switch (tool change zone limit).

End of cycle

TOOL DEPOSIT AUTOMATIC CYCLE

The automatic sequence includes the following steps:

- De-activation of 1st software limit switch (work zone limit) and activation of 2nd software limit switch (tool change zone limit) for Y axis.
- Spindle stop
- Z-axis positioning to head orientation position and C/A-axes positioning to tool change position.
- Z-axis positioning to safety distance.
- X-axis positining to the tool change point.
- Forward movement of tool magazine saddle.
- Chain rotation to the assigned tool place
- Spindle orientation for tool change operation
- Y-axis positioning to safety distance
- Check for selected tool place free (photocell)
- Z-axis positioning to tool change point
- Y-axis positioning to first deposit point
- Slow Y-axis positioning to actual tool change point
- Tool unclamping and transfer from spindle to magazine
- Slow Z-axis positioning to first pick up point
- Closure of tool clamping collet
- Z-axis return to safety position.
- Backward movement of tool magazine saddle.
- Z-axis positioning to safety distance.
- De-activation of 2nd software limit switch (work zone limit) and activation of 1st software limit switch (tool change zone limit).

End of cycle

TOOL CHANGE SECTION: PROGRAMMING

The tool automatic change is executed by programming the sequence:

TOOLCALL (Tool name) Indicate a value between 0 ÷ 32
Indicate the motion axis
Confirm with START

NOTE

For more detailed information about the tool programming and management, please refer to the programming manual of CNC HEIDENHAIN Manufacturer

If an automatic tool change cycle is activated while the tool change pushbutton panel is enabled, then a warning message is displayed and the cycle is stopped until the active pushbutton panel will be disabled.
Press START NC to continue.

A set of warnings and messages inform the operator about possible problems arising during the tool change cycle.

RESETTING THE TOOL CHANGER

It can happen that, during the execution of a tool change automatic cycle, such cycle is not ended up properly due to some problems.
Press RESET and start the machine again.
Use the tool change panel for any manual operation on tool magazine.

The tool magazine is properly reset when:

- The travelling saddle is backward and the door is closed.
- The tool positions in the chain and on spindle match the positions shown on the tool table.

INFORMATION

For NC tool table displaying and management see the HEIDENHAIN Manual.

- The Z and Y axes are set to safety position when the Y-axis returns within the work area.
- The tool clamping collet is closed with or without tool.

ATTENTION

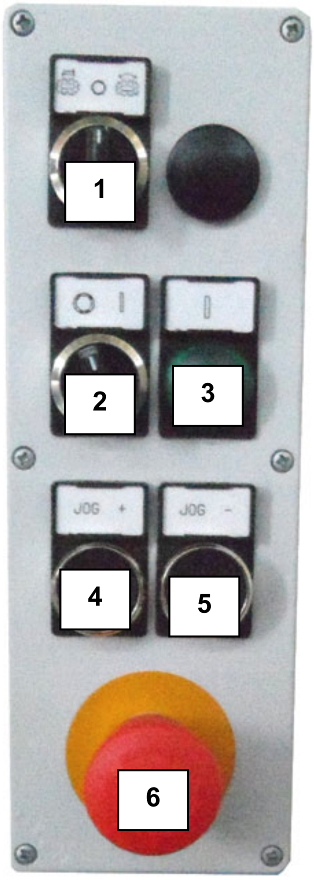
Particular attention shall be paid to the travelling saddle movement everytime the tool is inserted and the Z axis is in tool change position.

First unclamp the tool, move the Z axis in JOG mode to a safety position and then move the saddle backwards.

TOOL CHANGE KEYBOARD

This keyboard is positioned nearby the tool magazine.

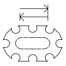
The controls for a safe moving of the tool magazine are available on this keyboard.



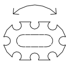
TOOL CHANGE keyboard

- **1 – FUNCTIONS selector**

  Two-position selector:

   **Tool magazine saddle forward/backward positioning enable**

  In this position the tool change saddle movements are selected.

   **Chain rotation enable**

  In this position the tool change chain movements are selected.

- **2 - Key-selector for KEYBOARD ENABLE**

  Key selector enabling the tool magazine keyboard.

- **3 - KEYBOARD ENABLE lamp**

  This lamp is on everytime the keyboard is enabled.

- **4/5 - JOG+/JOG- Pushbuttons**

  These push buttons carry out the command preset by means of the FUNCTION SELECTOR.
  The JOG push button works through impulses.
  As to the function TOOL MAGAZINE SADDLE FORWARD/BACKWARD it is not necessary to keep the JOG push button pressed in order to carry out the movement: a single pressing is enough. The FORWARD/BACKWARD TOOL MAGAZINE SADDLE movement automatically causes the opening/closing of the protection shutter.
  JOG+ forward positioning of magazine saddle.
  JOG- backward positioning of magazine saddle.

  For the CHAIN ROTATION enable function:

  1. pressing the JOG+, the tool magazine chain moves forward by one position
  2. pressing the JOG-, the tool magazine chain moves backward by one position.

  The “saddle forward” command is carried out only when the Y and Z axes are set to “safety position” (out of encumbrance).

- **6 – EMERGENCY stop push button**

  It is a red mushroom-shaped push button with yellow crown. Everytime it is pressed the machine enters the emergency state and cuts off the supply to power circuits.
  In order to release this push button, rotate it clockwise as indicated by the arrow on the push button itself.

MANUAL TOOL LOADING/UNLOADING FROM MAGAZINE

The operator can replace the tools in chain outside the machining activities.

This operation, which is necessary at the beginning of the machine operating cycle, is to be repeated everytime different tools are to be used (requested by new working cycles) or every time worn tools are to be replaced.

The operator enters the tool handling zone through the opening of the magazine protection.

The tool loading/unloading procedure from magazine is to be carried out by the operator as follows:

- Set the KEYBOARD ENABLE selector to position 1. The automatic tool change is disabled by the control unit once the operation in progress has been terminated. Wait for the lighting up of the KEYBOARD ENABLED lamp.
- Set the FUNCTION selector to CHAIN ROTATION ENABLE and make the chain rotate clockwise or counterclockwise through the push buttons JOG+ and JOG- so as to move the selected tool place to a position comfortable for the manual change.
- Set the KEYBOARD ENABLE selector key to position 0. The control unit does not allow the chain rotation and enables the opening of the protection door.

Open the door and load or unload the tool. The loading operation is as follows: push the tool in its

chain seat (in the angular position requested by the reference pin).

The unloading operation consists in extracting the tool while holding it firmly.

Close the door.

These operations shall be repeated until all desired loading/unloading operations have been completed.

ATTENTION

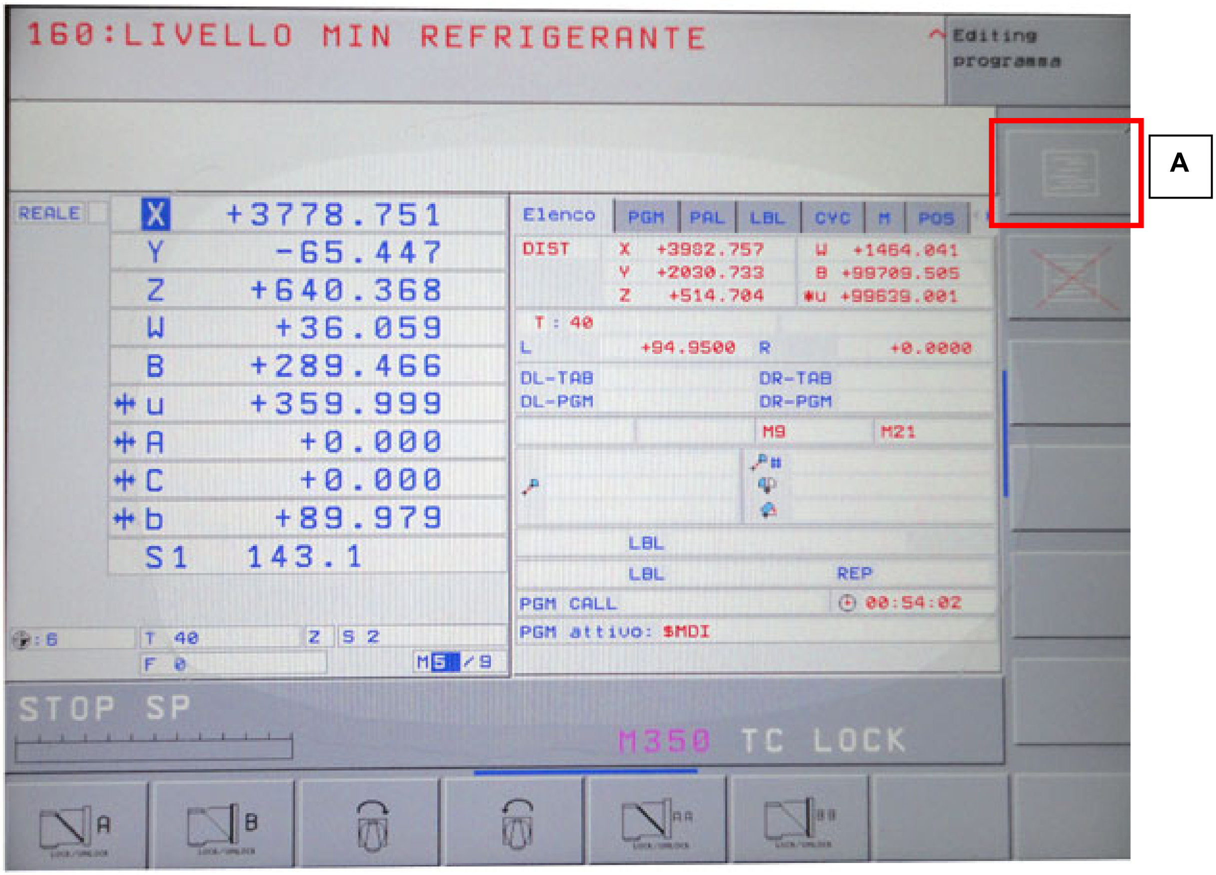
The described procedure is complete only in case a worn tool is to be replaced with a new one which will be placed on the same position in the chain.

In case of loading of different tools or tools which occupy different positions the magazine shall be updated while updating the suitable tool table, according to the procedure described in the HEIDENHAIN Manual.

END

VISUALIZATION OF MACHINE VALUES

Pressing the Help button (**A**) you may display 5 pages of machine data values which could provide usefully information to maintenance staff:

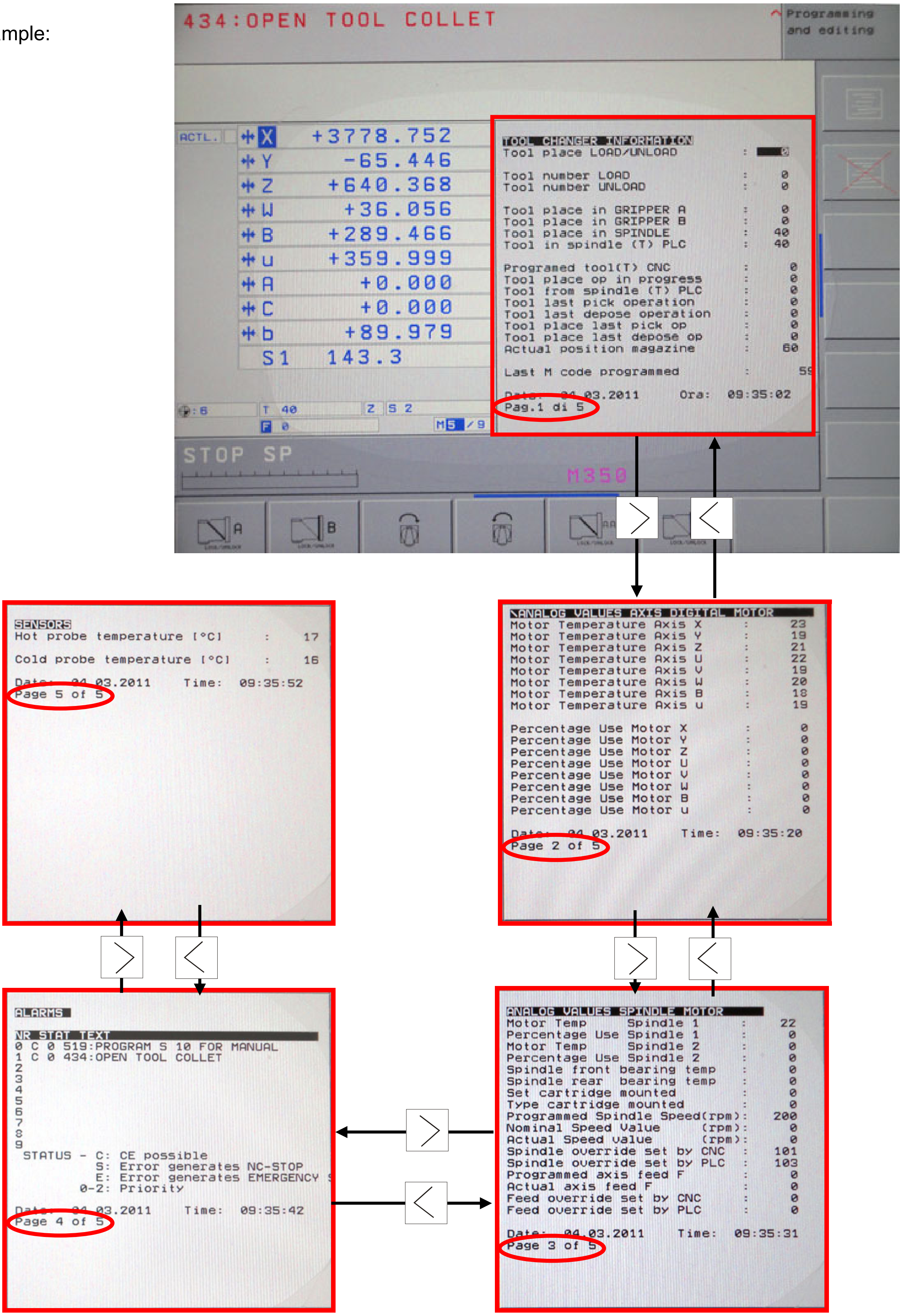


the 5 pages contain the following data:

- Information about tool changer (page 1)
- Analogue values about axes digital motors (page 2)
- Analogue values about spindle motor (page 3)
- Currently active alarms (page 4)
- Temperature sensors (page 5)

To navigate throughout the pages press either > or <

Example:


434:OPEN TOOL COLLET
Programming and editing
ACTL. X +3778.752
Y -65.446
Z +640.368
W +36.056
B +289.466
u +359.999
A +0.000
C +0.000
b +89.979
S1 143.3
STOP SP
M350
TOOL CHANGER INFORMATION
Tool place LOAD/UNLOAD :
Tool number LOAD : 0
Tool number UNLOAD : 0
Tool place in GRIPPER A : 0
Tool place in GRIPPER B : 0
Tool place in SPINDLE : 40
Tool in spindle (T) PLC : 40
Programed tool(T) CNC : 0
Tool place op in progress : 0
Tool from spindle (T) PLC : 0
Tool last pick operation : 0
Tool last depose operation : 0
Tool place last pick op : 0
Tool place last depose op : 0
Actual position magazine : 60
Last M code programmed :
Data: 04.03.2011 Ora: 09:35:02
Pag.1 di 5
ANALOG VALUES AXIS DIGITAL MOTOR
Motor Temperature Axis X : 23
Motor Temperature Axis Y : 19
Motor Temperature Axis Z : 21
Motor Temperature Axis U : 22
Motor Temperature Axis V : 19
Motor Temperature Axis W : 20
Motor Temperature Axis B : 18
Motor Temperature Axis u : 19
Percentage Use Motor X : 0
Percentage Use Motor Y : 0
Percentage Use Motor Z : 0
Percentage Use Motor U : 0
Percentage Use Motor V : 0
Percentage Use Motor W : 0
Percentage Use Motor B : 0
Percentage Use Motor u : 0
Date: 04.03.2011 Time: 09:35:20
Page 2 of 5
ANALOG VALUES SPINDLE MOTOR
Motor Temp Spindle 1 : 22
Percentage Use Spindle 1 : 0
Motor Temp Spindle 2 : 0
Percentage Use Spindle 2 : 0
Spindle front bearing temp : 0
Spindle rear bearing temp : 0
Set cartridge mounted : 0
Type cartridge mounted : 0
Programmed Spindle Speed(rpm): 200
Nominal Speed Value (rpm): 0
Actual Speed value (rpm): 0
Spindle override set by CNC : 101
Spindle override set by PLC : 103
Programmed axis feed F : 0
Actual axis feed F : 0
Feed override set by CNC : 0
Feed override set by PLC : 0
Date: 04.03.2011 Time: 09:35:31
Page 3 of 5
ALARMS
NR STAT TEXT
0 C 0 519:PROGRAM S 10 FOR MANUAL
1 C 0 434:OPEN TOOL COLLET
STATUS - C: CE possible
S: Error generates NC-STOP
E: Error generates EMERGENCY
0-2: Priority
Date: 04.03.2011 Time: 09:35:42
Page 4 of 5
SENSORS
Hot probe temperature [°C] : 17
Cold probe temperature [°C] : 16
Date: 04.03.2011 Time: 09:35:52
Page 5 of 5

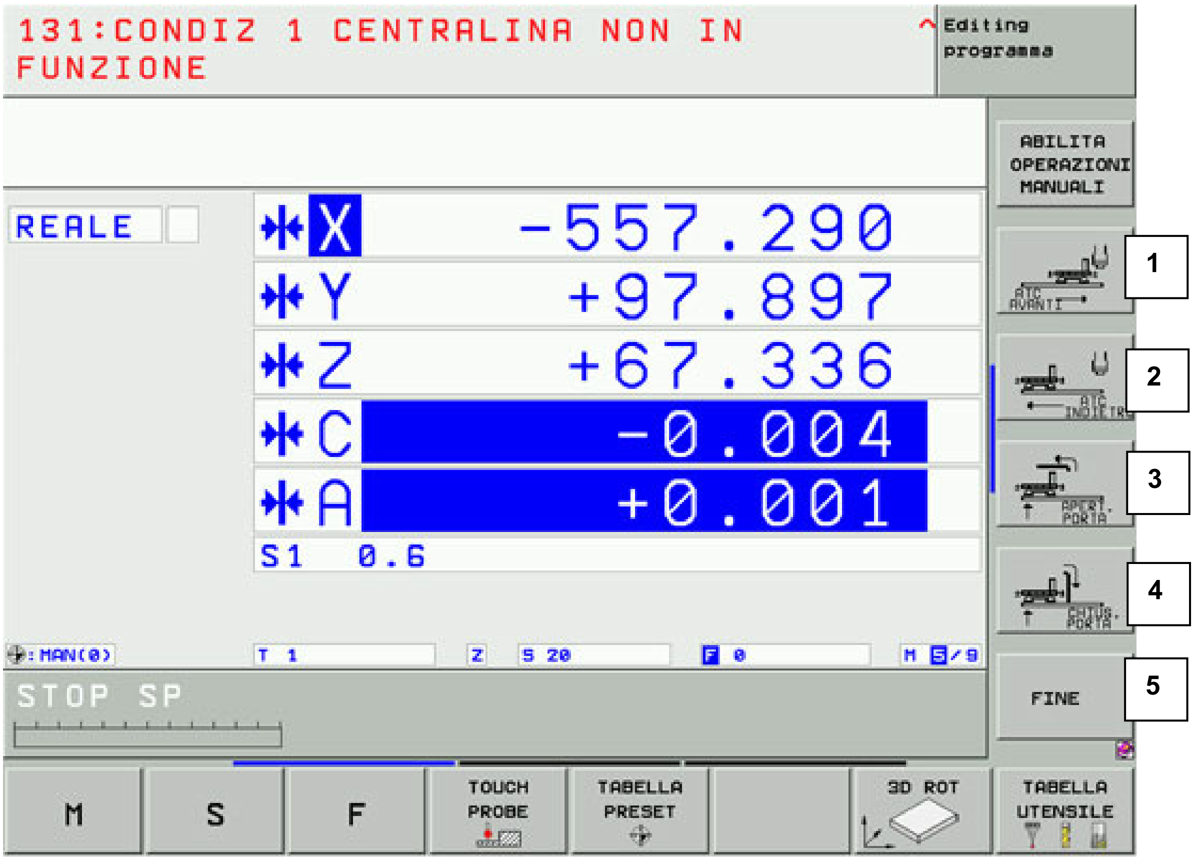
END

TOOL CHANGER RESTORE CYCLES

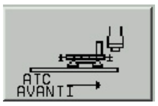
When a malfunction occurs during an automatic tool change cycle, it could be necessary to drive the various movements of the tool changer in order to manually restore the normal system operation.

To do this, operate as follows:

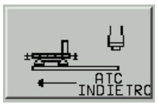
- select the manual operation mode.
- the following mask will appear, containing the softkeys on the side part of the screen.



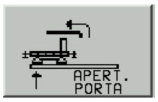
Such softkeys allow you to carry out the following restore operation on tool changer unit:

1 

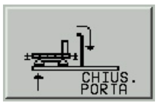
Tool magazine forward (ATC FORWARD)

2 

Tool magazine backward (ATC BACKWARD)

3 

Door opening control (DOOR OPEN.)

4 

Door closure control (DOOR CLOS.)

5 

Return to Heidenhain standard menu

Actually, they are the same operation that you can carry out manually using the **Tool Exchange Push-Button Panel**; however, when you operate with the screen masks, you may actually complete a tool change cycle interrupted because of sudden malfunction, keeping the synchronism with the CNC.
The only movement which is not allowed during the restore operation is the tool-holder chain rotation

END

AUXILIARY M FUNCTIONS

Listed below are all auxiliary M functions available to the operator.

INFORMATION

For M functions relating to:

- **MACHINE PART**
- **TOOL CHANGE PART**

please see the specific chapters in this manual.

Availability of these functions depends on the options selected and on units installed on the machine.

These functions allow operations to be performed or units activated under safety conditions for the operator and the machine. Please refer to the Numeric Control documentation for the programming and application of these functions.

NOTE

The use and programming of other functions is strictly forbidden since they may be destined for different uses making it impossible to guarantee full compliance with safety conditions.

M00 = END PART-PROGRAM
M02 = STOPS THE PART-PROGRAM AND STOPS ALSO THE SPINDLE
M03 = CLOCKWISE SPINDLE ROTATION
M04 = COUNTER-CLOCKWISE SPINDLE ROTATION
M05 = SPINDLE ROTATION STOP

M07 = SPRAY MIST COOLANT
M08 = TOOL EXTERNAL COOLANT
M18 = TOOL INTERNAL COOLANT
M09 = M07, M08, M18 RESET

M19 = SPINDLE ORIENTATION FOR 0 DEGREES
M20 = SPINDLE ORIENTATION – CYCLE 13
M30 = LIKE M02

M47 = MI18 CONTROL DEACTIVATION
M48 = RESET M49
M49 = ACTIVATION OF RENISHAW MI18 MICRO MONITORING

M54 = CHIP CONVEYOR ACTIVATION
M55 = CHIP CONVEYOR ACTIVATION ON REVERSE
M56 = CHIP CONVEYOR STOP

M61 = A AXIS CLAMPING
M62 = A AXIS RELEASE
M63 = C AXIS CLAMPING
M64 = C AXIS RELEASE

M69 = RESET M70
M70 = PROBE STATUS MONITORING

M77 = AIR BLOW ACTIVATION
M79 = RESET M77

M350 = MECHANICAL MACHINING
M451 = FAST ROUGHING
M351 = NORMAL ROUGHING
M452 = FINISHING
M352 = SUPER- FINISHING
M453 = FAST SEMI-FINISHING
M353 = NORMAL SEMI-FINISHING

M419 = PROBE ORIENTATION

END

ALARMS AND MESSAGES

The paragraph provides the information concerning the Alarms and Messages defined by JOBS and displayed on the screen of the milling center.

INFORMATION

For what concerns the description of alarms defined by the CNC supplier, please refer to the HEIDENHAIN manuals.

How to interpret the ALARMS & MESSAGES table

1

180: HYDRAULIC OIL FILTER DIRTY	
i	BS711

2 3

1. Code and Alarm or Message description text
2. Comment or instruction for problem solution, or:

i	Self-explanatory message
E	Machine in Emergency status
R	Press Reset and, if necessary, restart the machine
Jobs	If the problem persists, please contact Jobs Service Department
NA	Not used on the machine in question
Hold	Axes and/or spindle in Hold status

3. Components to control (for more details, please refer to the enclosed component location table)

A MESSAGE disappears automatically as soon as the cause that generated it is removed.

WARNING

The operator must carefully evaluate the contents of messages issued by the machine.

Failure to comply woth this warning may lead to serious damages.

NOTE

Should the information provided by the following table not be sufficient to solve the problem, please contact the JOBS Service Department.

0:PLC IN SERVICE MODE

Verificare programma

1: MACHINE IN AXIS HOLD

5: C AXIS IN POSITIVE EMERGENCY

6: C AXIS IN NEGATIVE EMERGENCY

8: E-STOP PRESSED

Re-activate the pressed button prior to restart the machine

9: MACHINE AXES IN EMERGENCY

Axes zero not done. Carry out axes re-entry procedure (SL100)

10: MODAL SWITCH NOT OK

Check the modal selector switch wiring

11: FORCED MANUAL MODE FOR DOOR OPEN

Close the door prior to activate the automatic mode

12: DOOR OPENING ALLOWED

13: DOOR OPENING REQUEST

Turn the mode selector switch to manual or test position prior to turn the door opening request selector

15: DINA MODE NOT OPERATING

DINA

16: DINA MODULE OVERSPEED: MACHINE SWITCHING OFF IN PROGRESS

24: DEAD MAN BUTTON IS MISSING

Hold axes and spindle, program stop. Keep the dead man button pressed in maual/test mode

27: CNC NOT IN MANUAL MODE

Cycle start inhibited; switch the CNC to manual mode or turn mode selector to Auto position

28: CNC IN RUN FULL SEQUENCE

Cycle start inhibited; switch the CNC to MDA mode/single block or switch mode selector to Auto

30: ERROR M10 M11 MANAGEMENT

Hold Jobs

31: C AXIS NOT LOCKED

BP504

32: C AXIS NOT UNLOCKED

BP554

33: A AXIS NOT LOCKED

BP505

34: A AXIS NOT UNLOCKED

BP555

35: X AXIS NOT UNLOCKED	
E R i	**BP501**
36: X AXIS NOT LOCKED	
E R i	**BP501**
37: Y AXIS NOT UNLOCKED	
E R i	**BP502**
38: Y AXIS NOT LOCKED	
E R i	**BP502**
39: Z AXIS NOT UNLOCKED	
E R i	**BP521**
40: Z AXIS NOT LOCKED	
E R i	**BP521**
41: X1 AXIS NOT UNLOCKED	
E R i	**BP509**
42: X1 AXIS NOT LOCKED	
E R i	**BP509**
43: LINEAR AXES LOCKED M600	
E R	
54: ZERO SPINDLE TIMEOUT	
Execute the M20 control (cycle 13) Jobs	
55:C AXIS IN STROKE END +	
Axes Zero procedure not executed. Carry out axes return procedure (SL100)	**BSC+**
56: C AXIS IN STROKE END -	
Axes Zero procedure not executed. Carry out axes return procedure (SL100)	**BSC-**
57:SPINDLE S1 NOT ACTIVE	
i	
58: SPINDLE S2 NOT ACTIVE	
NA	
60: SPINDLE DOES NOT ROTATE:COLLET NOT OK	
Spindle rotation enabled only when collet closed with tool	
61: SPINDLE CANNOT ROTATE - TOOL CHANGE IN PROGRESS	
Wait for tool change cycle completion	
62: NO SPINDLE MOUNTED	
NA	
63: SPINDLE DRIVE NOT READY	
E R i **Jobs**	**SPINDLE MOTOR DRIVE**
65: SPINDLE IN AXIS MODE	
E R i **Jobs**	**SPINDLE MOTOR DRIVE**
66: SPINDLE NOT ROTATING	
E R i **Jobs**	**SPINDLE MOTOR DRIVE**

67: SPINDLE NOT AT SPEED	
Jobs	SPINDLE MOTOR DRIVE
72: SPINDLE ZERO NOT DONE	
Turn the spindle for a few seconds	
73: TIMEOUT SPINDLE ORIENTATION	
Jobs	
75: FRONT BEARING TEMP ALARM	
Jobs Verify bearings temperature ; See “Visualization of Machine values” paragraphs	
76: FRONT BEARING TEMP WARNING	
Check correct spindle operation; Verify bearings temperature; See “Visualization of Machine values” paragraphs	
77: REAR BEARING TEMP ALARM	
Jobs Verify bearings temperature ; See “Visualization of Machine values” paragraphs	
78: REAR BEARING TEMP WARNING	
Check correct spindle operation; Verify bearings temperature; See “Visualization of Machine values” paragraphs	
79:ELECTROSPINDLE NOT OFF	
The operation can be carried out only with spindle off	
80: FEEDING CONDITIONS NOT CONFIRMED	
Hold Axes; Check for possible additional messages	HEIDENHAIN
81: MP PRE-LUBRICATION TIME MISSING	
82:PRE-LUBRICATION IN PREOGRESS	
83: FISCHER SPINDLE DLS ERROR	
84:SPINDLE LUBRICATION AIR PRESSURE NOT OK	
	BP494
85: SPINDLE PRE-LUBRICATION MISSING	
86: SPINDLE PRE-HEATING MISSING	
87:CARRY OUT SPINDLE PRE-HEATING	
88: FISCHER UNIT OFF M464	
89: FISCHER UNIT CONTROLS OFF	
90:SPINDLE COOL.SYST. THERMAL SW. NOT OK	
	QM072
91: SP COOLING SYSTEM TEMPERATURE HIGH	
Jobs	KRA70J-E39.2

92: SP COOLING SYSTEM PRESSURE LOW	
	KRA70J-E39.3
93: X Y Z AXES CHILLER THERMAL SW. NOT OK	
	QM071
94: X Y Z AXES CHILLER TEMPER HIGH	
	KOL24J-E33.3
95: X Y Z AXES CHILLER PRESS.LEV LOW	
	KOL24J-E33.4
96: C-A AXES CHILLER THERMAL SW. NOT OK	
	QM073
97: C-A AXES CHILLER TEMPER HIGH	
	KRA70J-E34.2
98: C-A AXES CHILLER PRESS.LEV LOW	
	KRA70J-E34.3
100: A AXIS CHILLER FLOWMETER NOT OK	
	BC550
101: A1 AXIS CHILLER FLOWMETER NOT OK	
	BC551
102: C AXIS CHILLER FLOWMETER NOT OK	
	BC552
103: SPINDLE CHILLER EL.PAN. THERMAL SW. KO	
	QM072
104: SPINDLE CHILLER FLOWMETER NOT OK	
	BC527
105: X Y Z AXES CHILLER EL.PAN. THERMAL SW. KO	
	QM071
106: C-A AXES CHILLER EL.PAN. THERMAL SW. KO	
	QM073
107:X AXIS PRIMARY LINE NOT OK	
	BC528
108: X1 AXIS PRIMARY LINE NOT OK	
	BC534
109: Y AXIS PRIMARY LINE NOT OK	
	BC531
110: Z AXIS PRIMARY LINE NOT OK	
	BC533
111: X-X1 AXIS SECONDARY LINE NOT OK	
	BC546
112: Y-Z AXIS SECONDARY LINE NOT OK	
	BC547

115: TC CLAMP/RELEASE PISTONS NOT OK	
i	TOOL MAG.
116: TOOL CLAMP COLLET NOT OPEN	
i Jobs	TOOL MAG.
117: TC ERROR : SPINDLE ENABLED	
i	
118: TOOL CLAMP COLLET NOT CLOSED	
i Jobs	
129:EL. CAB. CONDITIONER 1 NOT SUPPLIED	
i	JET40
130: EL. CAB. CONDITIONER 2 NOT SUPPLIED	
i	JET40
131: EL.CABINET CONDIT. NOT WORKING	
i Jobs	JET40
132: EL.CABINET CONDITIONER HIGH TEMP.	
i	JET40
133: CONDITIONER 1 THERMAL SWITCH KO	
i	QM053
134: EL. CAB. CONDITIONER 2 NOT WORKING	
NA	
135: EL.CABINET CONDITIONER 2 HIGH TEMP.	
NA	
136: CONDITIONER 2 THERMAL SWITCH KO	
NA	
137: ELECTRICAL CABINET THERMOSTAT TRIPPED	
i	STEQ
138:MOUNTED PROBE RELEASE ERROR	
i	
139:PROBE BATTERY DISCHARGED	
i	
140:PROBE OFF	
i	
141:PROBE NOT READY	
i	
142:STYLUS DEFLECTED BEFORE CYCLE START	
i	
143: STYLUS DEFLECTED	
i	
144:PROBE CYCLE INTERRUPTED	
i	

145:BATTERY DISCHARGED	
147:SPRAY MIST NOT ENABLED	
Press the enabling push-button on Jobs panel	
148: SPRAY MIST MINIMUM LEVEL	
149: COOLANT NOT ENABLED	
Press the coolant delivery enable button on Jobs panel	
152:FILTER 1 CLOGGED	
153: FILTER 2 CLOGGED	
154:COOLANT TANK MIN. LEVEL	
	BL388- IN4 ASI ADR22
155: COOLANT TANK MAX. LEVEL	
	BL389- IN3 ASI ADR22
156: COOLANT TANK THERMAL SWITCH KO	
	QM302
157: COOLANT TANK AUTOMATIC SW. KO	
160: MIN LEVEL COOLANT	
	BL388- IN4 ASI ADR22
161: MAX LEVEL COOLANT	
	BL389- IN3 ASI ADR22
162: COOLANT TANK THERMAL SW.TRIPPED	
	QM302
163: COOLANT TANK PANEL THERMAL SW. KO	
164: COOLANT FLOWSWITCH NOT OK	
165: RECYCLING TANK 1 THERMAL SW. KO	
	QM320
166: RECYCLING TANK 1 MAX LEVEL	
	BL389-IN4 ASI ADR30
167: RECYCLING TANK 1 ERROR	
168: RECYCLING TANK 2 THERMAL SW. KO	
	QM321
169: RECYCLING TANK 2 MAX LEVEL	
	BL389-IN3 ASI ADR31

170: RECYCLING TANK 2 ERROR	
i	
171: AIR BLOW NOT ENABLED	
Press the enabling push-button on Jobs panel	
172: DUST SUCTION NOT ENABLED	
Press the enabling push-button on Jobs panel	
180: HYDRAULIC OIL FILTER DIRTY	
i	**BS711**
181: MIN.LEVEL HYDRAULIC OIL TANK	
i	**BL796**
182: HYDRAULIC PRESSURE LOW	
i	**BP798**
183: HYDRAULIC SERVICES NOT OK	
i	**BP798**
184: HYDRAULIC OIL TEMP HIGH	
i	**BR709**
185: HYDRAULIC UNIT THERMAL SWITCH TRIPPED	
i	**QM701**
186: Z-AXIS BALANCING NOT OK	
E i	**QY782, BP782, BP783**
187: MOVE Z AXIS TO PLUS LIMIT	
Move Z axis to positive limit switch	
188: Z-AXIS ACCUM. RECHARGE NEEDED	
i	**QY782, BP782**
189: Z AXIS ACCUMULATOR RECHARGE CORRECTLY FINISHED	
i	
190: ACCUMULATOR NOT RECHARGED	
E i	**QY782, BP782**
192: LUBRICATION PNEUMATIC CIRCUIT NOT OK	
i	
193: AXES LUBRICATION ERROR	
i Press the enabling push-button on Jobs panel	**BS703-D06-D07-D08**
194: MIN LEVEL LUBRICATION	
i	**BL797**
195: Z AXIS ACCUMULATOR RECHARGE ERROR	
i	**QY782, BP782**
199: PNEUMATIC SYSTEM NOT PRESSURIZED	
i	**BP778**
200: AXIS 1 SERVO DRIVE NOT READY	
E R i **Jobs**	**X MOTOR DRIVE**

201: AXIS 2 SERVO DRIVE NOT READY	
Jobs	Y MOTOR DRIVE
202: AXIS 3 SERVO DRIVE NOT READY	
Jobs	Z MOTOR DRIVE
203: AXIS 4 SERVO DRIVE NOT READY	
Jobs	C MOTOR DRIVE
204: AXIS 5 SERVO DRIVE NOT READY	
Jobs	A MOTOR DRIVE
205: AXIS 6 SERVO DRIVE NOT READY	
Jobs	S MOTOR DRIVE
206: AXIS 7 SERVO DRIVE NOT READY	
Jobs	u MOTOR DRIVE
207: AXIS 8 SERVO DRIVE NOT READY	
NA	
208: AXIS 9 SERVO DRIVE NOT READY	
NA	
209: SPINDLE SERVO DRIVE NOT OK	
Jobs	SPINDLE MOTOR DRIVE
210: AXIS 1 MOTOR OVERTEMPERATURE	
Jobs	X MOTOR DRIVE
211: AXIS 2 MOTOR OVERTEMPERATURE	
Jobs	Y MOTOR DRIVE
212: AXIS 3 MOTOR OVERTEMPERATURE	
Jobs	Z MOTOR DRIVE
213: AXIS 4 MOTOR OVERTEMPERATURE	
Jobs	C MOTOR DRIVE
214: AXIS 5 MOTOR OVERTEMPERATURE	
Jobs	A MOTOR DRIVE
215: AXIS 6 MOTOR OVERTEMPERATURE	
Jobs	S MOTOR DRIVE
216: AXIS 7 MOTOR OVERTEMPERATURE	
Jobs	u MOTOR DRIVE
217: AXIS 8 MOTOR OVERTEMPERATURE	
NA	
218: AXIS 9 MOTOR OVERTEMPERATURE	
NA	
219: SPINDLE MOTOR OVERTEMPERATURE	
Jobs	SPINDLE MOTOR DRIVE

220: AXIS 1 OVERLOAD WARNING	
Jobs Check X motor & wiring	**X MOTOR DRIVE**
221: AXIS 2 OVERLOAD WARNING	
Jobs Check Y motor & wiring	**Y MOTOR DRIVE**
222: AXIS 3 OVERLOAD WARNING	
Jobs Check Z motor & wiring	**Z MOTOR DRIVE**
223: AXIS 4 OVERLOAD WARNING	
Jobs Check C motor & wiring	**C MOTOR DRIVE**
224: AXIS 5 OVERLOAD WARNING	
Jobs Check A motor & wiring	**A MOTOR DRIVE**
225: AXIS 6 OVERLOAD WARNING	
Jobs Check S motor & wiring	**S MOTOR DRIVE**
226: AXIS 7 OVERLOAD WARNING	
Jobs Check u motor & wiring	**u MOTOR DRIVE**
227: AXIS 8 OVERLOAD WARNING	
NA	
228: AXIS 9 OVERLOAD WARNING	
NA	
229: SPINDLE OVERLOAD WARNING	
R **Jobs** Check spindle motor & wiring	**SPINDLE MOTOR DRIVE**
230: AXIS 1 OVERLOAD ALARM	
E R **Jobs** Check X motor & wiring	**X MOTOR DRIVE**
231: AXIS 2 OVERLOAD ALARM	
E R **Jobs** Check Y motor & wiring	**Y MOTOR DRIVE**
232: AXIS 3 OVERLOAD ALARM	
E R **Jobs** Check Z motor & wiring	**Z MOTOR DRIVE**
233: AXIS 4 OVERLOAD ALARM	
E R **Jobs** Check C motor & wiring	**C MOTOR DRIVE**
234: AXIS 5 OVERLOAD ALARM	
E R **Jobs** Check A motor & wiring	**A MOTOR DRIVE**
235: AXIS 6 OVERLOAD ALARM	
E R **Jobs** Check S motor & wiring	**S MOTOR DRIVE**
236: AXIS 7 OVERLOAD ALARM	
E R **Jobs** Check u motor & wiring	**u MOTOR DRIVE**
237: AXIS 8 OVERLOAD ALARM	
NA	
238: AXIS 9 OVERLOAD ALARM	
NA	

239: SPINDLE OVERLOAD ALARM	
E R i **Jobs** Check spindle motor & wiring	**SPINDLE MOTOR DRIVE**
240: AXES RELEASE ERROR	
i **Jobs**	
241: SERVO DRIVES NOT OFF	
i **Jobs**	
242: AXIS 12 SERVO DRIVE NOT READY	
i **Jobs**	
252: PTC SUPPLY NOT OK	
E R i **Jobs**	
253: X/X1/Y AXES OVERTEMPERATURE	
E R i **Jobs**	
257: A AXIS OVERTEMPERATURE	
E R i **Jobs**	
258: A1 AXIS OVERTEMPERATURE	
E R i **Jobs**	
259: Z AXIS OVERTEMPERATURE	
E R i **Jobs**	
260: SECONDARY OR NOT CONTROLLED AXIS	
i **PLC error; Jobs**	
261: WRONG FEED RATE	
i **PLC error; Jobs**	
262: AXIS-ZERO NOT DONE	
i **PLC error; Jobs**	
263: NO STROBE ACTIVE	
i **PLC error; Jobs**	
264: POSITIONING NOT ACTIVE	
i **PLC error; Jobs**	
265: POSITION NOT REACHED	
i **PLC error; Jobs**	
266: POSITION NOT REACHABLE	
i **PLC error; Jobs**	
267: POSITIONING NOT POSSIBLE	
i **PLC error; Jobs**	
268: MOVEMENT ABORTED	
i **PLC error; Jobs**	
269: POSITIONING IN PROGRESS	
i **PLC error; Jobs**	

270: AXIS NOT EXISTING	
PLC error; Jobs	
271: AXIS NOT DEFINED AS PLC AXIS	
PLC error; Jobs	
272: AXIS ALREADY POSITIONED	
PLC error; Jobs	
273: POSITION OUT OF RANGE	
PLC error; Jobs	
274: PLC AXIS IN OPEN LOOP	
PLC error; Jobs	
275: FEED RATE NOT ALLOWED	
PLC error; Jobs	
276: PLC AXIS ZERO NOT DONE	
PLC error; Jobs	
277: POSITIONING ERROR	
PLC error; Jobs	
278: RETURN CODE ERROR	
PLC error; Jobs	
279: AXIS NOT IN CORRECT POSITION	
PLC error; Jobs	
280: AXIS ZEROING ERROR	
PLC error; Repeat zero axes procedure	
290: SPINDLE DOES NOT EXIST	
PLC error; Jobs	
291: SPINDLE FEED OR DIR NOT CORRECT	
PLC error Jobs	
292: OPERATING MODE NOT CORRECT	
PLC error Jobs	
293: SPINDLE HOME NOT STARTED	
PLC error Jobs	
294: SPINDLE ALREADY MOVING	
PLC error Jobs	
295: ANOTHER AXIS IN MOVEMENT	
PLC error Jobs	
296: SPINDLE NOT SERVO CONTROLLED	
PLC error Jobs	
300:00 Laser system not ready	
	BLUM

301:01 Laser system not closed	
i	BLUM
302:02 Meas.system active before measurement	
i	BLUM
303:03 Scattering out of tolerance	
i	BLUM
304:04 Probing value error	
i	BLUM
305:05 Measurement signal not found	
i	BLUM
306:06 No measurement signal	
i	BLUM
307:07 No tool loaded T-No=0	
i	BLUM
308:08 Tool limit value exceeded	
i	BLUM
309:09 Collision danger cycle 585	
i	BLUM
310:10 Excessive thermal drift	
i	BLUM
311:11 Collision danger	
i	BLUM
312:12 Wrong tool axis definition	
i	BLUM
313:13 Wrong number of cutting edges	
i	BLUM
314:14 Wrong meas. data in TOOL.T	
i	BLUM
315:15 Check value of Q3xx parameter	
i	BLUM
316:16 Wrong parameter table	
i	BLUM
317:17 Wrong measuring evaluation	
i	BLUM
318:18 Spindle speed not reached	
i	BLUM
319:19 Faulty trigger position	
i	BLUM
328: BLUM SLIDING WAY NOT FORWARD	
i	

329: BLUM SLIDING WAY NOT BACKWARD	
330: CHIP CONVEYOR NOT ENABLED	
Press the enabling button before M command	
332: CHIP CONVEYOR 1 NOT OK	
	BS313
333: CHIP CONVEYOR 2 NOT OK	
	BS314
334: CHIP CONVEYOR 3 NOT OK	
NA	
335: CHIP CONVEYOR 1 THERM.SW.TRIPPED	
	QM313
336: CHIP CONVEYOR 2 THERM.SW.TRIPPED	
	QM314
337: CHIP CONVEYOR 3 THERM.SW.TRIPPED	
NA	
345:PAPER FILTER ERROR	
346: PAPER FILTER ENDED	
350: M-FUNCTION NOT CODED	
Check program	
351: ENABLING M FUNCTION MISSING	
Check program	
352: TOOL CHANGER IN MANUAL MODE	
Automatic tool change inhibited	
354:SPINDLE COLLISION	
355: REPORT CARTRIDGE NUMBER ERROR	
356: REPORT WRITING ERROR	
361: THERMAL SW. FA025 TRIPPED	
	FA025
362: THERMAL SW. FA027 TRIPPED	
	FA027
363: THERMAL SW. FA011 TRIPPED	
	FA011
364: THERMAL SW. FA12 TRIPPED	
NA	

366: THERMAL SW. FA039 TRIPPED	
	FA039
367: THERMAL SW. FA001 TRIPPED	
NA	
368:THERMAL SW. FA009 TRIPPED	
N	
369:THERMAL SW. FA11ASI TRIPPED	
	FA11ASI
370:THERMAL SW. FA030 TRIPPED	
	FA030
375:THERMAL SW. FA005 TRIPPED	
	FA005
376:THERMAL SW. FA026 TRIPPED	
	FA026
377:THERMAL SW. FA014 TRIPPED	
	FA014
378:THERMAL SW. FA032 TRIPPED	
	FA032
379:THERMAL SW. FA025 TRIPPED	
	FA025
380:THERMAL SW. FA32 TRIPPED	
	FA32
381:THERMAL SW. FA88 TRIPPED	
	FA88
382:THERMAL SW. QM701 TRIPPED	
	QM701
383:THERMAL SW. FA39 TRIPPED	
	FA39
390: TOOL CHANGE CYCLE ERROR	
391: NO TOOL ON SPINDLE FOR DEPOSIT	
392: MAGAZINE PLACE NOT FREE	
	BS167
393: TOOL ON SPINDLE FOR PICK-UP	
394: MAGAZINE PLACE NOT OCCUPIED	
395: NO HEAD MOUNTED	

396: NO CARTRIDGE MOUNTED

400: CHECK TOOL TABLE

Check tool place configuration on relevant CNC page

401: MANUAL TOOL DEPOSIT: NC-START

402: MANUAL TOOL PICKING: NC-START

403: ATC IN MANUAL MODE

Automatic tool change inhibited

404: PICK AUTO TOOLCALL Tx

405: ATC NOT POSSIBLE

PLC ErrorJobs

406: PICK MAN TOOLCALL Tx

407: MANUAL TOOL CHANGE Tx

408:DEP AUTO T0

409: DEP AUTO

410:DEP MAN T0

411: DEP MANUAL

412: PICKING PLACE ERROR

Check programmed tool number. **Jobs**

413: TOOL PRESENT IN SPINDLE

E R

414: TOOL NOT PRESENT IN SPINDLE

E R

415: ERR 2ND TOOLCALL

416: ERR TOOLCALL Tx

417: DEP PLACE NOT OK FOR CURRENT SP

418: CHANGE TOOL ON COLLET

419: T PRESELECTION ONLY IN AUTO MODE	
i	
420: HEAD NOT IN POSITION FOR ATC	
i	
421: T PLACE IN SPINDLE NOT OK	
i	
422: T NUM IN SPINDLE NOT OK	
i	
423: ATC MANUAL OPERATION ENABLED	
i	
427: T AXIS DRIVE NOT ENABLED	
i	
428: T AXIS MOTOR NOT POWERED	
i	
430: ATC MANUAL DOOR NOT CLOSED	
E R i	**BS169**
431: AUTOMATIC PROCEDURE IN PROGRESS	
i R	
432: ATC HANDBOX ENABLED	
E R Automatic tool change inhibited	
433: TOOL MAG. NOT FORWARD	
i	**BS151**
434: TOOL MAG. NOT BACKWARD	
i	**BS150**
435: DEPOSIT PLACE BUSY	
i	**BS167**
436: PICK-UP PLACE EMPTY	
i	**BS167**
437: TC NOT RESET AFTER EXTRA-STROKE	
i	
438: Y AXIS MOV. ONLY TO - DIRECTION	
During the tool change phase correction, the movement of Y axis is allowed only in - direction	
439: Z AXIS MOV. ONLY TO + DIRECTION	
During the tool change phase correction, the movement of Z axis is allowed only in + direction	
440: X AXIS IN ENCUMBRANCE	
i	**BS591**
441: ATC DOOR NOT OPEN	
i	**BS143**
442: ATC DOOR NOT CLOSED	
i	**BS144**

443: TC SLIDING WAY RETURN NOT POSSIBLE	
444: Y AXIS IN TOOL CHANGE AREA	
	BS590
445: Z AXIS IN ENCUMBRANCE	
	BS589
447: ATC AUTO IN PROGRESS	
450: CARTRIDGE ALREADY MOUNTED	
498: LOGBOOK OPTION: AXES AND SPINDLE EXCEEDED THE LIMIT	
499: LOGBOOK OPTION: AXES OR SPINDLE UNDER THE LIMIT	
560:CYLINDER 1 NOT CLAMPED MASTER SIDE	
NA	
561: CYLINDER 2 NOT CLAMPED SLAVE SIDE	
NA	
562: CYLINDER 3 NOT RELEASED MASTER SIDE	
NA	
563: CYLINDER 4 NOT RELEASED SLAVE SIDE	
NA	
564: CYLINDER 1 NOT RELEASED MASTER SIDE	
NA	
565: CYLINDER 2 NOT RELEASED SLAVE SIDE	
NA	
566: CYLINDER 3 NOT CLAMPED MASTER SIDE	
NA	
567: CYLINDER 4 NOT CLAMPED SLAVE SIDE	
NA	
601: MODULE NOT SUBMIT CALLED MOD 9094	
PLC Error, Jobs	
602: FILE TYPE DOES NOT EXIST MOD 9094	
PLC Error, Jobs	
603: NO FILE WITH M STATUS FOUND MOD 9094	
PLC Error, Jobs	
604: LINE NUMBER NOT IN FILE MOD 9094	
PLC Error, Jobs	

605: INCORRECT ELEMENT NUMBER MOD 9094
PLC Error, Jobs
606: ELEMENT VALUE OUT OF RANGE MOD 9094
PLC Error, Jobs
611: INVALID POCKET NUMBER MOD 9341
PLC Error, Jobs
612: INVALID MAG NUMBER MOD 9341
PLC Error, Jobs
613: INVALID MODE MOD 9341
PLC Error, Jobs
614: INVALID T NUMBER OR TYPE MOD 9341
PLC Error, Jobs
615: RESERVE NOT POSSIBLE MOD 9341
PLC Error, Jobs
616: ERR MAGAZINE RULES MOD 9341
PLC Error, Jobs
617: MODULE NOT SUBMIT CALLED MOD 9341
PLC Error, Jobs
618: FILE ERROR TABLE MOD 9341
PLC Error, Jobs
619: MODULE EXECUTION CANCELED MOD 9341
PLC Error, Jobs
620:ERR LOCK TOOL TABLE MOD 9341
PLC Error, Jobs
657:INVALID TOOL PLACE NUMBER
i
670:INVALID TOOL NUMBER
i
671:INVALID UNLOAD TOOL NUMBER
i
672: TOOL PLACE SEARCH TIMEOUT
i
676:INVALID TOOL PLACE
i
678:INVALID POCKET OR T NUM MOD 9096
PLC Error, Jobs
679:MODULE NOT SUBMIT CALLED MOD 9096
PLC Error, Jobs

680:FILE ERROR TABLE MOD 9096
PLC Error, Jobs
681:INVALID POCKET NUMBER MOD 9340
PLC Error, Jobs
682:INVALID MAG NUMBER MOD 9340
PLC Error, Jobs
683:INVALID MODE MOD 9340
PLC Error, Jobs
684:INVALID T NUMBER OR TYPE MOD 9340
PLC Error, Jobs
685:MODULE NOT SUBMIT CALLED MOD 9340
PLC Error, Jobs
686:FILE ERROR TABLE MOD 9340
PLC Error, Jobs
687:MODULE EXECUTION CANCELED MOD 9340
PLC Error, Jobs
688:NO TOOL POCKET FOUND MOD 9340
PLC Error, Jobs
689:INTERNAL ERROR MOD 9340
PLC Error, Jobs
691:INVALID MODE MOD 9342
PLC Error, Jobs
692:INVALID MAG NUMBER MOD 9342
PLC Error, Jobs
693:MODULE NOT SUBMIT CALLED MOD 9342
PLC Error, Jobs
694:TOOL NOT FOUND MOD 9342
PLC Error, Jobs
695:FILE ERROR TABLE MOD 9342
PLC Error, Jobs
696:NO TOOL MAG FOUND MOD 9342
PLC Error, Jobs
697:NO TOOL POCKET FOUND MOD 9342
PLC Error, Jobs
698:INTERNAL ERROR MOD 9342
PLC Error, Jobs
710:DRIVE NOT READY: AXIS 10
E R i Jobs X1 AXIS DRIVE

711: DRIVE NOT READY: AXIS 11	
NA	
712: DRIVE NOT READY: AXIS 12	
NA	
713: DRIVE NOT READY: AXIS 13	
NA	
720:MOTOR OVERTEMPERATURE: AXIS 10	
E R i **Jobs**	**X1 AXIS DRIVE**
721: MOTOR OVERTEMPERATURE: AXIS 11	
NA	
722: MOTOR OVERTEMPERATURE: AXIS 12	
NA	
723: MOTOR OVERTEMPERATURE: AXIS 13	
NA	
730: OVERLOAD PRE-ALARM: AXIS 10	
Jobs Check motor and wirings X1 axis	**X1 AXIS DRIVE**
731: OVERLOAD PRE-ALARM: AXIS 11	
NA	
732: OVERLOAD PRE-ALARM: AXIS 12	
NA	
733: OVERLOAD PRE-ALARM: AXIS 13	
NA	
740: OVERLOAD ALARM: AXIS 10	
E R **Jobs** Check motor and wirings X1 axis	**X1 AXIS DRIVE**
741: OVERLOAD ALARM: AXIS 11	
NA	
742: OVERLOAD ALARM: AXIS 12	
NA	
743: OVERLOAD ALARM: AXIS 13	
NA	

END

MOTORS

	NAME	DESCRIPTION	PROTECTION	POSITION	PAGE
1	M302	HIGH PRESSURE COOLANT PUMP MOTOR	QM302	h.p. tank	1 (+a.p.)
2	M304	DRUM ROTATION PUMP	QM304	h.p. tank	1 (+a.p.)
3	M305	DRUM CLEANING PUMP MOTOR	QM305	h.p. tank	1 (+a.p.)
4	M313	RIGHT LONGITUDINAL CHIP CONVEYOR	QM313	right long.chip conv.	1 (+t.t.l.dx)
5	M314	LEFT LONGITUDINAL CHIP CONVEYOR	QM314	left long.chip conv.	1 (+t.t.l.sx)
6	M320	RECYCLING PUMP 1 MOTOR	QM320	recycl.pump 1	1 (+p.ril.nr.1)
7	M321	RECYCLING PUMP 2 MOTOR	QM321	recycl.pump 2	1 (+p.ril.nr.2)
8	M701	HYDRAULIC POWER UNIT PUMP MOTOR	QM701	service cabinet	2 (+C.I.)
9	M704	HYDRAULIC UNIT FAN MOTOR	FA704	service cabinet	2 (+C.I.)
10	MA1	COLLECTOR SIDE A AXIS MOTOR	HEIDENHAIN	head	68
11	MA2	CABLE SIDE A AXIS MOTOR	HEIDENHAIN	head	68
12	MAC	C AXIS MOTOR	HEIDENHAIN	head	30 (+H.D.)
13	MM	ELECTROSPINDLE FISCHER	HEIDENHAIN	head	70

TOOL MAGAZINE MOTORS

	NAME	DESCRIPTION	PROTECTION	POSITION	PAGE
1	MAu	u AXIS (CHAIN) MOTOR	HEIDENHAIN	Tool mag. 1	5 (+m.ut.1)

LINEAR MOTORS

	NAME	DESCRIPTION	PROTECTION	POSITION	PAGE
1	MXM	X AXIS MOTOR MASTER	HEIDENHAIN	crossbeam	9 (+H.D.)
2	MXS	X AXIS MOTOR STROKER	HEIDENHAIN	crossbeam	9 (+H.D.)
3	MX1M	X1 AXIS MOTOR MASTER	HEIDENHAIN	crossbeam	15 (+H.D.)
4	MX1S	X1 AXIS MOTOR STROKER	HEIDENHAIN	crossbeam	15 (+H.D.)
5	MYM	Y AXIS MOTOR MASTER	HEIDENHAIN	cross saddle	21 (+H.D.)
6	MYS	Y AXIS MOTOR STROKER	HEIDENHAIN	cross saddle	21 (+H.D.)
7	MZ	Z AXIS MOTOR	HEIDENHAIN	cross saddle	26 (+H.D.)

SOLENOID VALVES

	NAME	DESCRIPTION	PLC OUTPUT	PLC POSITION	ELEMENT POSITION	PAGE
1	QY143	COMMAND AUTOMATIC DOOR'S OPENING	A155.0	Tool mag. 1	Tool mag. 1	2 (+m.ut.1)
2	QY144	COMMAND AUTOMATIC DOOR'S CLOSING	A155.1	Tool mag. 1	Tool mag. 1	2 (+m.ut.1)
3	QY150	COMMAND MAGAZINE BACKWARDS	A153.1	Tool mag. 1	Tool mag. 1	1 (+m.ut.1)
4	QY151	COMMAND MAGAZINE FORWARDS	A153.0	Tool mag. 1	Tool mag. 1	1 (+m.ut.1)
5	QY380	SPRAY MIST LINE DUCT CLEANING TOOL EXTERNAL COOLANT	A188.1	service cabinet	service cabinet	64
6	QY381	INNER TOOL COOLANT	OUT1	ASI ADR21	h.p. tank	2 (+a.p.)
7	QY382	COOLANT FOR OUTER NOZZLES	OUT2	ASI ADR21	h.p. tank	2 (+a.p.)
8	QY384	SPRAY MIST LINE DUCT CLEANING TOOL INTERNAL COOLANT	A186.0	service cabinet	service cabinet	64
9	QY424	TOOL COOLET CLOSED CLAMPING COMMAND	Q131.1	ASI ADR7	crossbeam	56
10	QY426	TOOL RELEASE 100BAR	Q132.6	ASI ADR8	crossbeam	57
11	QY462	MAIN CLAMPING VALVE FISCHER SYSTEM	A189.1	service cabinet	service cabinet	64
12	QY477	AIR LUBRICATION FISCHER SYSTEM	A189.0	service cabinet	service cabinet	64
13	QY501	BRAKE X-AXIS (ZIMMER)	Q132.1	ASI ADR9	crossbeam	58
14	QY502	BRAKE Y-AXIS (ZIMMER)	Q132.2	ASI ADR9	crossbeam	58
15	QY504	C-AXIS CLAMPING	Q132.4	ASI ADR8	crossbeam	57
16	QY505	A AXIS CLAMPING	Q132.5	ASI ADR8	crossbeam	57
17	QY509	BRAKE X1-AXIS (ZIMMER)	Q132.0	ASI ADR9	crossbeam	58
18	QY521	COMMAND RELEASE Z-AXIS	A34.3	electric cabinet	crossbeam	58
19	QY522	DRIVE RECHARGE ACCUMULATOR ZIMMER Z-AXIS	Q137.4	ASI ADR18	cross saddle	60
20	QY725	START RECHARGE PRESSURE FOR RECHARGE ZIMMER Z AXIS	A183.0	service cabinet	service cabinet	64
21	QY749	SPRAY-MIST AIR/OIL OUTER TOOL	Q133.0	ASI ADR11	crossbeam	1 (+s.mist)
22	QY756	DUCT CLEANING EXTERNAL TOOL AIR BLOW	A187.1	service cabinet	service cabinet	64
23	QY773	TOOL TAPER CLEANING	A183.1	service cabinet	service cabinet	64
24	QY777	AXES LUBRICATION PUMP	A187.0	service cabinet	service cabinet	64
25	QY778	GENERAL AIR	A184.1	service cabinet	service cabinet	64
26	QY782	Z AXIS ACCUMULATOR RECHARGEMENT	Q131.2	ASI ADR7	crossbeam	56
27	QY786	SPRAY-MIST OIL INNER TOOL	Q133.1	ASI ADR11	crossbeam	1 (+s.mist)
28	QY811	AIR BARRIER LASER BLUM	A185.0	service cabinet	service cabinet	2 (+blum)
29	QY812	SPEAR VALVE LASER BLUM	A185.1	service cabinet	service cabinet	2 (+blum)
30	QY820	C AND A AXES BRAKE CLAMPING SAFETY	Q133.5	ASI ADR10	crossbeam	58.1

SOLENOID VALVES

	NAME	DESCRIPTION	PLC OUTPUT	PLC POSITION	ELEMENT POSITION	PAGE
31	QY821	RECHARGE ACCUMULATOR FOR BRAKE A-AXIS AND C-AXIS	Q133.6	ASI ADR10	crossbeam	58.1
32	QY822	RELEASE CHECK BRAKE A-AXIS	Q133.4	ASI ADR10	crossbeam	58.1
33	QY823	C AXIS BRAKE RELEASE UNCLAMPING	Q131.0	ASI ADR7	crossbeam	56

MICRO - SENSORS - PROXIMITY SWITCHES

	NAME	DESCRIPTION	INPUT PLC	POS. PLC	EL.POSIZION	PAGE
1	BS091	electric cabinet door switch	-	-	electric cabinet	20
2	BS092	electric cabinet door switch	-	-	electric cabinet	20
3	BS093	electric cabinet door switch	-	-	electric cabinet	37 (+H.D.)
4	BS143	AUTOMATIC DOOR OPEN	E148.0	Tool mag. 1	Tool mag. 1	1 (+m.ut.1)
5	BS144	AUTOMATIC DOOR CLOSED	E148.1	Tool mag. 1	Tool mag. 1	1 (+m.ut.1)
6	BS150	MAGAZINE BACKWARDS	E151.0	Tool mag. 1	Tool mag. 1	1 (+m.ut.1)
7	BS151	MAGAZINE FORWARDS	E150.1	Tool mag. 1	Tool mag. 1	1 (+m.ut.1)
8	BS167	UPPER PLACE IN MAGAZINE OCCUPIED	E151.1	Tool mag. 1	Tool mag. 1	3 (+m.ut.1)
9	BS169	TOOL MAGAZINE MANUAL DOOR CLOSE	-	-	Tool mag. 1	2 (+m.ut.1)
10	BS313	RIGHT LONGITUDINAL CHIP CONVEYOR CLOGGED	IN2	ASI ADR27	ght long.chip con	1 (+t.t.l.dx)
11	BS314	LEFT LONGITUDINAL CHIP CONVEYOR CLOGGED	IN2	ASI ADR28	eft long.chip conv	1 (+t.t.l.sx)
12	BS589	Z-AXIS NOT IN ENCUMBRANCE FOR T.C. + Z-AXIS IN RECHARGE POSITION	E136.3	ASI ADR17	cross saddle	59
13	BS590	Y-AXIS NOT IN THE WAY WITH T.CHANGE	E136.2	ASI ADR17	cross saddle	59
14	BS591	X AXIS NOT ENCUMBRANCE WITH JIMS MAGAZINE/OVERHEAD CRANE	E131.2	ASI ADR7	crossbeam	56
15	BS703	X AXIS LUBRICATION CYCLE END (RIGHT SIDE)	E133.1	ASI ADR11	crossbeam	1 (+s.mist)
16	BS711	OIL FILTER DIRTY	E176.0	service cabinet	service cabinet	62
17	BS751	ELECTRIC LOCK OPERATOR DOOR	-	-	door 1	66
18	BS752	DOOR ELECTRIC LOCK 2	-	-	door 2	67
19	BSA+	A AXIS EMERGENCY +	IN1	ASI ADR2	head	74
20	BSA-	A AXIS EMERGENCY -	IN1	ASI ADR2	head	74
21	BSC+	C AXIS LIMIT SWITCH +	E33.5	electric cabinet	head	61
22	BSC-	C AXIS LIMIT SWITCH -	E33.6	electric cabinet	head	61
23	BSD06	X1 AXIS LUBRICATION CYCLE END (LEFT SIDE)	E133.2	ASI ADR11	crossbeam	1 (+s.mist)
24	BSD07	MICRO OF END LUBRICATION CYCLE Y/Z-AXES ON RIGHT SIDE	E137.6	ASI ADR18	cross saddle	60
25	BSD08	MICRO OF END LUBRICATION CYCLE Y/ZAXES ON LEFT SIDE	E137.7	ASI ADR18	cross saddle	60
26	BSRA	A AXIS RESET	E32.4	electric cabinet	head	74
27	BSRC	C AXIS RESET	E32.3	electric cabinet	head	61
28	BSRT1	T1 AXIS RESET	E150.0	Tool mag. 1	Tool mag. 1	3 (+m.ut.1)
29	BSRX	X AXIS RESET	E131.0	ASI ADR7	crossbeam	56
30	BSRX1	X1 AXIS RESET	E131.1	ASI ADR7	crossbeam	56

COMPONENT LOCATION TABLE - J1294 - LINX COMPACT 30

	NAME	DESCRIPTION	INPUT PLC	POS. PLC	EL.POSIZION	PAGE
31	BSRY	END OF STROKE RESET Y-AXIS OPTION	E136.0	ASI ADR17	cross saddle	59
32	BSRZ	END OF STROKE RESET Z-AXIS OPT	E136.1	ASI ADR17	cross saddle	59
33	BSX	X AXIS LIMIT SWITCH	IN1	ASI ADR4	crossbeam	56
34	BSX1	X1 AXIS LIMIT SWITCH	IN2	ASI ADR4	crossbeam	56
35	BSY	Y AXIS LIMIT SWITCH	IN1	ASI ADR5	cross saddle	59
36	BSZ+	Z AXIS LIMIT SWITCH+	IN2	ASI ADR5	cross saddle	59
37	BSZ-	Z AXIS LIMIT SWITCH-	IN1	ASI ADR6	cross saddle	59

PRESSURE SWITCHES

	NAME	DESCRIPTION	INPUT PLC	POS. PLC	EL.POSITION	PAGE
1	BP462	PRESSURE OK FISCHER SYSTEM	E190.0	service cabinet	service cabinet	64
2	BP494	FISCHER UNIT AIR PRESSURE SWITCH	-	-	head	74
3	BP495	FISCHER UNIT OIL PRESSURE SWITCH	-	-	head	74
4	BP501	BRAKE X-AXIS ZIMMER	E132.0	ASI ADR9	crossbeam	58
5	BP502	BRAKE Y-AXIS ZIMMER	E132.2	ASI ADR9	crossbeam	58
6	BP504	C-AXIS CLAMPING	E132.4	ASI ADR8	crossbeam	57
7	BP505	A-AXIS CLAMPING	E132.6	ASI ADR8	crossbeam	57
8	BP509	BRAKE X1-AXIS ZIMMER	E132.1	ASI ADR9	crossbeam	58
9	BP521	PRESSURE RELEASE (ZIMMER) Z AXIS	E137.4	ASI ADR18	cross saddle	60
10	BP522	RECHARGE ACCUMULATOR ZIMMER X AXIS	E137.5	ASI ADR18	cross saddle	60
11	BP554	C-AXIS RELEASE	E132.5	ASI ADR8	crossbeam	57
12	BP555	A-AXIS RELEASE	E132.7	ASI ADR8	crossbeam	57
13	BP778	AIR UNDER PRESSURE	E178.0	service cabinet	service cabinet	62
14	BP782	Z-AXIS ACCUMULATOR RECHARGE MENT	E131.3	ASI ADR7	crossbeam	56
15	BP783	SAFETY PRESSURE Z-AXIS BALANCING	E34.4	elect. Cabinet	crossbeam	58
16	BP798	HYDRAULIC SERVICES	E177.1	service cabinet	service cabinet	62
17	BP821	PRESSURE OK ACCUMULATOR FOR A AXIS BRAKE	E133.4	ASI ADR10	crossbeam	58.1

FLOWMETER

	NAME	DESCRIPTION	INPUT PLC	POS. PLC	EL.POSITION	PAGE
1	BC	FREE	E182.1	service cabinet	service cabinet	63
2	BC527	FISCHER EL.SPINDLE COOL.LIQUID FLOWMETER	E190.1	service cabinet	service cabinet	64
3	BC528	FLOWMETER CONTROL LINE PRIMARY MOTORS ACCURACY X-AXIS	E179.1	service cabinet	service cabinet	62
4	BC531	FLOWMETER CONTROL LINE PRIMARY MOTOR ACCURACY Y-AXIS	E180.1	service cabinet	service cabinet	62
5	BC533	FLOWMETER CONTROL LINE PRIMARY MOTORS ACCURACY Z-AXIS	E181.0	service cabinet	service cabinet	63
6	BC534	FLOWMETER CONTROL LINE PRIMARY MOTORS ACCURACY X1-AXIS	E180.0	service cabinet	service cabinet	62
7	BC546	FLOWMETER CONTROL LINE SECONDARY MOTOR X/X-AXIS1	E182.0	service cabinet	service cabinet	63
8	BC547	FLOWMETER CONTROL LINE SECONDARY MOTORS Y/Z-AXIS	E181.1	service cabinet	service cabinet	63
9	BC550	A-AXIS MOTOR DELIVERY LINE CONTROL FLOWMETER	E179.0	service cabinet	service cabinet	62
10	BC551	FLOWMETER CONTROL LINE A1-AXIS MOTOR	E191.0	service cabinet	service cabinet	64
11	BC552	C-AXIS MOTOR RETURN LINE CHECK FLOWMETER	E191.1	service cabinet	service cabinet	64

LEVEL SWITCHES

	NAME	DESCRIPTION	INPUT PLC	POS. PLC	EL.POSITION	PAGE
1	BL362	PRE-ALARM SUCTION FILTER OBSTRUCTED	IN2	ASI ADR21	h.p. tank	2 (+a.p.)
2	BL388	COOLANT TANK MINIMUM LEVEL	IN4	ASI ADR22	h.p. tank	3 (+a.p.)
3	BL388	RECYCLING PUMP 1 MINIMUM LEVEL	IN3	ASI ADR30	recycl.pump 1	1 (+p.ril.nr.1)
4	BL388	RECYCLING PUMP 2 MINIMUM LEVEL	IN4	ASI ADR31	recycl.pump 2	1 (+p.ril.nr.2)
5	BL389	COOLANT TANK MAXIMUM LEVEL	IN3	ASI ADR22	h.p. tank	3 (+a.p.)
6	BL389	RECYCLING PUMP 1 MAXIMUM LEVEL	IN4	ASI ADR30	recycl.pump 1	1 (+p.ril.nr.1)
7	BL389	RECYCLING PUMP 2 MAXIMUM LEVEL	IN3	ASI ADR31	recycl.pump 2	1 (+p.ril.nr.2)
8	BL396	DRUM FILTER LEVEL OK	IN1	ASI ADR21	h.p. tank	2 (+a.p.)
9	BL796	HYDRAULIC SERVICES OIL LEVEL	E177.0	service cabinet	service cabinet	62
10	BL797	AXES LUBRICATION GREASE MIN LEVEL	E178.1	service cabinet	service cabinet	62
11	BLA41	MIN.OIL LEVEL SPRAY-MIST OIL OUTER/INNER TOOL	E133.0	ASI ADR11	crossbeam	1 (+s.mist)

THERMOSTATS

	NAME	DESCRIPTION	INPUT PLC	POS. PLC	EL.POSITION	PAGE
1	BR704	THERMOSTAT HYDRAULIC UNIT COOLING VALVE	-	-	hydraulic unit	2 (+C.I.)
2	BR709	HYDRAULIC SYSTEM THERMOSTAT	E176.1	service cabinet	hydraulic unit	62
3	STEQ	ELECTRIC CABINET THERMOSTAT	E32.1	elect. Cabinet	elect. Cabinet	34

SELECTORS

	NAME	DESCRIPTION	PLC INPUT	POS. PLC	EL.POSITION	PAGE
1	SL 100	Emergency by-pass selector	IN1	ASI ADR3	el.cabinet	23

PROGRAMMING MANUAL
MEASURING CYCLES

P87.0634-030.330 / P87.0634-030.410 / VERSION V5F / ENGLISH

CONTROL: HEIDENHAIN TNC 426/430 / iTNC 530

SINCE SOFTWARE VERSION **TNC:** 280 476 04 / **iTNC:** 340 420 01

LASER SYSTEMS MINI - MICRO - NANO

Laser System for Tool Setting and Tool Breakage Detection

Important notice!

Read programming and operating manual for BLUM Laser system carefully before using the measuring system and the cycles!

The measuring cycles version V5F* are designed for Heidenhain control TNC 426/430 since software version 280 476 04 and iTNC 530 since software version 340 420 01. The measuring cycles can be used with all Blum laser systems incl. the Blum laser NT systems.

The contents of the floppy disk and the related documentation (together called „data") are protected by copyright. BLUM-Novotest GmbH reserves all rights on the data or parts thereof, especially the right of mechanical and electronical reproduction, lending out, leasing, modification, recording and processing in numerical controls or other electronic systems (such as PCs). Passing on as well as reproduction of the data or their use on more than one CNC requires the express authorization of BLUM-Novotest GmbH.

Contraventions are subject to damages.

All data were carefully checked. Nevertheless, no guarantee can be given for completeness, correctness and errors. The indications in this document are subject to regular control and required corrections will be contained in the following issues.

BLUM-Novotest GmbH is not liable for damages on the machine due to programming errors in the measuring cycles, faulty applications or modifications made after commissioning of the measuring cycles. No guarantee can be given for faulty performance of the measuring cycles due to control errors of the TNC.

Installing the measuring cycles in the numerical control of the machine implies that the above mentioned warranty exclusions are recognized.

We reserve the right of technical modifications, which improve the product. All suggestions for improvement are gladly accepted.

A detailed installation and programming manual for the BLUM Laser system is available in German and English. Programming manuals are also available in other languages upon request.

BLUM-Novotest GmbH
Production Metrology
Kaufstrasse 14
D-88287 Gruenkraut-Gullen / GERMANY

Tel.: +49-(0)751-6008-0
Fax: +49-(0)751-6008-156
Mail: info@blum-novotest.com
Mail: snc@blum-novotest.com

BLUM-Novotest Ltd.
Production Metrology
33 Townfields, Lichfield
Staffordshire WS13 8AA / GREAT BRITAIN

Tel.: +44-(0)1543-257111
Fax: +44-(0)1543-251746
Mail: info@blum-novotest.co.uk

BLUM LMT, Inc.
Production Metrology
250 Grandview Drive, Suite 10
Ft. Mitchell, KY 41017 / USA

Tel.: +1-(859)-344-6789
Fax: +1-(859)-344-6799
Mail: solutions@blumlmt.com

Contents

1 General Issues

The BLUM Laser measuring system is a precise optical measuring device for automated tool measuring inside the working area of a machining centre under operating conditions. Measuring the tool geometry makes it possible to recognize incorrectly loaded tools or inaccurately preset tools in time, as well as worn or damaged tools. This helps to avoid further damages to the following tool or to the part.

1.1 Application Possibilities

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

- Calibration of the laser system
- Measurement of tool length and radius
- Compensation of the temperature related drift
- Cutting edge monitoring (cutting edge breakage) for straight / round cutting edges
- Tool breakage detection (shank breakage)

1.2 Program Overview

Following files are stored on the PLC partition of the control:

PLC:\BLUM\BLUM_000.TAB	Parameter table for machine specific definitions

Following files are stored on the TNC partition of the control:

TNC:\BLUM\BLUM581.H	Example program for cycle 581
TNC:\BLUM\BLUM582.H	Example program for cycle 582
TNC:\BLUM\BLUM583.H	Example program for cycle 583
TNC:\BLUM\BLUM584.H	Example program for cycle 584
TNC:\BLUM\BLUM585.H	Example program for cycle 585
TNC:\BLUM\BLUM586.H	Example program for cycle 586
TNC:\BLUM\BLUM587.H	Example program for cycle 587
TNC:\BLUM\BLUM588.H	Example program for cycle 588
TNC:\BLUM\E_588.TAB	Result file for cycle 588, Q366≠0
TNC:\BLUM\INFO.A	Information about Blum software version

The program examples show, which parameters must be defined for the call of the measuring cycles and how the measuring cycles must be called out of a machining program.

Following files are optional (i.e. not official part of Blum measuring cycles and therefore no manual / description in national language available) and can be passed to the customer by the manufacturer if desired. A short program description is available in the header of the program:

TNC:\BLUM\IDENTIFY.H	Tool identification program with tolerance supervision
TNC:\BLUM\S_LASPOS.H	Auxiliary program to determine the measuring positions in the laser beam
TNC:\BLUM\S_PARA.H	Auxiliary program to change the parameters in BLUM_000.TAB
TNC:\BLUM\S_RUNOUT.H	Auxiliary program to determine the radial run-out on the reference tool
TNC:\BLUM\S_SWITCH.H	Auxiliary program to set the soft end switch positions in BLUM_000.TAB
TNC:\BLUM\T_REPEAT.H	Test program for repetitive measurement
TNC:\BLUM\T_SIGNAL.H	Test program to check the input/output signals of the laser system
TNC:\BLUM\TOOLSETTING	Tool setting of all tools in the tool magazine

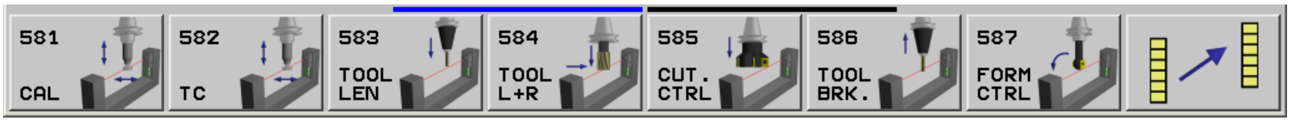
Following files are optional (i.e. not official part of Blum measuring cycles and therefore no manual / description in national language available) and can be passed to the customer by the manufacturer after a **machine specific modification**, if desired.

TNC:\BLUM\BREAKAGE.H	Selects the correct program FAST_58*.H for breakage detection, depending on the tool type (drill, mill) and the tool parameter (R>4mm, RBREAK≠0, LOFFS, CUT≠0)
TNC:\BLUM\FAST_585.H	Fast single cutting edge monitoring (about 50% faster than cycle 585) usage only for simple tools (ball nose mills, end mills, face mills, etc.)
TNC:\BLUM\FAST_586.H	Fast tool shank breakage detection (about 60% faster than cycle 586) usage only for simple tools (drills, thread drills, ball nose mills, end mills)

The cycles are selected by following buttons and inserted into a machining program:

- Key "TOUCH PROBE" (on the operating panel)
- Softkey symbol "BLUM LaserControl" (1^{st}. softkey level, 2^{nd}. softkey line)

- Softkey symbols with cycle numbers "581...587" (2^{nd}. softkey level, 1^{st}. softkey line)



- Softkey symbols with cycle numbers "588...xxx" (2^{nd}. softkey level, 2^{nd}. softkey line)

<u>Following BLUM measuring cycles are available on the TNC control:</u>

581.CYC	Measuring cycle calibration in radial and axial axis
582.CYC	Measuring cycle temperature compensation in radial and axial axis
583.CYC	Measuring cycle length measurement of centric tools
584.CYC	Measuring cycle length and radius measurement of eccentric tools
585.CYC	Measuring cycle single cutting edge monitoring at straight cutting edge geometry
586.CYC	Measuring cycle tool breakage detection (shank breakage)
587.CYC	Measuring single cutting edge monitoring at round cutting edge geometry
588.CYC	Measuring cycle tool length measurement referring to circle centre and edge radius measurement on rounded cutting edge geometry

<u>Following auxiliary programs are available on the TNC control:</u>

578.H	Auxiliary program executed at the cycle beginning (commands to be executed are defined by the machine manufacturer)
579.H	Auxiliary program executed at the cycle end (commands to be executed are defined by the machine manufacturer)

1.3 Program levels and Program nesting

The handing over of Q-parameter values from the machining program (program level 1) to the cycle (program level 2) or to the auxiliary programs 578.H and/or 579.H (program level 3) or from 578.H to 579.H is only possible via the global Q-parameters Q200…Q399, but not via the local Q-parameters Q0…Q99.

Example for the handing over of Q-parameter values:

```
BEGIN PGM Machining MM                  (Program level 1)
…
Q0 = 5
Q200 = 1
TCH PROBE 583 TOOL LENGTH
```

↓

```
Intermediate storage of parameter Q0…Q99 on the stack
```

↓

```
BEGIN PGM 583 MM                        (Program level 2)
…
Q0 = 4711
Q197 = 583
CYCL DEF 12.0 PGM CALL
CYCL DEF 12.1 PGM  578.H   ;execution at cycle start
M99
```

↓

```
BEGIN PGM 578 MM                        (Program level 3)
…
FN 9: IF Q197 EQU 583 GOTO LBL xxx
FN 9: IF Q200 EQU 1 GOTO LBL xxx  ;contents Q200 = 1!
…
END PGM 578 MM
```

↓

```
; PGM 583                               (Program level 2)
…
FN 9: IF Q0 EQU 4711 GOTO LBL xxx ;contents Q0 = 4711!
…
Q197 = 583
CYCL DEF 12.0 PGM CALL
CYCL DEF 12.1 PGM  579.H   ;execution at cycle end
M99
```

↓

```
BEGIN PGM 579 MM                        (Program level 3)
…
FN 9: IF Q197 EQU 583 GOTO LBL xxx
FN 9: IF Q200 EQU 1 GOTO LBL xxx  ;contents Q200 = 1!
…
END PGM 579 MM
```

↓

```
; PGM 583                               (Program level 2)
…
END PGM 583 MM
```

↓

```
Restoring of parameter Q0…Q99 from the stack
```

↓

```
; PGM Machining                         (Program level 1)
…
FN 9: IF Q0 EQU 5 GOTO LBL xxx ;contents Q0 = 5!
…
END PGM Machining MM
```

1.4 Parameter Overview

When starting the measuring cycle the parameter table PLC:\BLUM\BLUM_000.TAB is read and the system-specific data are loaded in Q-parameter. During commissioning the parameters must be assigned with valid values by the machine manufacturer. The parameters Q0...Q99 are internal, local Q-parameters and only valid within the measuring cycles. The external, global Q-parameters with the same Q-number are neither changed nor overwritten by the measuring cycles (condition MP7251=0).

1.4.1 Description of the cycle parameters

Reserved Q-parameters for special functions of the TNC:

Q-Para	Definition / Meaning
Q185	Temperature compensation value in the X-axis
Q186	Temperature compensation value in the Y-axis
Q187	Temperature compensation value in the Z-axis
Q197	Actual cycle number *Handing over of the actual running cycle number to the auxiliary programs 578.H and 579.H* *Value range: 581 <= Q197 <= 588*
Q199	Tool status parameter for tool breakage, tool wear and cutting edge control (set in the cycle) *Q199 = 0 → tool dimension within the permitted tolerance LTOL / RTOL, LBREAK / RBREAK tool enabled (TL= without registration)* *Q199 = 1 → tool dimension out of permitted tolerance LTOL / RTOL tool locked (TL=Tool Locked)* *Q199 = 2 → tool dimension out of permitted tolerance LBREAK / RBREAK tool locked (TL=Tool Locked)*

Global Q-parameters:

Q-Para	Definition / Meaning
Q348	Nominal tool length *At postprocessor applications the travel motion of the tool is often calculated for a preset tool length. As minimum travel compensation only the deviation to this set length can be determined and entered into the tool table. If the tools are used according to this method the predefined set length must be transferred to the Blum cycle during each cycle call in parameter Q348.* *Connection: $L_{Real} = L_{TOOL} + DL_{TOOL} + Q90 + Q91 * Q348$* *with Q90 = additive tool length compensation (Q90 = parameter line 64 in BLUM_0**.TAB)* *with Q91 * Q348 = set tool length from postprocessor application (Q91 = parameter line 66)* *Value range: -999.999 mm <= Q348 <= +999.999 mm* *Default value: Q348 = 0.000 mm [input only in mm, no inch values allowed!]*
Q349	Nominal tool radius *At postprocessor applications the travel motion of the tool is often calculated for a preset tool radius. As minimum travel compensation only the deviation to this set radius can be determined and entered into the tool table. If the tools are used according to this method the predefined set radius must be transferred to the Blum cycle during each cycle call in parameter Q349.* *Connection: $R_{Real} = R_{TOOL} + DR_{TOOL} + Q92 + Q93 * Q349$* *with Q92 = additive tool radius compensation (Q92 = parameter line 65 in BLUM_0**.TAB)* *with Q93 * Q349 = set tool radius from postprocessor application (Q93 = parameter line 67)* *Value range: -999.999 mm <= Q349 <= +999.999 mm* *Default value: Q349 = 0.000 mm [input only in mm, no inch values allowed!]*

Call Q-parameters:

Q-Para	Definition / Meaning
Q350	Measuring mode *Determination, if the tool must be measured initially or if an already measured tool should be checked with respect to wear.* *At tool measurement (Q350=0/±3) the cycle calculates the real actual value for tool length/ radius, overwrites the value L/R in the tool memory TOOL.T and sets tool compensation DL/DR=0.* *During tool check (Q350=±1) the measured actual value for tool length/radius is compared with the set value L/R of tool memory TOOL.T. The cycle calculates the deviation with the correct signs and notes it as tool compensation DL/DR in TOOL.T. If the Delta-value is larger than the permitted wear tolerance LTOL/RTOL, the tool is locked (status TL=L in TOOL.T).* *At tool control (Q350=±2) the measured actual value for tool length/radius is compared with the set value L/R of tool memory TOOL.T. If the difference is larger than the permitted wear tolerance LTOL/RTOL, the tool is locked (status TL=L in TOOL.T). The tool data L/R resp. DL/DR are <u>not</u> updated.* *Attention: The measuring mode Q350=±1/±2 (checking/control) is automatically set to Q350=0/+3 (measuring), if condition L_{min} < (L+DL) < L_{max} or R_{min} < (R+DR) < R_{max} is not fulfilled.* *At measuring mode Q350= 0/+1/+2 the tool length resp. tool radius within the given tool limit data is first roughly determined with a search function and then measured (→ low collision risk at wrong tool data). The measurement is carried out from light to dark; coolant drops may disturb the measuring sequence. Such disturbances can be corrected in the cycle by means of plausibility checks and repetition measurements.* *At measuring mode Q350= -1/ -2/ -3 the tool is – referring to the tool data L + R as per the tool table – directly positioned in the laser beam in rapid – without search function – (→ increased collision risk at wrong tool data). The measurement is carried out from dark to light; coolant drops will scarcely influence the measuring sequence and the measuring accuracy (NT measuring mode).* *At measuring mode Q350= +3 the tool length resp. tool radius within the given tool limit data is first roughly determined with a search function (→ low collision risk at wrong tool data). The measurement is carried out from dark to light; coolant drops will scarcely influence the measuring sequence and the measuring accuracy (combination of search function and NT measuring mode).* *<u>Cycle 583, 584, 588:</u>* *<u>Pushing measuring strategy into the beam with search function:</u>* *Q350 = 0 → measuring with search function (default value)* *Q350 = +1 → checking (condition: L_{min} < (L+DL) < L_{max} or R_{min} < (R+DR) < R_{max})* *Q350 = +2 → control (condition like Q350=+1), tool data are <u>not</u> updated* *<u>Pulling measuring strategy out of the beam with search function:</u>* *Q350 = +3 → measuring with search function* *<u>Pulling measuring strategy out of the beam without search function:</u>* *(condition: tool setting within ±5 mm accuracy !)* *Q350 = -1 → checking (condition: L_{min} < (L+DL) < L_{max} or R_{min} < (R+DR) < R_{max})* *Q350 = -2 → control (condition like Q350=-1), tool data are <u>not</u> updated* *Q350 = -3 → measuring without search function* *Value range: -3 <= Q350 <= +3* *Default value: Q350 = 0*

Overview measuring mode for cycle 583,584,588			
Q350	**Description**	**Meas. strategy**	**Tool data**
-3	*Measuring* *in NT mode*	*Positioning of the tool directly in the laser beam*	• *Length / radius must be preset (±5mm)* • *Length L / radius R are updated* • *Wear DL=0 / DR=0 are set* • *Tool is not evaluated Q199=0*
-2	*Control* *in NT mode*	*Positioning of the tool directly in the laser beam*	• *Length / radius must be measured* • *Length L / radius R are not updated* • *Wear DL / DR are not updated* • *Tool is evaluated Q199=0/1/2*
-1	*Check* *in NT mode*	*Positioning of the tool directly in the laser beam*	• *Length / radius must be measured* • *Length L / radius R are not changed* • *Wear DL / DR are updated* • *Tool is evaluated Q199=0/1/2*
±0	*Measuring*	*Search function to determine the tool data*	• *Length / radius can be unknown* • *Length L / radius R are updated* • *Wear DL=0 / DR=0 are set* • *Tool is not evaluated Q199=0*
+1	*Check*	*Search function to determine the tool data*	• *Length / radius must be measured* • *Length L / radius R are not changed* • *Wear DL / DR are updated* • *Tool is evaluated Q199=0/1/2*
+2	*Control*	*Search function to determine the tool data*	• *Length / radius must be measured* • *Length L / radius R are not updated* • *Wear DL / DR are not updated* • *Tool is evaluated Q199=0/1/2*
+3	*Measuring* *with search function in NT mode*	*Search function to determine the tool data*	• *Length / radius can be unknown* • *Length L / radius R are updated* • *Wear DL=0 / DR=0 are set* • *Tool is not evaluated Q199=0*

Notice:

The following minimum spindle rotation speeds must be defined in the TOOL CALL instruction, if the measuring mode is defined as Q350= -3, Q350= -2, Q350= -1, Q350= +3 (Positioning of the tool directly in the laser beam), otherwise the measuring mode will be changed automatically to search function:

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

Q-Para	Definition / Meaning
Q351	Scope of measurement *Determination, which tool geometry should be measured (part or complete measurement).* *Q351 = 0 → measurement of length and radius (default value)* *Q351 = 1 → length measurement only* *Q351 = 2 → radius measurement only (condition: L_{min} < (L+DL) < L_{max})* *Value range: 0 <= Q351 <= 2* *Default value: Q351 = 0*
Q352	Cutting edge monitoring ±1=yes / 0=no *Cycle 584:* *Determination, if single cutting edge monitoring should be effected and the evaluation strategy (condition: 0 < (R+DR) < R_{max})* *Q352 = -1 → yes, effect single cutting edge monitoring with inverse evaluation (tool is broken, if DYN signal = high level, e.g. recommended for saw blades)* *Q352 = 0 → no, doesn't effect single cutting edge monitoring (valid only in cycle 584)* *Q352 = +1 → yes, effect single cutting edge monitoring with normal evaluation (tool is broken, if DYN signal = low level, e.g. recommended for end mills)* *Value range: -1 <= Q352 <= +1* *Default value: Q352 = 0* *Cycle 585, 587:* *With default value Q352=0 the cycle is performed in the same manner as Q352=+1*
Q353	Control travel *Cycle 585:* *Q365=0: Continuous control of a cutting range (e.g. milling cutter)* *The cutting edge control starts in a distance LOFFS from the tool tip. In order to control the radial tolerance of all cutting edges in the determined cutting range the Z-axis moves continuously by the travel (Q353) in negative axis direction.* *Q365>=1: Step by step control of a cutting edge range (e.g. thread milling cutter)* *The cutting edge control starts in a distance LOFFS from the tool tip (e.g. first tooth row on the thread milling cutter). During one revolution the radius tolerance on this cutting edge position is controlled. According to the number of measuring points (Q365) all further tooth rows are controlled concerning cutting edge breakage, each displaced by step width (Q353).* *The start position (LOFFS), the travel (Q353) and the number of measuring points (Q365) must be selected in a way that avoids collision with the measuring system. The max. immersion depth of the tool tip under the laser beam is LOFFS+Q365*Q353.* *At Q353=0 it is checked without displacement via 1 revolution, whether the signal falls from high to low.* *Value range: 0.000 mm <= Q353 + LOFFS <= 54.000 mm (MICRO)* *0.000 mm <= Q353 + LOFFS <= 20.000 mm (NANO)* *Default value: Q353 = 5.000 mm [0.200 inch]* *Cycle 587:* *The cutting edge monitoring starts on the tool tip and ends on the tool shank. In the first section the travel is done as circular path curve on the ball end or angle radius, in the second section it is done as straight, tangential path curve on the cylinder or taper shank. The travel (Q353) determines the length of way of the second straight section.* *Q353 = 0 → cutting edge monitoring on cutting radius only, not on tool shank* *Q353 > 0 → cutting edge monitoring on cutting radius and on the tool shank* *Value range: 0.000 mm <= Q353 + R2 <= 54.000 mm (MICRO)* *0.000 mm <= Q353 + R2 <= 20.000 mm (NANO)* *Default value: Q353 = 5.000 mm [0.200 inch]*

Q-Para	Definition / Meaning
Q354	Control feed (F) *During measurement the moved axes travel with the indicated control feed. The feed determines the step width of the cutting edge scanning.* *Example: step width = feed / rotation speed = 100 mm min^{-1} / 1250 min^{-1} = 0.08 mm* *i.e. each cutting edge is controlled with a resolution of 0.08 mm (for CUT=3, t=0.02 s).* *Value range: 1 mm/min <= Q354 <= 1000 mm/min* *Default value: Q354 = 100 mm/min [4.0 inch/min]*
Q355	Measuring position *With the parameter measuring position different measuring sequences can be selected (e.g. one- or double-ended radius measurement) or different measuring tasks can be effected (e.g. length measurement at the lower tool edge (drill, shank milling cutter, etc.) or length measurement at the upper tool edge (disk milling cutter, groove milling cutter, etc.).* *Q355 = -2 → length measurement of lower edge of tool and double-ended radius meas.ment* *Q355 = -1 → length measurement of lower edge of tool and one-ended radius measurement* *Q355 = 0 → error message (input value not allowed)* *Q355 = +1 → length measurement of upper edge of tool and one-ended radius measurement* *Q355 = +2 → length measurement of upper edge of tool and double-ended radius measurement* *(double-ended radius meas. must be released in BLUM_0**.TAB by the machine manufacturer)* *Value range : -2 <= Q355 <= +2* *Default value: Q355 = -1*
Q356	Measuring direction *Measurement of the rough tool length during tool breakage detection can be from dark to light out of the beam or from light to dark into the beam. For large quantities of coolant or tools with large core diameter > ∅1mm it is recommended to measure from dark to light, engraving stylus or tools with small core diameter < ∅1mm or permanently error messages at Q356=+1 measurement from light to dark is necessary.* *Q356 = −1 → measuring direction form light to dark into the beam (normally Z− direction)* *Q356 = 0 → only monitoring, if beam is interrupted by tool tip; no measuring block* *Q356 = +1 → measuring direction from dark to light out of the beam (normally Z+ direction)* *Value range: -1 <= Q356 <= +1* *Default value: Q356 = +1*
Q357	Eccentric control position *For tools with centrically flowing out of internal coolant, e.g. internally cooled thread drills, the tool breakage detection can not be effected in the tool axis but must be effected outside of the influence of the disturbances. The control position for tool breakage detection is the relative, radial distance from the tool axis to the measuring position on the tool cutting edge (middle of the laser beam):* *Value range: 0.000 mm <= Q357 <= +999.999mm (condition: Q357 < R)* *Default value: Q357 = 0.000 mm*
Q358	Radial offset to the initial point of the angle radius *For a ball-ended milling cutter the cutting radius begins in the tool axis, for a cutter head with angle radius outside the tool axis. The radial offset is the relative distance from the tool axis to the initial point of the cutting edge radius.* *Q358 = 0 → initial point for the cutting edge radius is the tool axis* *(e.g. for cutting edge control on the ball-ended milling cutter)* *Q358 > 0 → initial point for the cutting edge radius is outside the tool axis* *(e.g. for cutting edge control on torus milling cutter)* *Value range: 0.000 mm <= Q358 <= +999.999mm (condition: Q358 < R)* *Default value: Q358 = 0.000 mm*

Q-Para	Definition / Meaning
Q359	Additive length compensation *The additive length compensation is added to the measured tool length, e.g. when it comes to mill flat surfaces with tolerances or when tool lengths are indicated and managed with respect to a reference tool length. Also the parameter can be used to compensate length deviations due to different cutting edge geometries.* *Value range: –99.999 mm <= Q359 <= +99.999 mm* *Default value: Q359 = 0.000 mm*
Q360	Additive radius compensation *The additive radius compensation is added to the measured tool radius, e.g. when it comes to mill flat surfaces with tolerances or when tool radius are indicated and managed with respect to a reference tool radius. Also the parameter can be used to compensate radius deviations due to different cutting edge geometries.* *Value range: –99.999 mm <= Q360 <= +99.999 mm* *Default value: Q360 = 0.000 mm*
Q361	Number of repeated measurements for averaging. *Value range: 1 <= Q361 <= 10* *Default value: Q361 = 3*
Q362	Permitted scattering out of tolerance for single measuring values in case of repeated measuring *In case of programmed repeated measurements (Q361 > 1), the difference between the biggest and the smallest value is compared with this tolerance. If it is out of tolerance, the cycle is aborted and an error message is displayed.* *Value range: 0.001 mm <= Q362 <= 0.100 mm* *Default value: Q362 = 0.010 mm [0.0004 inch]*
Q363	Starting angle Alpha to the tool axis *<u>Cycle 587:</u>* *Normally only one cutting edge of a ball-ended mill passes the tool centre point, all other cutting edges end in a defined angle outside of the tool centre point. Make sure that the starting angle Alpha is great enough to start the shape scanning at a position, where control of all cutting edges is guaranteed.* *The starting angle Alpha is the starting point for the first circular scanning section.* *Q363 = 0° → e.g. for cutting edge control for a spot facer with angle radius* *Q363 > 0° → e.g. for cutting edge control for a ball-ended milling cutter (Q363 < 90°)* *Q363 = 90° → e.g. for cutting edge control only at the tool shank, not at the angle radius* *Value range: 0° <= Q363 <= 360°* *Default value: Q363 = 25°* *<u>Cycle 588:</u>* *With starting angle Alpha the first measuring point for the measurement of the cutting radius R2 and of the circle centre lies on round cutting edge geometry.* *Value range: 0° <= Q363 <= 360°* *Default value: Q363 = 0°*

Q-Para	Definition / Meaning
Q364	Target angle Beta to the tool axis *<u>Cycle 587:</u>* *The cutting edge monitoring at the cutting edge radius is performed on a circular curve from the starting angle Alpha to the target angle Beta.* *The target angle Beta is the end point of the first circular scanning section and also the starting point for the second straight scanning section, which is tangentially attached to the first section.* *Q364 = Q363 → e.g. for cutting edge control only at the tool shaft, not at the angle radius* *Q364 = 60° → e.g. for cutting edge control for a ball-ended mill with conical tool shaft 30°* *Q364 = 90° → e.g. for cutting edge control for a ball-ended mill with cylindrical tool shaft* *Q364 = 180° → e.g. for cutting edge control on torus milling cutter* *Value range: Q363 <= Q364 <= 360°* *Default value: Q364 = 90°* *<u>Cycle 588:</u>* *With target angle Alpha the last measuring point for the measurement of the cutting radius R2 and of the circle centre lies on round cutting edge geometry.* *Value range: Q363 <= Q364 <= 360°* *Default value: Q364 = 90°*
Q365	Number of measuring points *<u>Cycle 585:</u>* *At cycle 585 the individual tooth rows of thread milling cutters e.g. are controlled concerning cutting edge breakage depending on the indicated number.* *Value range: 0 <= Q365 <= 20* *Default value: Q365 = 0* *<u>Cycle 588:</u>* *With cycle 588 the area between the starting angle (Q363) and the target angle (Q364) is divided into angle segments of the same size according to the given number of measuring points. Then the measuring points on the circle arc are touched in axial and radial direction and measured. As result the circle centre and the cutting edge radius R2 are determined.* *Value range: 3 <= Q365 <= 20* *Default value: Q365 >= 7*
Q366 optional	Expansion of functions *Determining, whether expansion of functions at cycle 587 and 588 for special forming tools (e.g. ball end mill, quarter-type bar milling cutter, semi-circular milling cutter, etc.) should be carried out:* *At activated expansion of functions the angle input at start and target angle (Q363, Q364) is possible from 0°...360° and the input of positive and negative cutting edge radii R2 in TOOL..T is allowed. Details of an axial offset Q367 to determine the circle centre with reference to the measured / entered tool length are obligatory.* *Q366 = 0 → Determination $L_{CC\ Reg}$,R, $R2_{Reg}$,$DR2_{Reg}$ ref. to regression radius (only pos. R2)* *Q366 = 1 → Determination $L_{CC\ Reg}$,R, $R2_{Reg}$,$DR2_{Reg}$ ref. to regression radius (pos. / neg. R2)* *Q366 = 2 → Determination $L_{CC\ Nom}$,R, $R2_{Nom}$,$DR2_{Nom}$ ref. to nominal radius (pos. / neg. R2)* *Value range: 0 <= Q366 <= 2* *Default value: Q366 = 0*

Q-Para	Definition / Meaning
Q367 optional	Axial offset to the centre circle of the cutting edge radius *At a ball-screw milling cutter the centre is offset by the cutting edge radius R2 above the tool tip. The axial offset is the sign-depending, relative distance from the reference point (cycle depending) to the circle centre of the cutting edge radius. The axial offset is evaluated only, when the function expansion Q366>=1 is active.* *<u>Cycle 587:</u>* *At cycle 587 the axial offset is the sign-depending, relative distance from the tool length L+DL (=reference point) as entered into the tool table up to the circle centre of the cutting edge radius.* *<u>Cycle 588:</u>* *At cycle 588, 587 the axial offset is the sign-depending, relative distance from the tool length to the tool tip (=reference point) as measured before at ROFF up to the circle centre of the cutting edge radius.* *Q367 < 0 → circle centre of the cutting edge radius R2 is with reference to measured / entered tool length in negative length direction (to the tool carrier reference point) (e.g. ball-screw milling cutter, torus milling cutter)* *Q367 = 0 → circle centre of the cutting edge radius R2 and measured / entered tool length are on the same height (pls. refer to examples cycle 587, 588)* *Q367 > 0 → circle centre of the cutting edge radius R2 is with reference to measured / entered tool length in positive length direction (away from the tool carrier reference point) (pls. refer to examples cycle 587, 588)* *Value range: -999.999 mm <= Q367 <= +999.999 mm* *Default value: Q367 = 0.000 mm*
Q368 Optional	Number of cutting edges for cutting edge control *The input value for the number of cutting edges in the tool table TOOL.T is limited to 0<=CUT<=20. To avoid this input limitation for tools with multiple edges (e.g. saw blade, separation disk, etc) it is possible to define in the cycle call the number of tool cutting edges which must be controlled. At Q368=0 the value of CUT from TOOL.T is used as cutting edge number, at Q368≠0 the absolute value ABS Q368 is used as cutting edge number. At plus sign of Q368 a normal evaluation is carried out, at minus sign an inverse evaluation is carried out. Example:* *Q368 >= 0 → normal evaluation: tool is broken, if dynamic signal = low* *Q368 < 0 → inverse evaluation: tool is broken, if dynamic signal = high* *Value range: -1 <= Q368 <= +360* *Default value: Q368 = 0* *Q368 = -1 → cutting edge control on a separation disk (with a lot of short cutting edges or cylindrical grinding disk at basic speed S=3750 rpm / ABS Q368 with inverse evaluation (tool broken, if dynamic signal = high level)* *Q368 = 0 → cutting edge control on a shaft milling cutter at rotation speed S=3750 rpm / CUT and normal evaluation* *Q368 = +30 → cutting edge control on a saw blade (with 30 relatively long cutting edges on a large diameter at rotation speed S=3750 rpm / ABS Q368 with normal evaluation (tool broken, if dynamic signal = low level)*

Q-Para	Definition / Meaning
Q375 Optional	Approach Strategy *Definition in which axis sequence for control cycles 585, 586 and 587 the starting position should be approached.* *<u>Attention:</u> The cycle time may be reduced with this parameter, at the same time the risk of collision with the laser or with the workpiece will be increased!* *Q375 = 0 → Retraction of the length axis to safety position* *Positioning of beam, radius and length axis consecutively* *Q375 = 1 → Retraction of length axis to safety position* *Positioning of beam, radius and length axis simultaneously* *Q375 = 2 → Direct positioning of beam, radius and length axis simultaneously* *!! Attention: increased risk of collision !!* *Value range: 0 <= Q375 <= 2* *Default value: Q375 = 0*
Q376 Optional	Safety Distance *The distance determines whether the length axis is directly positioned into the beam (Q376=0) or pre-positioned by this value before/above the beam (Q376>0).* *Is Q376>0 defined the length axis is - for safety reasons - at first brought to this position before beam and radius axis will be positioned.* *<u>Attention:</u> Depending on initial position + approach strategy there will be increased risk of collision!* *Value range: 0,000 mm <= Q376 <= 1000,000 mm* *Default value: Q376 = 50*
Q388... Q399	Auxiliary variables for cycle 588 *The Q-parameters are used in cycle 588 as variables for temporary results and overwritten.*

1.4.2 Description of the Parameters in TOOL.T

Para	Description
L	Tool length *For Q350=0/+3 the tool length can be assigned with L=0 mm (unknown tool length), for Q350=±1/±2/-3 the tool length L must be assigned with valid value (known tool length).* *The tool length L **must** always be defined in the tool table TOOL.T for tools with great diameters. Due to this data input the cycles can recognize the risk of collision and can calculate automatically a new measuring position and shift to the side to prevent the collision.* *Value range: -999.999 mm <= L <= +999.999 mm*
DL	Tool length compensation *The parameters L + DL must/should be assigned with valid values (known tool length).* *Value range: -99.999 mm <= DL <= +99.999 mm*
R	Tool radius *For Q350=0/+3 the tool radius can be assigned with R=0 mm (unknown tool radius), for Q350=±1/±2/-3 the tool radius R must be assigned with valid value (known tool radius).* *The tool radius R **must** always be defined in the tool table TOOL.T for tools with great diameters. Due to this data input the cycles can recognize the risk of collision and can calculate automatically a new measuring position and shift to the side to prevent the collision.* *Value range: -999.999 mm <= R <= +999.999 mm*
DR	Tool radius compensation *The parameters R + DR must/should be assigned with valid values (known tool radius).* *Value range: -99.999 mm <= DR <= +99.999 mm*
R2	Tool radius R2: angle radius / cutting edge radius / ball-ended radius *To make a useful measurement the angle radius should be ABS R2 >= 2 mm. In any case, for calculation of the control way (cycle 587) or the measuring positions (cycle 588) the actual cutting edge radius must be input with maximum accuracy.* *Value range: 0.000 mm <= R2 <= R (R2 positive, e.g. ball-ended mill)* *Value range: 0.000 mm >= R2 (R2 negative, e.g. special form mills)*
DR2	Tool radius compensation R2 *The parameters R2 + DR2 must be assigned with valid values (known cutting edge radius).* *Value range: -99.999 mm <= DR2 <= +99.999 mm*
CUT	Number of tool cutting edges for concentricity control / cutting edge control *The spindle test speed is determined via the number of cutting edges (CUT) and the impulse time (Q38).* *Value range: 0 <= CUT <= 20*
LTOL	Wear tolerance: length *For Q350=±1/±2 the tool wear is determined as difference of measured length value and given theoretical value and checked concerning the specified limits of wear. If the length tolerance is exceeded, the tool is locked in the tool memory TOOL.T.* *Value range: 0.000 mm <= LTOL <= +0.9999 mm* *Default value: LTOL = 0.030 mm [0.001 inch]*
RTOL	Wear tolerance: radius *For Q350=±1/±2 the tool wear is determined as difference of measured radius value and given theoretical value and checked concerning the specified limits of wear. If the radius tolerance is exceeded, the tool is locked in the tool memory TOOL.T.* *Value range: 0.000 mm <= RTOL <= +0.9999 mm* *Default value: RTOL = 0.030 mm [0.001 inch]*

Para	Description
LOFFS	Axial measuring position for radius measurement *For eccentric tools as end-milling cutters, cutter head, etc. the tool radius may not be measured on the height of the tool tip but a little bit higher to ensure safe shadowing of the beam. The position for the radius measurement is the relative axial distance from the tool tip up to the measuring position on the tool cutting edge.* *<u>Cycle 584:</u>* *Value range: 0.000 mm <= LOFFS <= +54.000 mm (MICRO)* *0.000 mm <= LOFFS <= +20.000 mm (NANO)* *Default value: LOFFS ≈ angle radius R2 + 0.5 mm for focussed or parallel laser beam* *<u>Cycle 585:</u>* *Value range: 0.000 mm <= Q353 + LOFFS <= +54.000 mm (MICRO)* *0.000 mm <= Q353 + LOFFS <= +20.000 mm (NANO)* *Default value: LOFFS ≈ angle radius R2 + 0.5 mm for focussed or parallel laser beam*
ROFFS	Radial measuring position for length measurement *For eccentric tools as end-milling cutters, cutter head, etc. the tool length may not be measured in the tool axis but eccentric on the tool cutting edges. The position for the length measuring is the relative radial distance from the tool axis to the measuring position on the tool cutting edge.* *If a new line will be added in the tool table, the sign 'R' is entered in column ROFFS. This sign will be replaced in the measuring cycle by the value 0.0.* *Value range: 0.000 mm <= ROFFS <= +999 mm (character 'R' not allowed!)* *Default value: ROFFS ≈ radius R − edge radius R2 − 0.5 mm for focussed or parallel laser beam*
LBREAK	Breakage tolerance: length *For tool breakage detection the tool length is roughly measured and compared with the default value from the tool memory. When exceeding the specified value for more than 2*LBREAK resp. when underflowing the tolerance LBREAK, Q199=2 is set and the tool is locked in TOOL.T.* *In cycle 586 the breakage tolerance value length in the BLUM_0**.TAB line 80, which is valid for all tools, is used at LBREAK=0.* *Value range: 0.0000 mm <= LBREAK <= 0.9999 mm* *Default value: LBREAK = 0.500 mm ... 0.9999 mm [0.02 inch ... 0.04 inch]*
RBREAK	Breakage tolerance: radius, permitted concentricity tolerance for cutting edge control *For the cutting edge control it is checked, if the diameters of all cutting edges referring to the measured max. cutting edge is within the permitted concentricity tolerance. When the tolerance is exceeded, Q199=2 is set and the tool is locked in TOOL.T.* *In cycle 584 the breakage tolerance value radius in the BLUM_0**.TAB line 81, which is valid for all tools, is used at RBREAK=0.* *Value range: 0.0000 mm <= RBREAK <= 0.9999 mm* *Default value: RBREAK = 0.020 mm ... 0.100 mm [0.001 inch ... 0.004 inch]*

1.5 Process Flow and Error Messages

1.5.1 Process flow messages

The Heidenhain control offers no possibility to display the proceeding program actions as process flow message on the screen.

1.5.2 Error messages

If an error occurs, which cannot be repaired immediately, the detected status is displayed as an error message on the screen. The cycle stops in an endless loop, the program has to be aborted and started again using the NC-STOP / INTERNAL STOP key.

Code	Error message	Description
0	"Laser system not ready"	The measuring system is not ready, even though it has been activated, i.e. the beam is not free or ready for measurement. → Check function of shutter → Check function of laser diode → Check function of PLC output → Check measuring distance
1	"Laser system not closed"	The measuring system is not closed and protected against pollution, even though it is deactivated. → Check function of shutter → Check function of laser diode → Check function of PLC output
2	"Faulty measuring condition"	The measuring block cannot be performed as the measuring signal is already highly active or the tool is in a wrong starting position to the beam. → Check measuring pos. and parameter values → Interference signal due to dropping coolant
3	"Scattering out of tolerance"	The scattering out/spread of the individually measured values exceeds the permitted tolerances. → Check tolerance limits → Check pollution of the tool → Interference signal due to dropping coolant
4	"Faulty approach side value"	The parameter "radial approach side" (line 20) or "axial approach side" (line 21) in file BLUM_0**.TAB is not correctly defined. → Set parameter "*** approach side" to value +1/-1
5	"Trigger point not found"	The search algorithm to determine the laser beam middle for cycle 582 was interrupted without trigger point. The change of the trigger point position since the last calibration/temp. comp. is larger than 1mm. → Machine not yet warmed-up → Check measuring pos. and parameter values → Effect calibration cycle
6	"Meas. block without trigger signal"	No trigger signal was recognized during the last measuring block. → Check measuring pos. and parameter values → Check function of the light barrier → Interference signal due to dropping coolant

Code	Error message	Description
7	"No tool loaded T-No.=0"	No valid tool is loaded into the spindle, the measuring cycle cannot be started without tool. → Load tool with T-No. ≠0
8	"Faulty tool data dimensions"	The length or radius of the measured tool exceeds the permitted tolerance of table BLUM_0**.TAB (line 60...63). → Compare value L resp. R with max. permitted tool data in table BLUM_0**.TAB
9	"Function not possible"	A special NT-function was selected at cycle call, which can not be performed with this laser system (Hardware for NT-functions not available). → Please select other function e.g. Q350=0/1/2 instead of Q350=-1/-2/-3/+3
10	"Tolerance exceeded"	The measurement result exceeds the permitted tolerance LTOL/RTOL, e.g.: • Deviation between old and new calibration values • Temperature drift in Length or Radius axis too big • Tool wear too big → Check measuring value → Check permitted tolerance LTOL/RTOL → Machine not yet warmed-up
11	"Attention, risk of collision"	The parameters “starting angle Q363”, “target angle Q364”, “angle radius R2”, “control travel Q353" or "measuring position LOFFS" together are larger than the permitted immersion depth Q52. The tool immerses in Z-direction too deep into the beam, there is risk of collision with the measuring system (MicroCompact version) → Check listed parameters → Check the parameters "overtravel in ***-axis" in BLUM_0**.TAB (line 11...13) → Reduce Q353=control travel on the tool shank
12	"Faulty tool axis definition"	The tool axis as indicated in the last TOOL CALL block does not correspond with the length axis (Q2) in table BLUM_0**.TAB (line 7) or is not defined. → Check TOOL CALL block → Check parameter "length axis" in BLUM_0**.TAB
13	"Faulty number of cutting edges"	The number of cutting edges for the tool which must be checked is indicated in the tool table with CUT=0. The test speed for the cutting edge check cannot be determined. → Input number of cutting edges CUT correctly in the tool table
14	"Faulty tool data in TOOL.T"	A measuring parameter (LTOL, RTOL, LOFFS, ROFFS, CUT, LBREAK, RBREAK, etc.) in tool table TOOL.T has an unvalid value. Example: For cycle 582 no permitted drift LTOL/ RTOL for the reference tool is input. → Assign LTOL, RTOL in TOOL.T with valid value. Example: For cycle 584 ROFFS was not set to a numerical value. → Check parameter ROFFS<>R in TOOL.T

Code	Error message	Description
15	"Faulty call parameter Qxxx"	One or several call parameters (Q350...Q368) are assigned with a critical or invalid value. → Check all call parameters concerning valid values.
16	"Error parameter table"	The parameter table PLC:\BLUM\BLUM*.TAB could not be found or the parameter values are faulty. → Check state of markers/flags M1204, M1206, M1210 → Check availability of parameter table → Check parameter values in parameter table
17	"Error measuring evaluation"	On evaluation of the measuring data an irreparable error was found, e.g. division by zero. → Repeat cycle with changed parameters
18	"Faulty spindle speed"	The calculated speed for the single cutting edge monitoring was higher than the allowed max. spindle speed defined in the last TOOL CALL instruction or the spindle speed was not achieved in a preset period of time. The cutting edge control cannot be effected correctly. → Specify a higher speed with TOOL CALL T# Z S# . → Select measuring speed at cycle 584 so that no gear is shifted for the test speed.
19	"Faulty trigger position"	The difference between the actual axis position and the trigger position as indicated by the control after a measuring block is more than 20mm. The positioning of the axis with reference to the trigger position can cause collision. → To delete the control error the control must completely be switched off and on again. → TNC software version must be noted and the manufacturer of the machine must be informed.

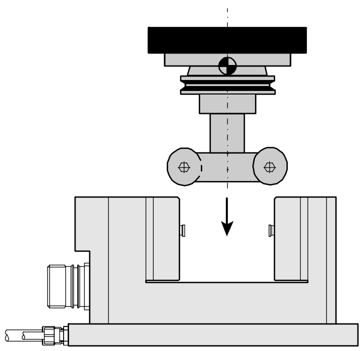
1.6 General Information

1.6.1 General issues

- The Blum software version installed on the control can be found out from the file PLC:\TCHPROB1.SYS or TNC:\BLUM\INFO.A. Example:

  Project: C:\CYCLE\TNC\ V5f\P870634-030-410-iTNC530-340490-V5fa.cdf → Blum software version V5FA for iTNC 530
- The program examples in directory TNC:\BLUM\BLUM58*.H show how the measuring cycles can be called out of a machining program.
- The measuring cycles can be installed completely or partially on the CNC control. The measuring cycle calibration is necessary in any case.
- The tool measurement is effected with the measuring block TCH PROBE 3.0 MEASURING in the machine coordinate system (REFSYSTEM:1), regardless whether cycle 19 or M128 are active (condition: iTNC530 since version 340 422 05 ff). For older control software versions an error message is displayed, if 3D-ROT is active.
- All travel movements (except of measuring blocks) can be influenced with the override potentiometer.
- When interrupting the cycle due to error messages the already faultless measured values are not input into the tool table TOOL.T.
- On request the manufacturer can program that the tool which should be measured/controlled will be locked in the tool table on cycle start and so will also be locked on cycle stop (i.e. the tool was not measured/controlled).
- Basically, calibration and measurement must be effected when the machine is warmed-up. A warming-up period of 15 min. for medium spindle rotation speed, switched-on coolant flow and moved CNC axes is recommended. This ensures always the same environmental conditions for the measuring system.
- To achieve high process safety and reliability in case of errors basically repetition measurements are effected. So, in connection with a useful evaluation strategy a high measuring accuracy can be achieved.
- At the beginning of the cycle the tool axis is moving into the safety position Z with rotating tool. When marker/flag M1207=1 is set, the cycle is stopped (e.g. to wait for a limit switch when swivelling out the measuring system or removing a protection device, etc.). Only with M1207=0 the cycle execution is enabled and the cycle is processed.
- At the beginning of the cycle the last programmed spindle speed S and the spindle state (M3, M4, M5) is read from the cycle and temporarily stored. At the end of the cycle the previous state of the spindle is re-made independantly from the determined tool state (tool okay/tool faulty), i.e. the operator must ensure that the spindle is activated for operation with M3/M4.

Risk of collision !



Notice:

The tool radius R must always be defined in the tool table TOOL.T for tools with great diameters. Due to this data input the cycles can recognize the risk of collision and can calculate automatically a new measuring position and shift to the side to prevent the collision.

1.6.2 Tool loading

The loading of the tool into the spindle must always be done before starting the measuring cycle. At TOOL CALL the tool no. T#, the tool axis and the spindle speed Sxxx must always be indicated.

In order to generate indexed tool numbers for tools with several correction data, e.g. step drill, T-groove milling slotter, etc. MP 7262 > 0 must be set. The indexed tool number e.g. T 5.1 is inserted into the tool table by means of the softkey "insert line" following tool number 5. The results of the measuring cycles are automatically assigned to the actual tool number.

Example for the tool loading in the CNC-mode:

```
...
TOOL DEF 5                              TOOL DEF 5.1
TOOL CALL 5 Z S3000 FMAX                TOOL CALL 5.1 Z S3000 FMAX
...
```

In order to consider the spindle speed depending length change during measurement it is recommended to measure the real tool length and radius at operating speed. Therefore measurements are always made with the spindle speed S as programmed in the last TOOL CALL block.

At S=0 the spindle state is set to M5.

At S<=360 the value is interpreted either as spindle rotation speed or as orientation angle and the spindle is positioned with cycle 13 and M19 to the indicated angle position (= function depending on parameter "spindle orientation at S<=360" in BLUM_0**.TAB line 83).

At S>360 the value is interpreted as spindle speed and the spindle is brought to rotation by M3.

At the end of the measuring cycle the spindle speed is set as indicated in the last TOOL CALL block.

1.6.3 Reading and writing of tool data

When a measuring cycle is called, the actual tool data (T-no., length L, radius R, comp. values DL, DR, measuring parameters LTOL, RTOL, LBREAK, RBREAK, LOFFS, ROFFS, CUT, etc.) are read in the tool table TOOL.T. The tool status TL in the tool management TM must be enabled, in order to start the measuring cycle.

Tool length parameter in tool management:

The variable value L from the tool management is interpreted by the measuring cycle as being the nominal length L_{TAB}, the variable value DL as being the actual, measurable length wear value DL_{TAB}. By means of the variable TOOL CALL a fictive length offset $DL_{TOOLCALL}$ can be performed (e.g. in case of machining with a tolerance or if a length offset is necessary, depending on the measured cutting edge geometry). The TNC control calculates the total tool length for the part program as:

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

The measuring cycle calculates the actual tool length for pre-positioning above the Laser beam as:

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

If L = 0, $L < L_{min}$ or $L > L_{max}$ tool length is supposed to be unknown. In this case, a rough search is being executed in the first measuring block. The starting position of the measuring block is located approx. 6 mm above the Laser beam with respect to the permitted maximum tool length L_{max}, the target position is located approx. 10 mm below the Laser beam with respect to the permitted minimum tool length L_{min}. This makes sure that both the longest and the shortest tool trigger a signal for rough length determination within the gauging stroke. All following approach cycles for fine measuring are then executed with respect to this roughly determined length.

If $L \geq L_{min}$ and $L \leq L_{max}$ tool length is supposed to be known. In this case, a rough search with measuring way limitation is being executed in the first measuring block. The starting position of the measuring block is located approx. 6 mm above the Laser beam with respect to the permitted maximum tool length L_{max}, the target position is located approx. 10 mm below the Laser beam with respect to the indicated tool length L_{TAB}. A signal is expected within this limited gauging stroke when the tool immerses in the Laser beam.

An evenly spreaded distribution of the total tool length among the 3 indicated tool length parameters is possible and without any influence on the part program, but it makes no sense when using Blum measuring cycles, because they process or determine only L_{TAB} and DL_{TAB}. The method of processing and determination of these tool length values and the storage of the acquired tool data in the tool management is defined in the measuring cycle and can not be split up in percentages.

The total length of the reference tool L+DL can be entered into the tool table with 0 mm (= zero tool), if L_{min} < 0 mm was entered into the table BLUM_0**.TAB (correct: L=−10.000 mm DL=+10.000 mm!). The tool lengths are then determined relating to this zero length so that positive and negative lengths are possible.

Warning! Risk of collision:

When tool length L is entered manually into the tool management, make sure not to exceed a maximum deviation of more than ± 5 mm of the real length for both, tool length determination and when entering a value into the tool management. There is a high risk of collision, if an incorrect value is entered, because an incorrectly entered tool length will lead to incorrect calculation of the starting and target position to which the tool is brought! If the tool enters the Laser beam already during pre-positioning, no signal will be triggered during the following measuring block preventing the measuring axis from collision!

This danger also exists, if the correctly measured tool length value L or DL, entered by the measuring cycle, is manually modified afterwards!

Tool radius parameter in tool management:

The variable value R from the tool management is interpreted by the measuring cycle as being a nominal radius value R_{TAB}, the variable value DR as a real, measurable radius wear value DR_{TAB}. By means of the variable TOOL CALL a fictive radius offset $DR_{TOOLCALL}$ can be performed (e.g. when machining with tolerance or in case of a necessary radius offset, depending on the measured cutting edge geometry). The TNC control calculates the total tool radius for the part program as:

$R = R_{TAB} + DR_{TAB} + DR_{TOOL\ CALL}$

The gauging cycle calculates the real tool radius for pre-positioning next to the Laser beam as:

$R = R_{TAB} + DR_{TAB}$ (without $DR_{TOOL\ CALL}$!)

If R = 0 or R > R_{max} the tool radius is supposed to be unknown. In this case a rough search is performed in the first measuring block. The starting position of the measuring block is located about 6 mm before the Laser beam with respect to the permitted maximum tool radius R_{max}, the target position is located about 2 mm behind the Laser beam with respect to the tool centreline. This makes sure that both the biggest and the smallest tool trigger a signal for rough radius determination within the measuring stroke. All the following approaching cycles for fine measuring are then executed with respect to this roughly determined tool radius.

If R > 0 and R ≤ R_{max} the tool radius is supposed to be known. In this case, a rough search with measuring way limitation is being executed in the first measuring block. The starting position of the measuring block is located about 6 mm before the Laser beam with respect to the specified tool radius R_{TAB}, the target position at about 2 mm behind the Laser beam with respect to the tool centreline. A signal is expected within this limited measuring stroke when the tool enters the Laser beam.

An evenly spreaded distribution of the total radius over the 3 mentioned tool radius parameters is possible and without any influence for the part program, but it makes no sense when using the Blum measuring cycles, because they process and determine only R_{TAB} and DR_{TAB}. The method of processing and determination of these radius values as well as the storage of the determined tool data in the tool management is definitely installed in the measuring cycles and cannot be split up into percentages.

Warning! Risk of collision:

When the tool radius R is entered manually into the tool management, make sure not to exceed a maximum deviation of more than ± 5 mm of the real radius for both, tool radius determination and when entering a value into the tool management. There is a high risk of collision, if an incorrect value is entered because an incorrectly entered tool radius involves incorrect calculation of the starting and target position to which the tool is brought! If the tool enters the Laser beam already during pre-positioning, no signal will be triggered during the following measuring block preventing the measuring axis from collision!

This danger also exists, if the correctly measured tool radius value R or DR, entered by the measuring cycle, is manually modified afterwards!

At the end of the cycle the measuring result is automatically written into the tool management:

- On faultless measurement the tool data (length L, radius R and compensation values DL, DR) are updated and the tool status TL is set to valid/enabled.
- On faulty measurement the tool data (length L, radius R and compensation values DL, DR) are not updated and the error message leads to a cycle abortion.

1.6.4 Switching mm/inch

Switching the control from "mm" to "inch" is possible without any problem. If a program is defined in inch (e.g. with BEGIN PRG xxx INCH and END PRG xxx INCH), all way and position depending constants are automatically interpreted as inch values. The call parameters can be put in as inch values, the result parameters such as tool length and radius are automatically changed and put into the tool table as inch values. The default values for the Q-parameters Q350...Q368 are round up or off when put in as inch value.

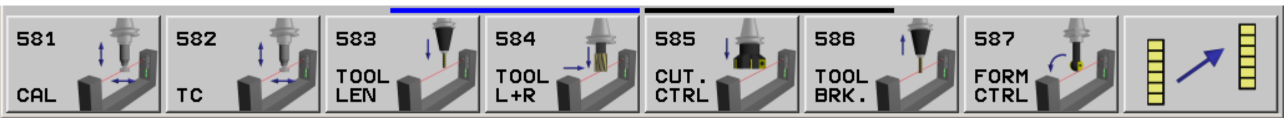
Notice: The parameters in BLUM_0**.TAB must always be inserted as "mm" values, even if the control works in "inch".

1.6.5 Call of the Measuring Cycles in DIN / ISO Code

The measuring cycles can be called from a machining program in DIN / ISO code without any problem. The cycles are selected with the following buttons and then inserted into a machining program:

- Key "TOUCH PROBE" (on the operating panel)
- Softkey symbol "BLUM LaserControl" (1st. softkey level, 2nd. softkey line)

- Softkey symbols with cycle numbers "581...587" (2nd. softkey level, 1st. softkey line)



- Softkey symbols with cycle numbers "588...xxx" (2nd. softkey level, 2nd. softkey line)

Example to call the measuring cycle G583 in DIN / ISO code:

```
...
T6 M6 S3000
G583 Q350=0 Q361=3 Q362=0.005 Q359=0.03
...
```

1.6.6 Operating Mode "Program Test" and "Block Search"

In the operating mode "Program Test" or on active "Block Search" the BLUM measuring cycles are skipped. If valid tool data for work piece machining must be available in the tool table, their input must be done manually or determined in advance by a separate measuring cycle.

1.7 *General Recommendations*

In order to obtain a satisfactory measuring result, the following recommendations must be observed:

1.7.1 Problems due to coolant

- Before measuring, the inner and outer coolant supply must be switched-off. If possible, do not measure immediately after the coolant was stopped. Insert a dwell time, if necessary.
- Tool cutting edges, which are covered with coolant or chips, should be cleaned by means of compressed air or centrifugal force at high spindle rpm. In the latter case, do not exceed the max. rpm indicated by the tool manufacturer.
- Tools, on which the switched-off inner coolant is still dripping, can partly be cleaned at high rpm. Even in this case, tool breakage detection is possible with limited accuracy (error < 0.1 mm).
- If coolant is splashing during the measuring, the air seal (barrier air) must stay switched-on. In order to avoid that coolant drops from inside the tool are ejected at lens protector level, change spindle rpm.

1.7.2 Problems due to coolant mist

- Coolant mist reduces the intensity of the light on the receiver with increasing density and increasing distance between transmitter and receiver. In this case, the amplification on the signal receiver must be increased.
- In case of very dense coolant mist, it may occur that the Laser light barrier does not get ready. In this case, the coolant mist must be sucked off or a dwell time has to be inserted until the mist has disappeared.
- Even at reduced light intensity, an accurate measuring can be performed, if the calibration is made right before tool measuring.
- Constant mist in the working area can be compensated, if the calibration and the tool measuring are performed on both sides and not just on one, and if the average of the results is calculated (e.g. for tool diameter). In this case, the Laser beam centre has to be used as a reference value (requires software modification). For time saving reasons, measuring on one side is generally preferred.

1.7.3 Problems due to a dirty optical lens

- If the lens gets dirty frequently, check if there is oil or water in the filter unit and change filter, if necessary. In the same way, the pneumatic pipes of the protecting tubes and the air seal have to be replaced by new ones, since the air permanently carries the deposits into the measuring system.
- The optical glass, which protects the transmitter and the receiver, must be kept extremely clean. Use a wet cloth for glasses to clean them, if they are dirty. Even finger prints may lead to incorrect measurements.
- If the pneumatic system with the filter unit is correctly installed, the cleanness of the optical system is usually guaranteed for a long time.

1.7.4 Influences on absolute accuracy

- In case of dense coolant mist between transmitter and receiver, the signal position is offset towards the Laser beam centre, i.e. the measuring signal is actuated earlier, and the geometry of the tool seems to be measured bigger (error approx. < 0.02 mm).
- If the tool cutting edges are very dirty, due to coolant (coolant film, not drops!) the geometry of the tool is measured bigger (error approx. < 0.03 mm).
- Compared to a tool presetting unit, which works with a CCD camera according to the principle of optical on- or through light, differences in the tool dimensions may occur, because with the laser system the tool dimensions are measured dynamically in the machine under real conditions. In case of length measuring the tool length is measured inclusive the clamping error (errors up to 0.070 mm were measured). In case of radius measuring the tool radius is measured inclusive the run-out error of the spindle, the retooling error and the flight circle.
- The surface quality (dull, brilliant, metallic) nearly does not have any influence on the absolute accuracy (error < 0.005 mm), just so the surface colour due to different coatings (HSS, VHM, PKD, TiN, TiCN).
- The recommended spindle rpm for tool length and radius measuring is the operating spindle speed.
- As far as the measuring speed is concerned, the system error due to the federate/rpm ratio requires special care. In order to obtain a resolution of 1µm, the following measuring speed must be respected:

  Feedrate [mm/min] <= 0.001 [mm] * spindle rpm [1 /min].

  The measuring speed must be kept constant during the whole measuring block and must not be influenced or reduced by override potentiometer.

2 Description of the Measuring Cycles

2.1 Cycle 581 – Calibration of the Laser System

Cycle 581 is the program for calibration of the BLUM laser system by means of a reference tool. Before measuring with the Laser light barrier, the system has to be calibrated, i.e. the exact position of the points of commutation referring to the machine coordinate system must be determined. Therefore, use a Blum reference tool, with a specific shape for calibration (cylindrical pin with a ∅ 8/12mm shaft and ∅ 12/20mm check). This reference tool is inserted into the taper in order to measure its length, diameter and height outside the machine with an accuracy of microns. A minimum radial run-out has to be ensured for calibration measuring.

The exact dimensions of the reference tool (L=length, R=radius, DL=height, DR=0.5*run-out) are loaded from the tool table. The calibration cycle "measures“ the reference tool with respect to the preset reference dimensions. The results are stored in different calibration parameters (Q-parameters) in the file BLUM_0**.TAB. These calibration parameters are not allowed to be overwritten or changed. These parameters are used during tool measuring for length and diameter calculation.

Following Q-parameters are used to call the program:

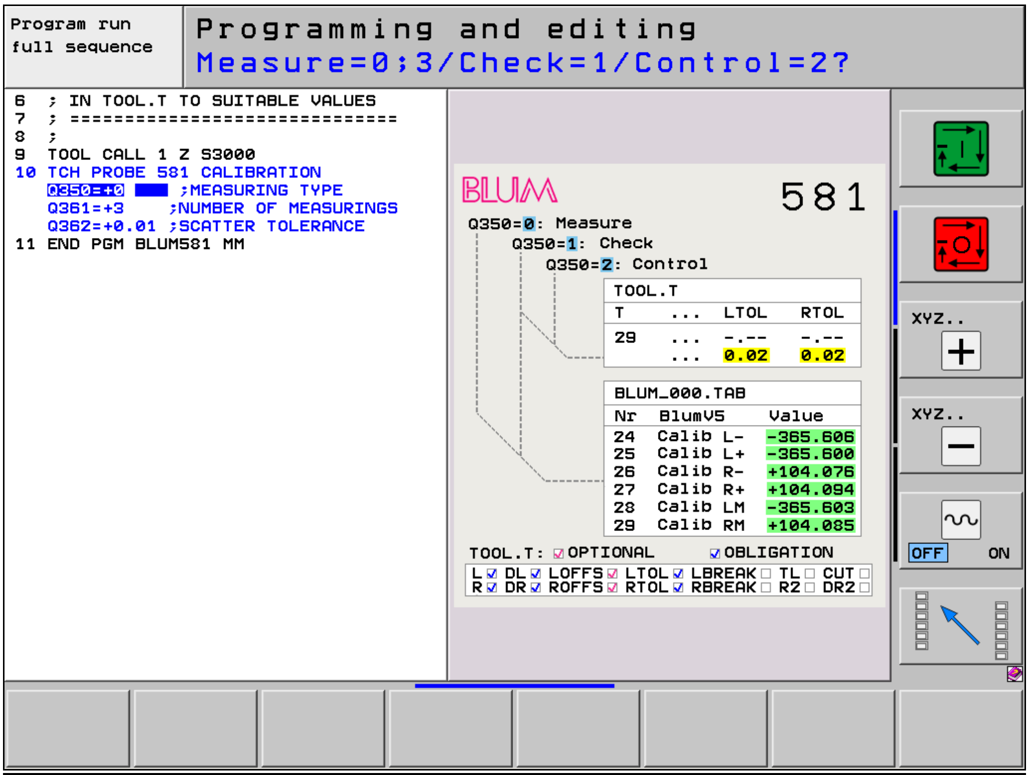
Q350	Measuring mode: (without function, only necessary for use in DIN/ISO execution) *The new, actual calibration values are determined, stored in BLUM_0**.TAB and compared with the old, still active calibration values, if they are within the permitted tolerances LTOL / RTOL. If the tolerances are exceeded by temperature drift or pollution on the tool, the error message "Tolerance exceeded" is displayed (= tolerance supervised calibration)* *Q350 = 0 → Measure: Stores the new calibration values, no supervision of tolerances between new and old calibration values (initial calibration)* *Q350 = 1 → Check: Stores the new calibration values, supervision of tolerances, calibration values are also stored as measuring positions in BLUM_0**.TAB* *Q350 = 2 → Control: no storage of calibration values, only supervision of tolerances*
Q361	Number of repetitive measurements for averaging
Q362	Permitted dispersion tolerance of the individual measuring values at repetitive measurements

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

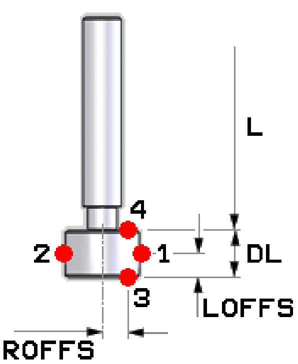
L=xxx,xxx	Length of reference tool *Measured distance between tool carrier reference point and **upper edge** of test diameter via dial gauge or tool pre-setting device (input* accuracy *in µm)*
R=xx,xxx	Radius of reference tool *Measured radius of test diameter via internal micrometer (input* accuracy *in µm)*
DL=xx,xxx	Tool compensation length = height reference tool *Measured distance between upper and lower edge of the test diameter via internal micrometer (input* accuracy *in µm)*
DR=x,xxx	Tool compensation radius = run-out *Measured run-out of the tool inserted in the spindle via dial gauge.* *(input: +0.5 * run-out = +0.5 * (max. − min. deflection))*

LTOL	Maximum deviation of the calibration values in the length axis (for comparative calibration) *In order to detect errors during calibration (due to pollution on the tool), a tolerance has been defined. If the detected deviation (difference between old and new calibration value) exceeds the permitted tolerance an error message is displayed.* *Value range: 0.0000 mm <= LTOL <= 0.9999 mm* *Default value: LTOL = 0.020 mm ... 0.050 mm [0.002 inch]*
RTOL	Maximum deviation of the calibration values in the radius axis (for comparative calibration) *In order to detect errors during calibration (due to pollution on the tool), a tolerance has been defined. If the detected deviation (difference between old and new calibration value) exceeds the permitted tolerance an error message is displayed.* *Value range: 0.0000 mm <= RTOL <= 0.9999 mm* *Default value: RTOL = 0.020 mm ... 0.050 mm [0.002 inch]*

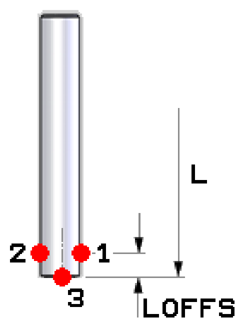
Example for the call of the measuring cycle in the CNC program:



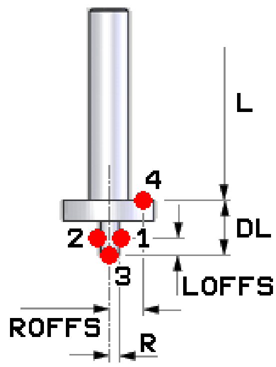
T	1
L	+80.5
DL	+10.0
R	+6.0
DR	+0.005
R2	xxx
DR2	xxx
TL	
CUT	xxx
ROFFS	+5.5
LOFFS	+5.0
LTOL	+0.02
RTOL	+0.02
LBREAK	xxx
RBREAK	xxx



ROFFS = R – 0.5
LOFFS = DL / 2



ROFFS = R – 0.5
LOFFS >= 1.5



ROFFS = 6.3
LOFFS = 2.0

Explanation

The reference tool as tool number T=1 must always be loaded into the spindle before starting the calibration cycle 581.

A multiple approach cycle with rough and fine resolution is performed for each approach direction with rotating spindle (S=3000 rpm). After the rough search of the testing edges three repetitive measurements (Q361) per each approaching side are effected and the dispersion of the individual measuring values is checked for the dispersion tolerance of 0.01 mm (Q362). The measuring results are saved as positive and negative calibration values in the table BLUM_0**.TAB. These values are not allowed to be extinguished or overwritten in this table!

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

Caution:

The reference tool should be shrunk into a tool holder and must be entered with a tool number in the tool management. Before measuring, the reference tool should be cleaned at all measuring places with a wet cloth for glasses.

A faulty input of calibration parameters may cause the following errors:

- Incorrect indications concerning the dimensions of the reference tool lead to incorrect measuring results.
- If calibration is missing or has not been performed correctly, the machine will move to wrong or undefined measuring positions during the next tool check.

Calibration and measuring must be performed at machine operating temperature (at least 15 min. after start up with rotating spindle).

Following parameters are determined during calibration:

Q40	Length calibration value L– *position: referring to laser beam middle in negative axial direction*
Q41	Length calibration value L+ *position: referring to laser beam middle in positive axial direction*
Q42	Radius calibration value R– *position: referring to laser beam middle in negative radial direction*
Q43	Radius calibration value R+ *position: referring to laser beam middle in positive radial direction*
Q44	Laser beam middle in the length axis LM *The laser beam middle is determined by the arithmetic average of the positive and negative length calibration value: Q44=(Q40+Q41)/2*
Q45	Laser beam middle in the radius axis RM *The laser beam middle is determined by the arithmetic average of the positive and negative radius calibration value: Q45=(Q42+Q43)/2*

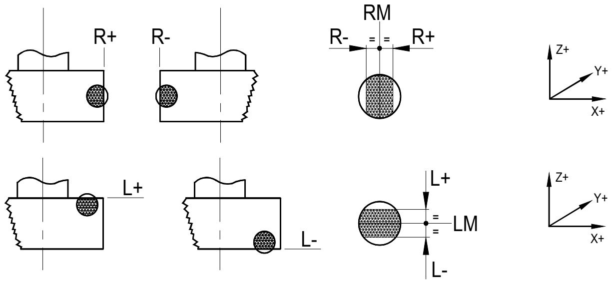


Fig.: Trigger point positions in the laser beam

2.2 Cycle 582 – Temperature Compensation of the CNC axes

Cycle 582 is the program for the compensation of the CNC axes with the BLUM laser measuring system by means of the reference tool. In order to compensate the temperature related dilation of a machine for increased accuracy requirements, the longitudinal variation of an axis due to temperature easily can be measured using the BLUM Laser system and the reference tool.

The exact dimensions of the reference tool (L=length, R=radius, DL=height, DR=0.5*run-out) are read from the tool table when the program is called. The program "measures" the reference tool and determines the radial and axial position of the middle of the laser beam. This position is then compared to the reference position of the TC reference measurement. The detected difference is used as an offset value which can be transferred into the additive origin offset. Thereafter, the part program is executed with the corrected values.

The temperature compensation of the CNC axes using the laser system makes sense only, if the laser itself is not influenced by large temperature drift, e.g. large temperature influences from hot coolant or chips. Laser systems “MINI version“ with long supports or “MICRO Compact version“ systems with long extensions intensify those negative influences. The best result is reached with a standard “MICRO Compact version“ laser system mounted directly on the machine table close to the work piece.

The determined displacement values can be read and evaluated via the NC machining program or via the PLC program (pls. clarify the evaluation strategy with machine manufacturer). When the measuring cycle is successfully executed, the marker/flag M1208=1 is set by the measuring cycle as identifier for the update of the displacement values in parameter table PLC:\BLUM\BLUM_0**.TAB.

The integration of temp. comp. cycle 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. Machining of the first work piece until all dimensions are in compliance with the theoretical values.
4. Reference measurement (Q350=0), in order to keep the actual status of the machine axes as a ref. value.
5. Machining of the next work piece.
6. Comparative measurement (Q350=1), in order to determine the actual axis offset with respect to the reference value.
7. Depending on thermal machine drift and the required accuracy, the TC comparative measurement can be performed before or after any desired number of machining tasks.

Following Q-parameters are used for the program call:

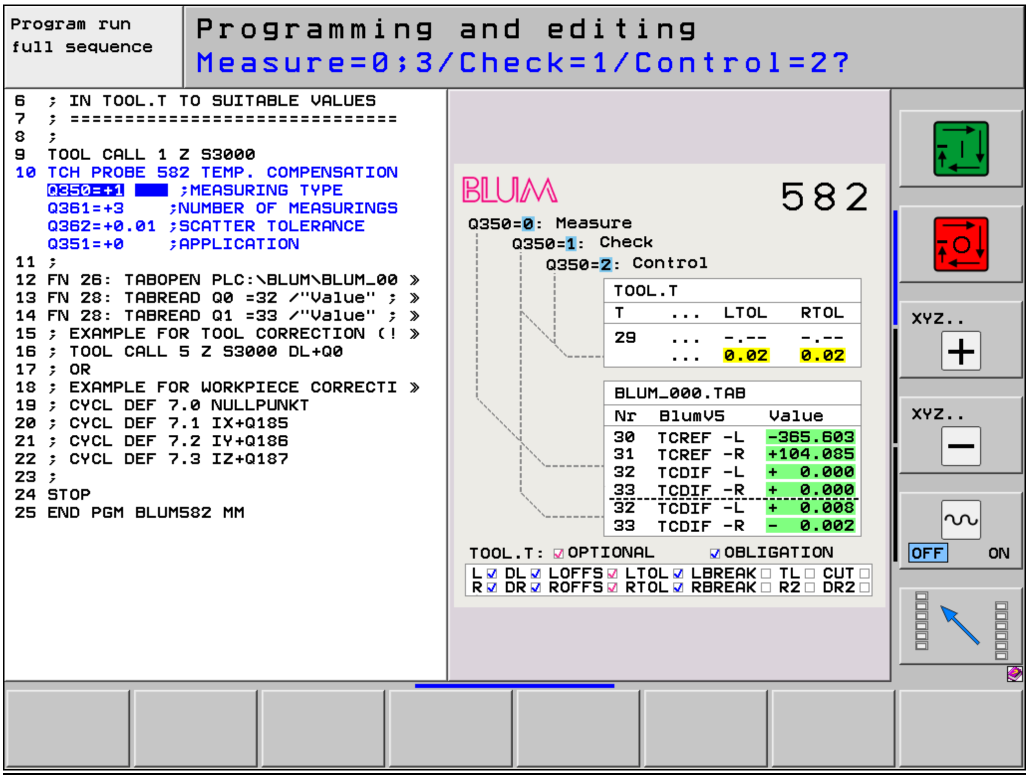
Q350	Measuring mode: compensate result from TC as reference value or comparison value *Q350 = 0 → measurement = reference measurement (default) with updating calibration values* *Q350 = 1 → check = comparison measurement with updating calibration values* *Q350 = 2 → check = comparison measurement without updating calibration values*
Q361	Number of repetitive measurements for averaging
Q362	Permitted dispersion tolerance of the individual measuring values at repetitive measurements
Q351 optional	Measuring scope

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

L=xxx,xxx	Length of reference tool *Measured distance between tool carrier reference point and upper edge of test diameter via dial gauge or tool pre-setting device (input* accuracy *in µm)*
R=xx,xxx	Radius of reference tool *Measured radius of test diameter via internal micrometer (input* accuracy *in µm)*
DL=xx,xxx	Tool compensation length = Height reference tool *Measured distance between upper and lower edge of the test diameter via internal micrometer (input* accuracy *in µm)*
DR=x,xxx	Tool compensation radius = Eccentricity / Run-out *Measured run-out of the tool inserted in the spindle via dial gauge.* *(input: +0.5 * run-out = +0.5 * (max. – min. deflection))*

LTOL	Maximum offset value for the length axis (for comparative measurement) *The extension of the machine is limited. In order to detect errors during measuring, a tolerance has been defined. If the detected deviation (difference between reference and actual value) exceeds the permitted tolerance an error message is displayed. This deviation value is entered into table PLC:\BLUM\ BLUM_0**.TAB, line 32. This value can be read, and depending on the evaluating strategy, it can be used as additive zero offset.* *Value range: 0.0000 mm <= LTOL <= 0.9999 mm* *Default value: LTOL = 0.020 mm ... 0.050 mm [0.002 inch]*
RTOL	Maximum offset value for the radius axis (for comparative measurement) *The extension of the machine is limited. In order to detect errors during measuring, a tolerance has been defined. If the detected deviation (difference between reference and actual value) exceeds the permitted tolerance an error message is displayed. This deviation value is entered into table PLC:\BLUM\ BLUM_0**.TAB, line 33. This value can be read, and depending on the evaluating strategy, it can be used as additive zero offset.* *Value range: 0.0000 mm <= RTOL <= 0.9999 mm* *Default value: RTOL = 0.020 mm ... 0.050 mm [0.002 inch]*

Example for the call of the measuring cycle in the CNC program:



T	1
L	+80.5
DL	+10.0
R	+6.0
DR	+0.005
R2	xxx
DR2	xxx
TL	
CUT	xxx
ROFFS	+5.5
LOFFS	+5.0
LTOL	+0.02
RTOL	+0.02
LBREAK	xxx
RBREAK	xxx

Explanation

The reference tool as tool number T=1 must always be loaded into the spindle before starting the measuring cycle 582.

For each approaching side three repetitive measurements (Q361) with rotating spindle (S=3000 rpm) with fine resolution are effected, the average is calculated and the dispersion of the individual measuring values is checked for the dispersion tolerance of 0.01 mm (Q362). The laser beam centre is calculated from the average of the positive and negative approaching side. The resulting compensation values for length and radius axis are compared with the stored TC reference values (Q350=1). The difference may not exceed the permitted tolerance in the length axis (LTOL) and the radius axis (RTOL).

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

Caution:

For reference measurement (Q350=0) the resulting TC values are entered as reference values into the table BLUM_0**.TAB. These parameters are not allowed to be erased or overwritten! The calibration values are also updated during TC reference measurement.

For comparative measurement (Q350=1) the difference between the actual TC offset values are defined and entered into the table BLUM_0**.TAB. The calibration values are updated only, if the resulting offset is within the permitted tolerance (LTOL.RTOL).

Following Q-parameters are defined resp. actuated during TC measurement:

Q40	Length calibration value L–
Q41	Length calibration value L+
Q42	Radius calibration value R–
Q43	Radius calibration value R+
Q44	Laser beam middle in the length axis LM
Q45	Laser beam middle in the radius axis RM
Q46	Reference laser beam middle in the length axis (for Q350=0 only) *The laser beam middle is determined by the arithmetic average of the positive and negative length calibration value.* *Usage: As reference value for the temperature depending offset of the length axis.*
Q47	Reference laser beam middle in the radius axis (for Q350=0 only) *The laser beam middle is determined by the arithmetic average of the positive and negative calibration value.* *Usage: As reference value for the temperature depending offset of the radius axis.*
	Offset in length axis (for Q350=1 only) *The difference between TC reference value and TC comparison value is entered into table PLC:\BLUM\BLUM_0**.TAB, line 32. This value can be read there and used as additive zero offset. This compensation must be added and adapted by the user to his individual evaluation strategy.* *Offset length axis (ΔL) = reference value (Q46) – actual value (L) (unit: mm)*
	Offset in radius axis (for Q350=1 only) *The difference between TC reference value and TC comparison value is entered into table PLC:\BLUM\BLUM_0**.TAB, line 33. This value can be read there and used as additive zero offset. This compensation must be added and adapted by the user to his individual evaluation strategy.* *Offset radius axis (ΔR) = reference value (Q47) – actual value (R) (unit: mm)*
Q185	Offset in X-axis
Q186	Offset in Y-axis
Q187	Offset in Z-axis

Example for the reading of the offset values in the following CNC program:

(as far as the function is not already realized by the machine manufacturer in the PLC!)

```
...
TCH PROBE 582 TEMP.COMPENSATION
 Q350=1       ; MEASURING MODE
 Q361=3       ; NUMBER OF MEASUREMENTS
 Q362=0.01    ; SCATTERING TOLERANCES
 Q351=0       ; MEASURING SCOPE
; POSITIVE SIGN FOR CORRECTION WORKPIECE ORIGIN
CYCL DEF 7.0 ZERO POINT
CYCL DEF 7.1 IX+Q185 ; ADDITIVE ZERO OFFSET dR IN THE RADIUS AXIS
CYCL DEF 7.2 IY+Q186 ; ADDITIVE ZERO OFFSET=0 IN THE BEAM AXIS
CYCL DEF 7.3 IZ+Q187 ; ADDITIVE ZERO OFFSET dL IN THE LENGTH AXIS
...
```

2.3 Cycle 583 – Tool Length Measurement of Centric Tools

Cycle 583 is the program for length measurement of centric tools with the BLUM laser system. The system can measure known tools (tool length L≠0 mm) or unknown tools (tool length L=0). The measuring result is written into the tool table under the actual tool number T-No.

Caution: For unknown tool data (L=0, R=0) there is always the risk of collision with the BLUM laser light barrier. Therefore we recommend basically the input of rough tool geometry data into table TOOL.T (input accuracy / maximum deviation < ± 5 mm).

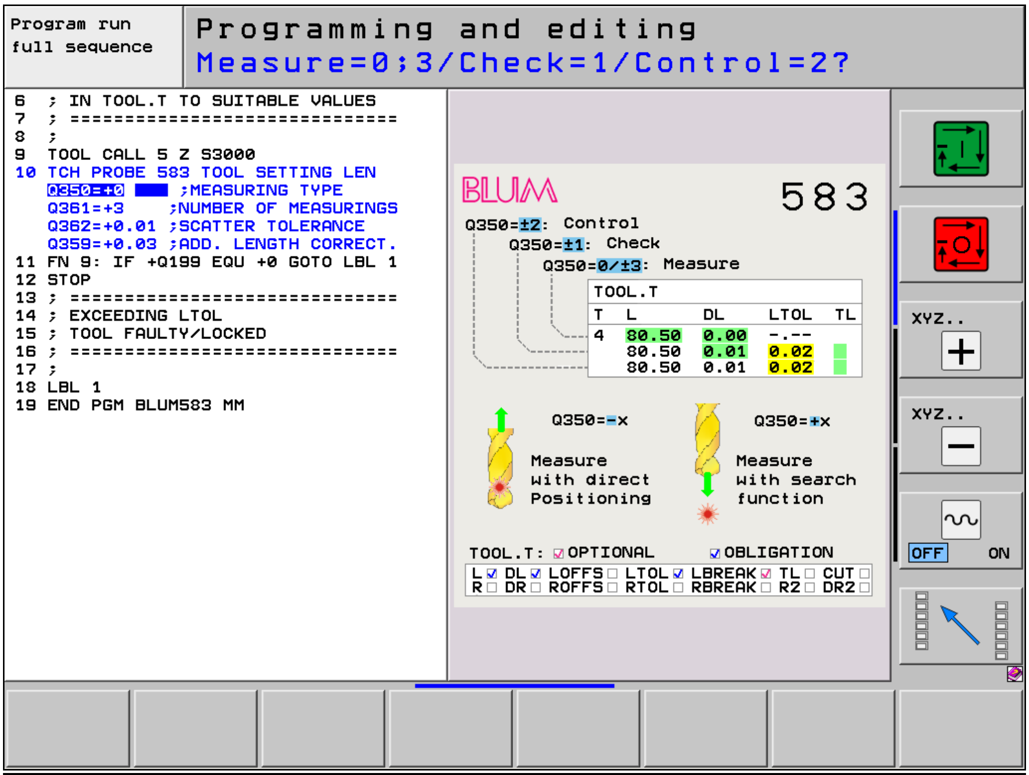
Following Q-parameters are used for program call:

Q350	Measuring mode	(note description of Q350 on page 9)
Q359	Additive length compensation	
Q361	Number of repetitive measurements for averaging	
Q362	Permitted scattering out of tolerance of individual measuring values for repetition measurement	

Following parameters must be defined in tool table TOOL.T:

L, DL	Tool length, tool length correction	necessary at Q350=±1/±2/-3)
LTOL	Wear tolerance: length	(only at Q350=±1/±2)

Example for the call of the measuring cycle in the CNC program:



T	1
L	+0.0
DL	+0.0
R	xxx
DR	xxx
R2	xxx
DR2	xxx
TL	
CUT	xxx
ROFFS	+0.0
LOFFS	xxx
LTOL	+0.02
RTOL	xxx
LBREAK	xxx
RBREAK	xxx

Explanation:

The tool with T-No.=5 is loaded into the spindle before call of cycle 583.

The tool length is measured three times (Q361) in the tool axis (ROFFS=0) via multiple approaching cycle and the scattering out of the individual measuring values concerning specified limits of 0.01 mm (Q362) is controlled. The additive length compensation value of 0.03 mm (Q359) is added to the tool compensation DL.

At **Q350=0/±3 (measure)** the measured tool length is written in memory L and the tool compensation set to DL=0.

At **Q350=±1 (check)** the theoretical tool length in memory L is not changed. The deviation between really measured actual tool length and theoretical tool length is calculated and written in memory DL. Then the tool compensation DL is checked concerning specified limits of wear LTOL and tool status TL is set.

At **Q350=±2 (control)** the theoretical tool geometries in memory L, DL is not changed. The deviation between really measured actual tool geometry and theoretical tool geometry is calculated and checked concerning specified limits of wear LTOL and tool status TL is set.

When exceeding the wear tolerance LTOL or the scattering tolerance Q362 the measurement is repeated (depending on parameter "number of repetitions on tolerance exceeding" in BLUM_0**.TAB line 84). So, it is possible to avoid to the greatest possible extent unique sources of error or sources of error by chance, e.g. falling coolant drops or adhering chips. The measuring value is again determined under slightly changed environmental conditions, e.g. repeated blow-off of the tool, and the result is either revised or confirmed.

⇨ For faultless tool the error marker/flag is set to Q199=0 and M1205=0. The machining of the parts can be continued without interruption.

⇨ For faulty tool the error marker/flag is set to Q199=1 or Q199=2 and M1205=1 and the tool is locked in tool table TOOL.T. All further actions, e.g. machine stop or loading of a sister tool, must be decided in the main program.

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

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

L, DL	Tool length, tool length compensation	(only at Q350=0/±1/±3)
TL	Tool locking	(only at Q350=±1/±2)

Following Q-parameters are set in the program flow:

Q199	Tool status parameters for tool breakage, tool wear and cutting edge control *Q199 = 0 → Tool dimension / cutting edge ok, continue work piece machining* *Q199 = 1 → Tool dimension outside the single wear tolerance (LTOL<DL<2*LTOL or LTOL<DL<LBREAK)* *Q199 = 2 → Tool dimension outside the double wear tolerance (DL>2*LTOL or DL>LBREAK)*

2.4 Cycle 584 – Tool Setting in Length and Radius with Concentricity Check

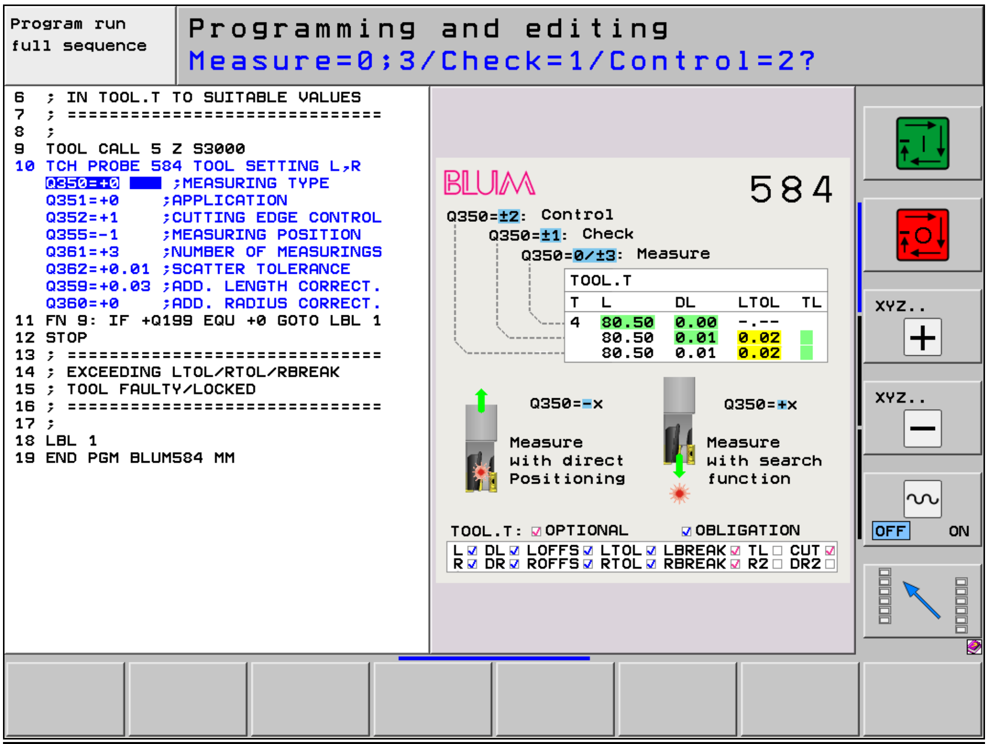
Cycle 584 is the program for the measurement of length and diameter of eccentric tools as well as an optional concentricity control with the BLUM laser system. The program can measure known tools (tool length/radius L≠0 / R≠0 mm) or unknown tools (tool length/radius L=0 / R=0). The result of the measurement is written into the tool table under the actual tool T-No. The measuring positions ROFFS, LOFFS must be defined tool specific in TOOL.T.

Caution: For unknown tool data (L=0, R=0) there is always the risk for collision with the BLUM laser light barrier. Therefore we recommend basically the input of rough tool geometry data into table TOOL.T (input accuracy / maximum deviation < ± 5 mm).

Following Q-parameters are used in the program call:

Q350	Measuring mode (note description of Q350 on page 9)
Q351	Measuring scope
Q352	Cutting edge control ±1=yes / 0=no
Q355	Measuring position
Q359	Additive length compensation
Q360	Additive radius compensation
Q361	Number of repetitive measurements for averaging
Q362	Permitted scattering out of tolerance of individual measuring values for repetitive measurement

Example for the call of the measuring cycle in the CNC program:



T	1
L	+0.0
DL	+0.0
R	+0.0
DR	+0.0
R2	xxx
DR2	xxx
TL	
CUT	5
ROFFS	+2.5
LOFFS	+1.5
LTOL	+0.02
RTOL	+0.02
LBREAK	xxx
RBREAK	+0.05

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

L, DL	Tool length, tool length correction (necessary at Q350=±1/±2/-3)
R, DR	Tool radius, tool radius correction (necessary at Q350=±1/±2/-3)
LTOL	Wear tolerance: length (only at Q350=±1/±2)
RTOL	Wear tolerance: radius (only at Q350=±1/±2)

LOFFS	Axial measuring position for radius measurement
ROFFS	Radial measuring position for length measurement
RBREAK	Breakage tolerance: radius (value from TOOL.T or BLUM_0**.TAB) (only at Q352=±1)
CUT	Number of tool cutting edges for concentricity control/cutting edge control (only at Q352=±1)

<u>Explanation:</u>

The tool with T-No.=5 is loaded into the spindle before call of cycle 584.

First the tool length is measured three times (Q361) 2.5 mm (ROFFS) outside the tool axis via multiple approaching cycle, and the scattering out of the individual measuring values concerning specified limits of 0.01 mm (Q362) is controlled. The additive length compensation value of 0.03 mm (Q359) is added to the tool compensation DL.

Then the tool radius is measured three times (Q361) 1.5 mm (LOFFS) above the tool tip, and the scattering out of the individual measuring values concerning specified limits of 0.01 mm (Q362) is controlled. The additive radius compensation value of 0.0 mm (Q360) is added to the tool compensation DR.

At least the 5 tool cutting edges (CUT) are controlled (Q352=1) with reduced test speed concerning specified limits of concentricity of 0.05 mm (RBREAK). The measuring position for the cutting edge control is axially 1.5 mm (LOFFS) above the tool tip and radially 0.05 mm (RBREAK) from the measured tool radius in the direction of the tool axis.

At **Q350=0/±3 (measure)** the measured tool geometries are written in memory L resp. R and the tool compensation set to DL=0 resp. DR=0.

At **Q350=±1 (check)** the theoretical tool geometries in memory L resp. R are not changed. The deviation between really measured actual tool geometry and theoretical tool geometry is calculated and written in memory DL resp. DR. Then the tool compensation DL resp. DR is checked concerning specified limits of wear LTOL resp. RTOL and tool status TL is set.

At **Q350=±2 (control)** the theoretical tool geometries in memory L, DL resp. R, DR are not changed. The deviation between really measured actual tool geometry and theoretical tool geometry is calculated and checked concerning specified limits of wear LTOL resp. RTOL and tool status TL is set.

When exceeding the wear tolerance LTOL or RTOL or the scattering tolerance Q362 the corresponding measurement is repeated (depending on parameter "number of repetitions on tolerance exceeding" in BLUM_0**.TAB line 84). So, it is possible to avoid to the greatest possible extent unique sources of error or sources of error by chance, e.g. falling coolant drops or adhering chips, which could be responsible for faulty results. The measuring value is again determined under slightly changed environmental conditions, e.g. repeated blow-off of the tool, and the result is either revised or confirmed.

⇨ For faultless tool the error marker/flag is set to Q199=0 and M1205=0. The machining of the parts can be continued without interruption.

⇨ For faulty tool the error marker/flag is set to Q199=1 or Q199=2 and M1205=1 and the tool is locked in tool table TOOL.T. All further actions, e.g. machine stop or loading of a sister tool, must be decided in the main program.

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

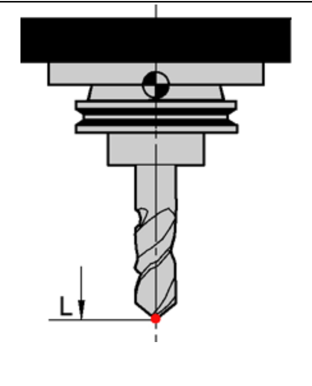
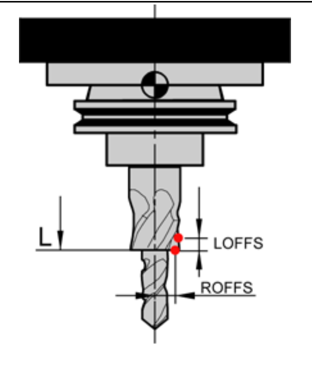
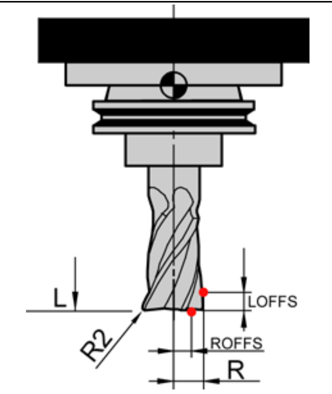
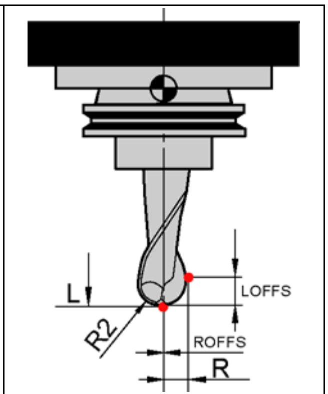
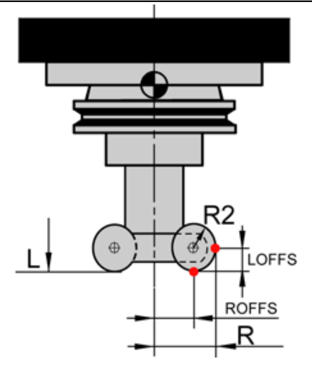
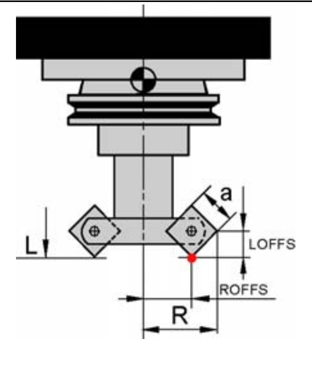
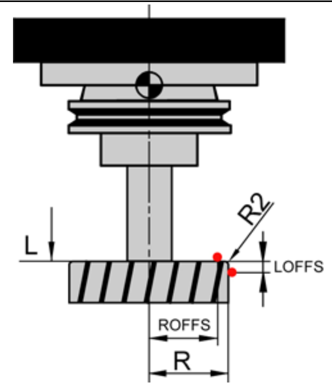
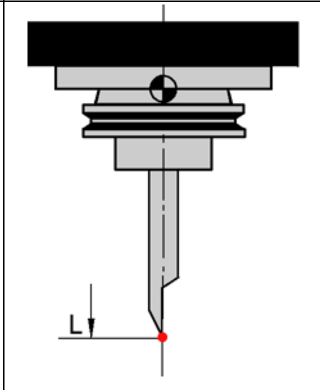
<u>Following parameters in TOOL.T are set in the program flow:</u>

L, DL	Tool length, tool length compensation (only at Q350=0/±1/±3)
R, DR	Tool radius, tool radius compensation (only at Q350=0/±1/±3)
TL	Tool locking (only at Q350=±1/±2)

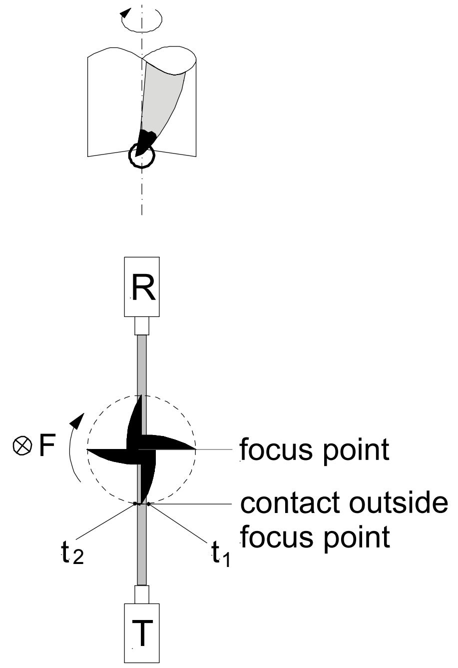
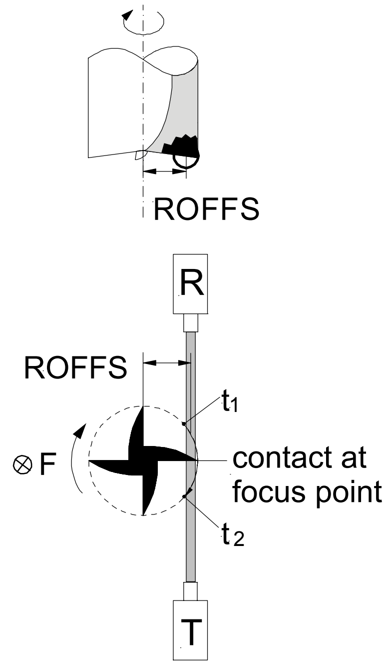
<u>Following Q-parameters are set in the program flow:</u>

Q199	Tool status parameter for tool breakage, tool wear and cutting edge control *Q199 = 0 → Tool dimension / cutting edge ok, continue work piece machining* *Q199 = 1 → Tool dimension outside the single wear tolerance* *(LTOL<DL<2*LTOL or DL<LBREAK or RTOL<DR<2*RTOL or DR<RBREAK)* *Q199 = 2 → Tool dimension outside the double wear tolerance, tool / cutting edge broken* *(DL>2*LTOL or DL>LBREAK or DR>2*RTOL or DR>RBREAK)*

Examples for the measuring positions for various tool types:

<table>
<tr>
<td>
ROFFS=0</td>
<td>
1) ROFFS=0
2) ROFFS=R-0.5</td>
<td>
ROFFS=R-R2-0.5
LOFFS=R2+0.5</td>
<td>
ROFFS=0
LOFFS=R2+0.5</td>
</tr>
<tr>
<td>
ROFFS=R-R2-0.5
LOFFS=R2</td>
<td>
ROFFS=R–0.707a-0.5
LOFFS=0.707a</td>
<td>
ROFFS=R-R2-0.5
LOFFS= -(R2+0.5)</td>
<td>
ROFFS=0…0.1</td>
</tr>
</table>

Reasons for the eccentric measuring positions for the length measurement:

<table>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Wrong!
1. Side shadowed profile of the cutter
2. Contact outside of the focus point
3. Rest time in the beam t2-t1 is very short</td>
<td>Correct!
1. Front shadowed profile of the cutter
2. Contact in the focus point
3. Rest time in the beam t2-t1 is greater</td>
</tr>
</table>

2.5 Cycle 585 – Single Cutting Edge Control on Straight Edge Geometry

To ensure faultless production of the parts, critical tools must be controlled before or after machining concerning cutting edge breakage or wear. The rotating tool is moved through the beam so that all cutting diameters can be checked concerning specified limits of concentricity.

Cycle 585 is the program for the single cutting edge control of multiblade tools with the BLUM laser system. The program is based on a tool with known tool data which are read from the tool memory via the tool no. T.

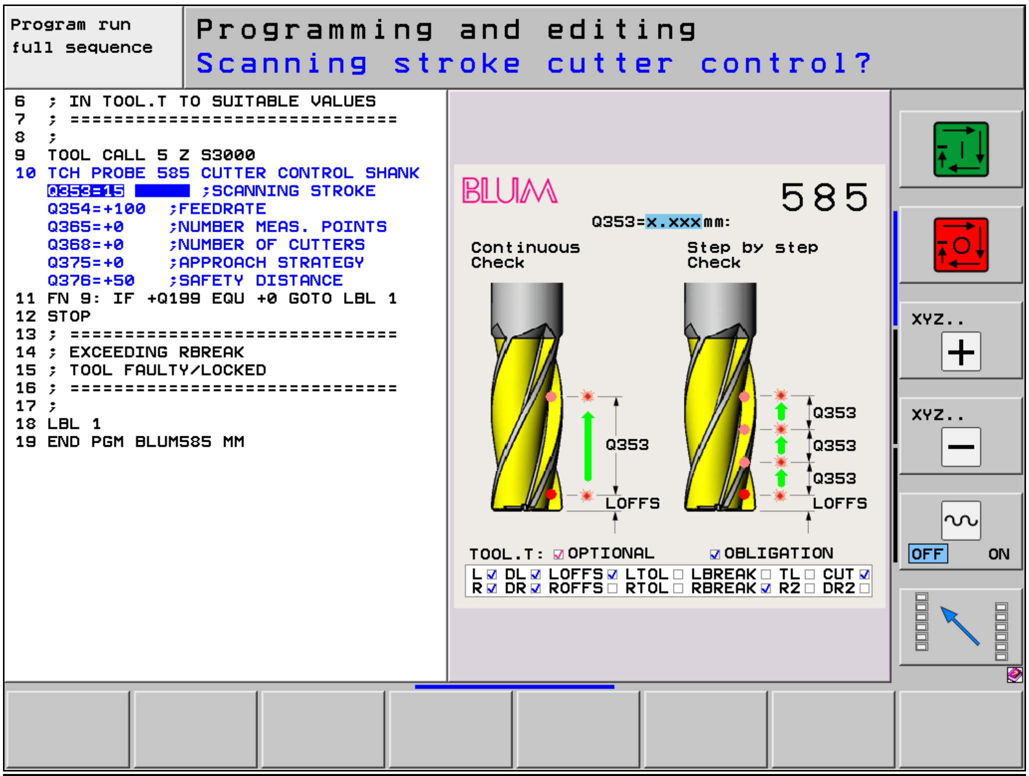
Following Q-parameters are used for program call:

Q353	Control travel on the tool shank
Q354	Control feed (F)
Q365	Number of measuring points
Q368 optional	Number of cutting edges *The number of cutting edges CUT is either read from the tool table TOOL.T or derived from Q368≠0 (ABS Q368 = number of cutting edges, SIGN Q368 = evaluation rule):* *Q368 > 0 → CUT=ABS Q368 (for multiblade disk mills with great diameter etc.)* *normal evaluation of signal Dynamic (DYN=HIGH → tool in order)* *Q368 = 0 → CUT from TOOL.T, default value for end mills, ball-nose mills , end-face mills etc.* *normal evaluation of signal Dynamic (DYN=HIGH → tool in order)* *Q368 < 0 → CUT=ABS Q368 (for grinding disks and saw blades use Q368=−1)* *invers evaluation of signal Dynamic (DYN=HIGH → tool n.i.o.)* ***Notice**: If Q368≠0, then ABS Q368 has priority against CUT from TOOL.T*
Q375 optional	Approach Strategy *Definition in which axis sequence the starting position should be approached.* *Q375 = 0 → Retraction of the length axis to safety position* *Positioning of beam, radius and length axis consecutively* *Q375 = 1 → Retraction of the length axis to safety position* *Positioning of beam, radius and length axis simultaneously* *Q375 = 2 → Direct positioning of beam, radius and length axis simultaneously* ***!! Attention: increased risk of collision !!***
Q376 optional	Safety Distance *The distance determines whether the length axis is directly positioned into the beam (Q376=0) or pre-positioned by this value before/above the beam (Q376>0).* *Is Q376>0 defined the length axis is - for safety reasons - at first brought to this position before beam and radius axis will be positioned.* ***Attention:** Depending on initial position + approach strategy there will be increased risk of collision!*

Following parameters must be defined in tool table TOOL.T:

L, DL	Tool length, tool length correction *The parameters L + DL must be assigned with valid values (known tool length).*
R, DR	Tool radius, tool radius correction *The parameters R + DR must be assigned with valid values (known tool radius).*
RBREAK	Breakage tolerance: radius, permitted concentricity tolerance for cutting edge control
LOFFS	Axial start/test position for concentricity control
CUT	Number of tool cutting edges for cutting edge control

Example for the call of the measuring cycle in the CNC program:



T	1
L	+95.6
DL	+0.01
R	+5.5
DR	-0.01
R2	xxx
DR2	xxx
TL	
CUT	5
ROFFS	xxx
LOFFS	+1.5
LTOL	xxx
RTOL	xxx
LBREAK	xxx
RBREAK	+0.02

Explanation:

The tool with T-No.=5 is loaded into the spindle before call of cycle 585.

The length L+DL and radius R+DR of the tool are taken from the tool table. For the cutting edge depending test speed the 5 tool cutting edges (CUT) are controlled concerning specified limits of concentricity of 0.02 mm (RBREAK). If the calculated test speed exceeds the max. determined spindle speed for this tool as indicated in the last TOOL CALL, the cycle is interrupted and the error message “error spindle speed” is displayed. This avoids setting of a too high spindle speed and the associated risk of tool breakage.

With **Q375=0** at first the length axis is brought to retraction position. The beam, the radius and at least the length axis is brought to the starting position 50 mm (Q376) before/above the beam.

The starting position of the concentricity control is axially 1.5 mm (LOFFS) above the tool tip L and radially 0.02 mm (RBREAK) from the tool radius R in tool axis direction.

At **Q365=0** the control of the defined cutting edge range is effected constantly. The tool, rotating with test speed, moves with a speed of 100 mm/min (Q354) on the programmed path curve from starting to target position (LOFFS) by 15 mm (Q353) continuously in negative axial direction. The programmed path curve for the cutting edge control is in equidistant distance to the real tool contour by the permitted concentricity tolerance. Appearance of a trigger signal is controlled within the travel.

At **Q365>=1** the control of the defined cutting edge range is effected step by step. The cutting edge control starts in distance LOFFS from the tool tip (e.g. first tooth row on the thread milling cutter). During one revolution the radius tolerance is controlled on this cutting edge position. According to the number of the measuring points (Q365) all further tooth rows are displaced each by the step width (Q353) to cutting edge breakage.

- At pitch=0 of the tooth rows (e.g. thread milling cutter) CUT="number of teeth per each row" and Q353="axial distance of tooth rows" must be set.
- At pitch>0 of the tooth rows (e.g. thread drill) CUT=1 and Q353="axial distance tooth to tooth" must be set.

If the cutting edge contour exceeds the permitted concentricity tolerance against the default values from the tool memory and the parameters, the measuring process is repeated again.

- ⇨ If no signal change is detectable within the measuring way, all cutting edges are within the permitted concentricity tolerance. For faultless tool the error marker/flag is set to Q199=0 and M1205=0. The machining of the parts can be continued without interruption.
- ⇨ If a signal change is detectable within the measuring way, minimum one cutting edge is outside the permitted concentricity tolerance. For faulty tool the error marker/flag is set to Q199=2 and M1205=1 and the tool is locked in tool table TOOL.T. All further actions, e.g. machine stop or loading of a sister tool must be decided in the main program.

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

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

TL	Tool locking

Following Q-parameters are set in the program flow:

Q199	Tool status parameter for tool breakage and cutting edge control *Q199 = 0 → Tool / cutting edge ok, continue work piece machining* *Q199 = 2 → Tool / cutting edge broken, load in sister tool*

2.6 Cycle 586 – Tool Breakage Detection

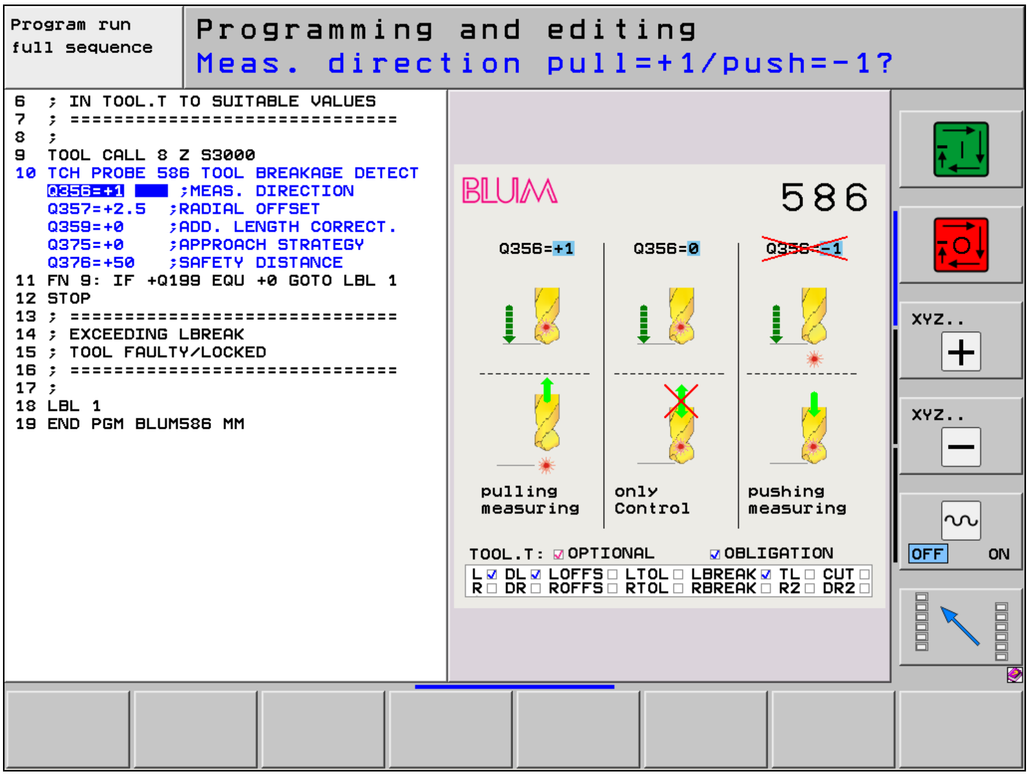
To ensure faultless production of the parts, critical tools must be controlled before or after machining concerning tool breakage or wear. The rotating tool is moved through the beam so that the tool length is measured and can be controlled concerning specified limits of length.

Cycle 586 is the program for the tool breakage monitoring of centric tools with the BLUM laser system. The program is based on a tool with known tool data which are read from the tool memory via the tool no. T.

<u>Following Q-parameters are used for program call:</u>

Q356	Measuring direction
Q357	Eccentric control position
Q359 optional	Additive length compensation (= tool core length – tool tip length !)
Q375 optional	Approach Strategy *Definition in which axis sequence the starting position should be approached.* *Q375 = 0 → Retraction of the length axis to safety position* *Positioning of beam, radius and length axis consecutively* *Q375 = 1 → Retraction of length axis to safety position* *Positioning of beam, radius and length axis simultaneously* *Q375 = 2 → Direct positioning of beam, radius and length axis simultaneously* ***!! Attention: increased risk of collision !!***
Q376 optional	Safety Distance *The distance determines whether the length axis is directly positioned into the beam (Q376=0) or pre-positioned by this value before/above the beam (Q376>0).* *Is Q376>0 defined the length axis is - for safety reasons - at first brought to this position before beam and radius axis will be positioned.* ***<u>Attention:</u>*** *Depending on initial position + approach strategy there will be increased risk of collision!*

<u>Example for the call of the measuring cycle in the CNC program:</u>



T	1
L	+84.5
DL	+0.01
R	xxx
DR	xxx
R2	xxx
DR2	xxx
TL	
CUT	xxx
ROFFS	xxx
LOFFS	xxx
LTOL	xxx
RTOL	xxx
LBREAK	+0.999
RBREAK	xxx

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

L, DL	Tool length, tool length correction *The parameters L + DL must be assigned with valid values (known tool length).*
LBREAK	Breakage tolerance: length (value from TOOL.T or BLUM_0**.TAB)

Explanation:

The tool with T-No.=8 is loaded into the spindle before call of cycle 586.

With **Q375=0** at first the length axis is brought to retraction position. The beam, the radius and at least the length axis is brought to the starting position 50 mm (Q376) before/above the beam.

At **Q356=+1** the tool tip is positioned 5 mm below the laser beam middle, offset by 2.5 mm (Q357) from the tool axis, so that the beam is interrupted. The measuring block for measurement of the tool length is effected out of the beam "from dark to light" in Z+ direction. If the permitted tolerance of 0.9999 mm (LBREAK) is exceeded in tool length, the measuring block is repeated again.

At **Q356=0** the tool tip is positioned 5 mm below the laser beam middle, so that the beam is interrupted. In this position, the signals "Laser ok"=0 and "Dynamic"=0 are checked. A tool length measurement is not performed.

At **Q356=–1** the tool tip is positioned 5 mm above the laser beam middle, offset by 2.5 mm (Q357) from the tool axis, so that the beam is enabled. The measuring block for measurement of the tool length is effected into the beam in Z– direction "from light to dark" (e.g. also for control of engraving styli or tools with very small shank diameters). If the permitted tolerance of 0.9999 mm (LBREAK) is exceeded in tool length, the measuring block is repeated again.

At the measuring strategy "from dark to light" out of the beam (Q356=+1) not the length to the tool tip is measured but the length of the tool core. If there is a too big difference between core and tip length, e.g. at shaft milling cutters with large insertion angle on the lower edge, the breakage tolerance LBREAK can also be easily exceeded when the tool is not broken. In this case the difference value ΔL=(core length – tip length) which must be considered in the parameter Q359 as length compensation value can be transferred to the cycle. The tool is broken, if: $(L_{TAB} + DL_{TAB} + Q359 - L_{Actual}) > LBREAK$

- ⇨ If the tool is ok, the error marker/flag is set to Q199=0 and M1205=0. The machining of the parts can be continued without interruption.
- ⇨ If the tool is broken the error marker/flag is set to Q199=2 and M1205=1 and the tool is locked in the tool table TOOL.T. All further actions, e.g. machine stop or loading of a sister tool must be decided in the main program.

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

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

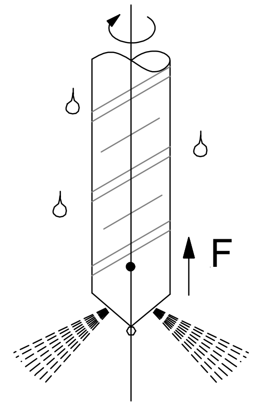
TL	Tool locking

Following Q-parameters are set in the program flow:

Q199	Tool status parameters for tool breakage and cutting edge control *Q199 = 0 → tool / cutting edge ok, continue work piece machining* *Q199 = 2 → tool / cutting edge broken, load in sister tool*
Q359	Result parameter length control Q359 = -999.999 → *Tool is faulty (Q199=2), because the light barrier was not interrupted by the tool tip in the start position or a trigger signal was detected on the dynamic output (e.g. if the tool is too short: L_{Actual} << L+DL)* Q359 = +999.999 → *Tool is faulty (Q199=2), because the light barrier was interrupted by the tool tip in the start position, but no trigger signal was detected on tool length measurement (e.g. if the tool is too long: L_{Actual} >> L+DL)* *Q359 = x.xxx mm → measured deviation = actual – nominal value from tool length control*

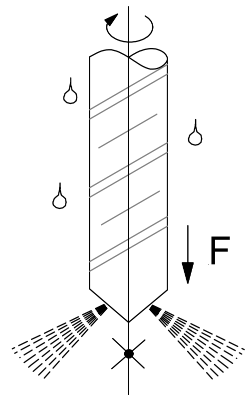
Examples for the usage of cycle 586:

Tool breakage detection of drills



Correct!

```
...
TCH PROBE 586 TOOL BREAKAGE DETECTION
 Q356=+1       ; MEAS. DIRECTION
 Q357=0        ; RADIAL OFFSET
 Q359=0        ; ADD. LENGTH COMPENSATION
 Q375=0        ; APPROACH STRATEGY
 Q376=50       ; SAFETY DISTANCE
...
```

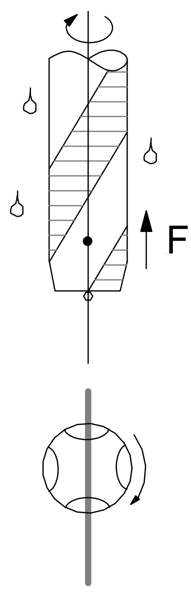


Wrong!

```
...
TCH PROBE 586 TOOL BREAKAGE DETECTION
 Q356=-1       ; MEAS. DIRECTION
 Q357=0        ; RADIAL OFFSET
 Q359=0        ; ADD. LENGTH COMPENSATION
 Q375=0        ; APPROACH STRATEGY
 Q376=50       ; SAFETY DISTANCE
...
```

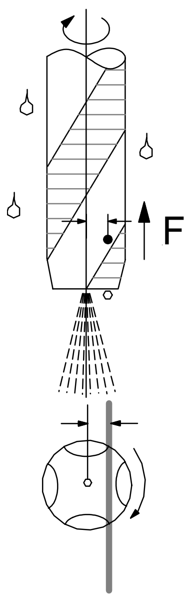
(Wrong measuring direction)

Tool breakage detection of tapping drills



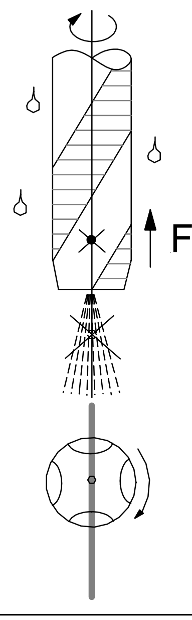
Correct!

```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357=0
 Q359=0
 Q375=0
 Q376=50
...
```



Correct!

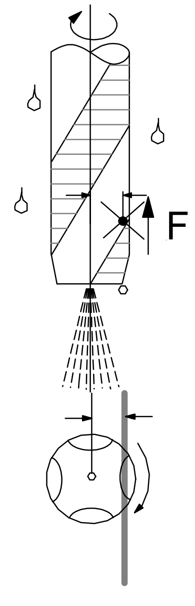
```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357>=0
 Q359=0
 Q375=0
 Q376=50
...
```



Wrong!

```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357=0
 Q359=0
 Q375=0
 Q376=50
...
```

(wrong meas. position!)

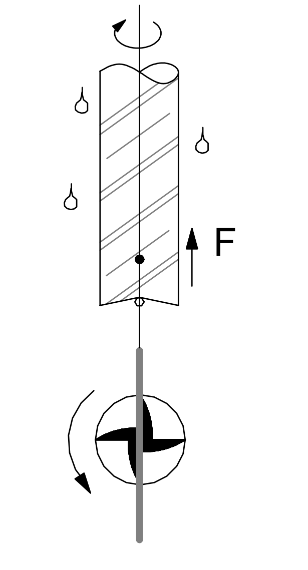


Wrong!

```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357>>0
 Q359=0
 Q375=0
 Q376=50
...
```

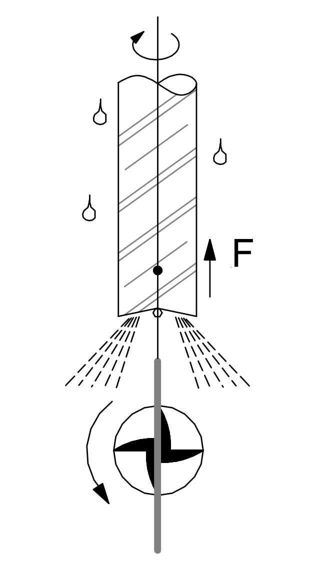
(wrong meas. position!)

Tool breakage detection of end-face mills



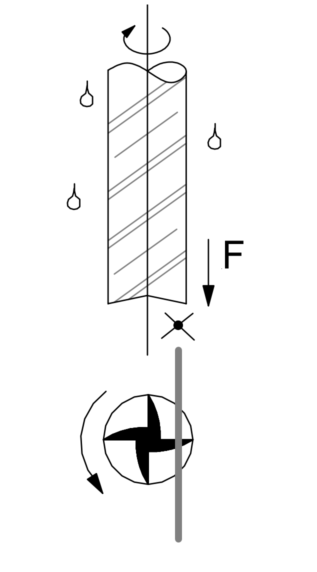
Correct!

```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357=0
 Q359<=0
 Q375=0
 Q376=50
...
```



Correct!

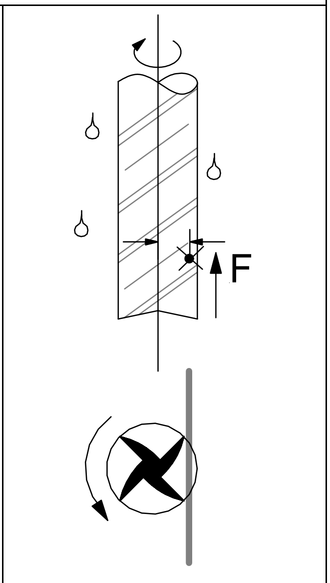
```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357=0
 Q359<=0
 Q375=0
 Q376=50
...
```



Wrong!

```
...
TCH PROBE 586 BREAKAGE
 Q356=-1
 Q357>=0
 Q359=0
 Q375=0
 Q376=50
...
```

(wrong meas. direction!)

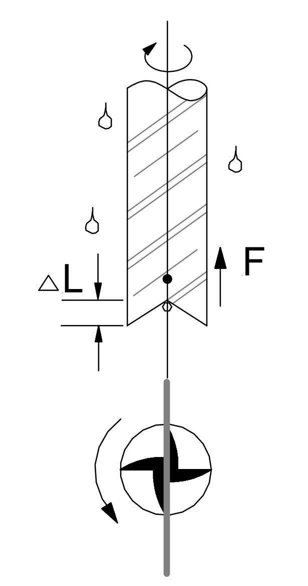


Wrong!

```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357>>0
 Q359=0
 Q375=0
 Q376=50
...
```

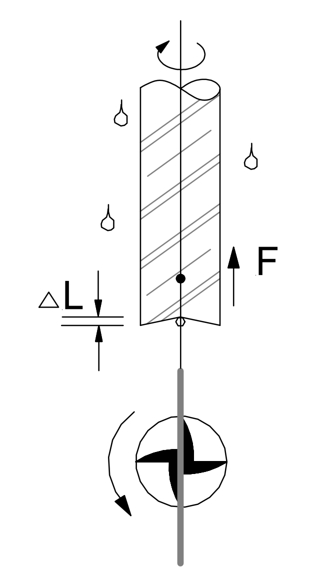
(wrong meas. position!)

Tool breakage detection of end-face mills



Correct!

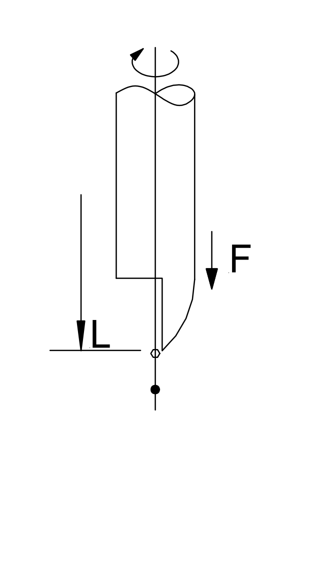
```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357=0
 Q359=-ΔL
 Q375=0
 Q376=50
...
```



Correct!

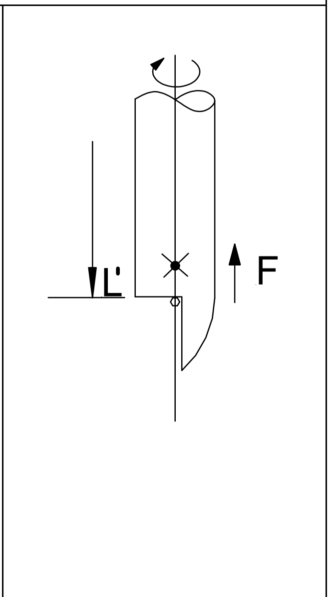
```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357=0
 Q359=-ΔL
 Q375=0
 Q376=50
...
```

Tool breakage detection of engraving mills



Correct!

```
...
TCH PROBE 586 BREAKAGE
 Q356=-1
 Q357=0
 Q359=0
 Q375=0
 Q376=50
...
```



Wrong!

```
...
TCH PROBE 586 BREAKAGE
 Q356=+1
 Q357=0
 Q359=0
 Q375=0
 Q376=50
...
```

(wrong meas. direction)

2.7 Cycle 587 – Single Cutting Edge Control on Rounded Edge Geometry

To ensure a faultless production of the parts, critical tools must be controlled before and after machining with respect to cutting edge breakage or wear. The rotating tool is moved through the beam so that all cutting edges along their contour are controlled concerning specified limits of form. The form control can only be effected with the required accuracy with a focussed laser system (MICRO Compact System). Furthermore, always both sides must be calibrated with the BLUM reference tool (Q57=0), as radial and axial laser beam middle must be taken into consideration for path calculation.

Cycle 587 is the program for the single cutting edge control of multiblade tools with ball-ended head / angle radius with the BLUM laser system. The program is based on a tool with known tool data which are read from the tool memory via the tool no. T.

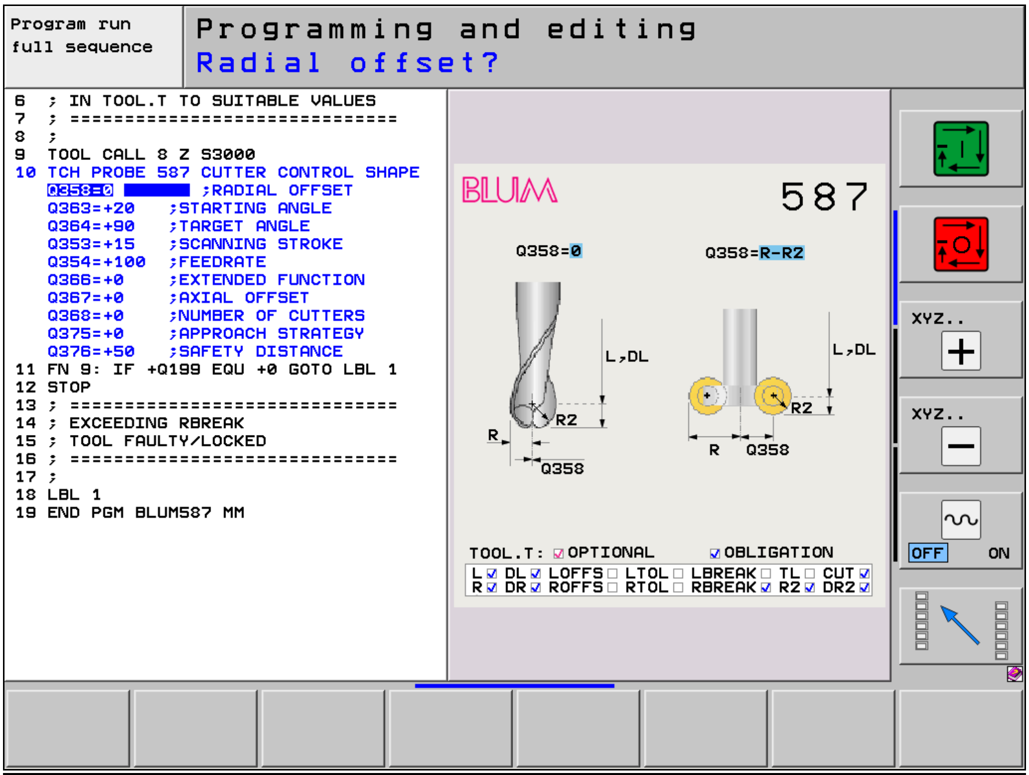
<u>Following Q-parameters are used for program call:</u>

Q353	Control travel on the tool shank
Q354	Control feed (F)
Q358	Radial offset to the starting point of the angle radius
Q363	Starting angle Alpha to symmetry axis
Q364	Target angle Beta to symmetry axis
Q366 optional	Extended functionality (0=inactive / 1,2=active) *Q366 = 0 → L=L_{Tip}, R, $R2_{Reg}$, $DR2_{Reg}$ in TOOL.T is set referring to regression radius* *Q366 = 1 → L=$L_{CC\ Reg}$, R, $R2_{Reg}$, $DR2_{Reg}$ in TOOL.T is set referring to regression radius* *Q366 = 2 → L=$L_{CC\ Nom}$, R, $R2_{Nom}$, $DR2_{Nom}$ in TOOL.T is set referring to nominal radius*
Q367 optional	Axial Offset (signed axial distance from the tool length L in TOOL.T to the circle centre of R2) (only at Q366≠0)
Q368 optional	Number of cutting edges *The number of cutting edges CUT is either read from the tool table TOOL.T or derived from Q368≠0 (ABS Q368 = number of cutting edges, SIGN Q368 = evaluation rule):* *Q368 > 0 → CUT=ABS Q368 (for saw blades with great diameter and multiple teeth, etc.)* *normal evaluation of signal Dynamic (DYN=HIGH → tool in order)* *Q368 = 0 → CUT from TOOL.T, default value for end mills, ball-nose mills , end-face mills etc.* *normal evaluation of signal Dynamic (DYN=HIGH → tool in order)* *Q368 < 0 → CUT=ABS Q368 (for grinding disks and saw blades use Q368=−1)* *inverse evaluation of signal Dynamic (DYN=HIGH → tool n.i.o.)* ***<u>Notice</u>****: If Q368≠0, then ABS Q368 has priority against CUT from TOOL.T*
Q375 optional	Approach Strategy *Definition in which axis sequence the starting position should be approached.* *Q375 = 0 → Retraction of the length axis to safety position* *Positioning of beam, radius and length axis consecutively* *Q375 = 1 → Retraction of length axis to safety position* *Positioning of beam, radius and length axis simultaneously* *Q375 = 2 → Direct positioning of beam, radius and length axis simultaneously* ***!! Attention: increased risk of collision !!***
Q376 optional	Safety Distance *The distance determines whether the length axis is directly positioned into the beam (Q376=0) or pre-positioned by this value before/above the beam (Q376>0).* *Is Q376>0 defined the length axis is - for safety reasons - at first brought to this position before beam and radius axis will be positioned.* ***<u>Attention:</u>*** *Depending on initial position + approach strategy there will be increased risk of collision!*

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

L, DL	Tool length, tool length correction *The parameters L + DL must be assigned with valid values (known tool length).* *(referring to the circle centre L_{CC} or to the tool tip L_{Tip})*
R, DR	Tool radius, tool radius correction *The parameters R + DR must be assigned with valid values (known tool radius).*
R2, DR2	Tool radius R2: angle radius / cutting edge radius / ball-ended radius *The parameters R2 + DR2 must be assigned with valid values (known cutting edge radius)*
RBREAK	Breakage tolerance: radius, permitted form tolerance for cutting edge control
CUT	Number of tool cutting edges for cutting edge control

Example for the call of the measuring cycle in the CNC program:



T	1
L	+84.5
DL	+0.1
R	+7.0
DR	+0.0
R2	+7.0
DR2	+0.0
TL	
CUT	5
ROFFS	xxx
LOFFS	xxx
LTOL	xxx
RTOL	xxx
LBREAK	xxx
RBREAK	+0.02

Explanation:

The tool with no. T=8 is loaded into the spindle before call of cycle 587.

Length L+DL and radius R+DR of the tool are taken from the tool table. For the cutting edge depending test speed the 5 tool cutting edges (CUT) are controlled concerning specified limits of form of 0.02 mm (RBREAK) on the cutting edge radius 7.0 mm (R2) and on the tool shank W=15 mm (Q353). If the calculated test speed exceeds the max. determined spindle speed for this tool as given in the last TOOL CALL, the cycle is interrupted and the error message “error spindle speed” is displayed. This avoids setting of a too high spindle speed and the associated risk of tool breakage.

With **Q375=0** at first the length axis is brought to retraction position. The beam, the radius and at least the length axis is brought to the starting position 50 mm (Q376) before/above the beam.

The tool rotating with test speed moves with measuring feed on the programmed path from starting to target position. The programmed path curve for the form control is in equidistant distance to the real tool contour by the permitted form tolerance. Appearance of a trigger signal is controlled within the travel.

If the cutting edge contour exceeds the permitted concentricity tolerance against the default values from the tool memory and the parameters, the measuring process is repeated again.

⇨ If no signal change is detectable within the measuring way, all cutting edges are within the permitted concentricity tolerance. For faultless tool the error marker/flag is set to Q199=0 and M1205=0. The machining of the parts can be continued without interruption.

⇨ If a signal change is detectable within the measuring way, minimum one cutting edge is outside the permitted concentricity tolerance. For faulty tool the error marker/flag is set to Q199=2 and M1205=1 and the tool is locked in tool table TOOL.T. All further actions, e.g. machine stop or loading of a sister tool must be decided in the main program.

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

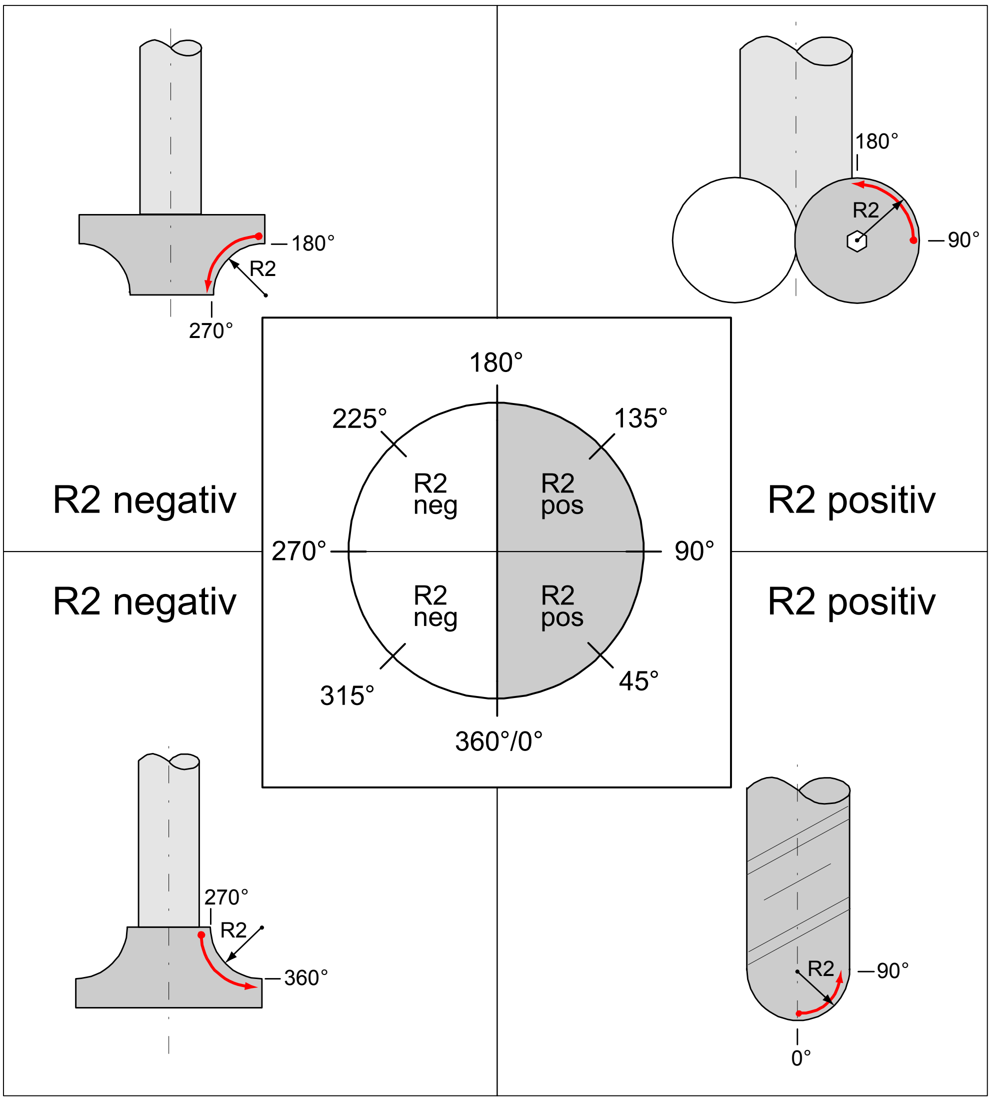
Following parameters in TOOL.T are set in the program flow:

TL	Tool locking

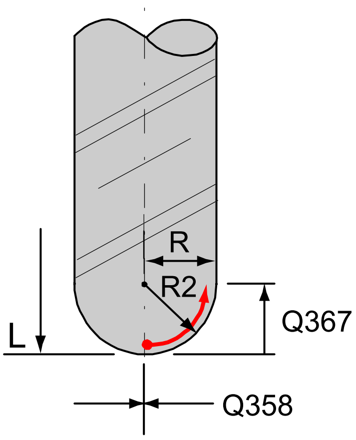
Following Q-parameters are set in the program flow:

Q199	Tool status parameter for tool breakage and cutting edge control *Q199 = 0 → tool / cutting edge ok, continue work piece machining* *Q199 = 2 → tool / cutting edge broken, load in sister tool*

Setting of the edge radius R2, starting angle Q363, target angle Q364 depending on the tool geometry



Example for the call of the measuring cycle depending on the predefined tool length L in TOOL.T



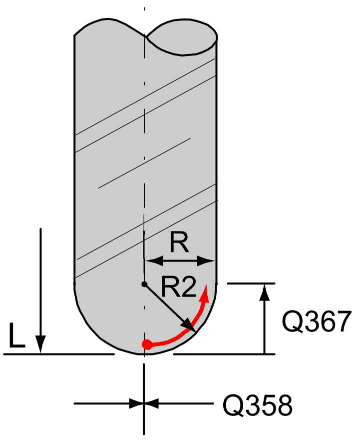
Blum cycle call:

```
...
TOOL CALL 7 Z S3000 FMAX
TCH PROBE 587 CUTTING EDGE CONTROL KF
 Q358=0       ; RADIAL OFFSET
 Q363>=10     ; STARTING ANGLE
 Q364<=90     ; TARGET ANGLE
 Q353=5       ; TRAVEL
 Q354=100     ; FEED
 Q366=2       ; EXTENDED FUNCTION
 Q367=-R2     ; AXIAL OFFSET
 Q368=0       ; NUMBER OF CUTTERS
 Q375=0       ; APPROACH STRATEGY
 Q376=50      ; SAFETY DISTANCE
...
```

Notice: spindle speed is calculated automatically

Predefined parameter in TOOL.T:

L, DL ; Nominal tool length (**L**$_{Tip}$)
R, DR ; Nominal tool radius
R2>0, DR2 ; Nominal edge radius e.g. R2=+5.0mm
CUT ; Number of cutters
RBREAK ≈0.05...0.2mm



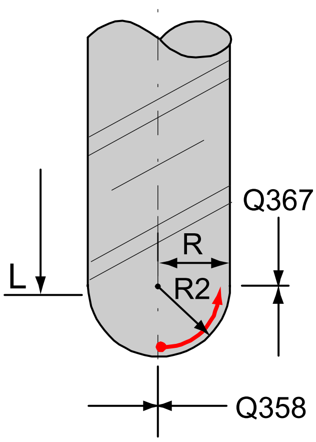
alternatively:

```
...
TOOL CALL 7 Z S3000 FMAX
TCH PROBE 587 CUTTING EDGE CONTROL KF
 Q358=0       ; RADIAL OFFSET
 Q363>=10     ; STARTING ANGLE
 Q364<=90     ; TARGET ANGLE
 Q353=5       ; TRAVEL
 Q354=100     ; FEED
 Q366=0       ; EXTENDED FUNCTION
 Q367=0       ; AXIAL OFFSET
 Q368=0       ; NUMBER OF CUTTERS
 Q375=0       ; APPROACH STRATEGY
 Q376=50      ; SAFETY DISTANCE
...
```

Notice: spindle speed is calculated automatically

Predefined parameter in TOOL.T:

L, DL ; Nominal tool length (**L**$_{Tip}$)
R, DR ; Nominal tool radius
R2>0, DR2 ; Nominal edge radius e.g. R2=+5.0mm
CUT ; Number of cutters
RBREAK ≈0.05...0.2mm



Blum cycle call:

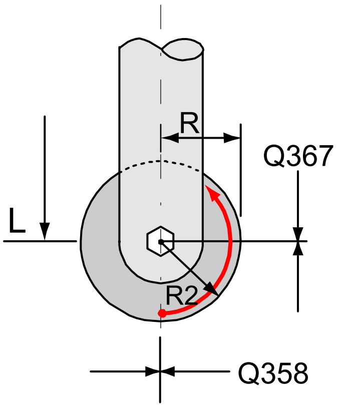
```
...
TOOL CALL 7 Z S3000 FMAX
TCH PROBE 587 CUTTING EDGE CONTROL KF
 Q358=0       ; RADIAL OFFSET
 Q363>=10     ; STARTING ANGLE
 Q364<=90     ; TARGET ANGLE
 Q353=5       ; TRAVEL
 Q354=100     ; FEED
 Q366=2       ; EXTENDED FUNCTION
 Q367=0       ; AXIAL OFFSET
 Q368=0       ; NUMBER OF CUTTERS
 Q375=0       ; APPROACH STRATEGY
 Q376=50      ; SAFETY DISTANCE
...
```

Notice: spindle speed is calculated automatically

Predefined parameter in TOOL.T:

L, DL ; Nominal tool length (**L**$_{CC}$)
R, DR ; Nominal tool radius
R2>0, DR2 ; Nominal edge radius e.g. R2=+5.0mm
CUT ; Number of cutters
RBREAK ≈0.05...0.2mm

Example for the call of the measuring cycle depending on the predefined tool length L in TOOL.T



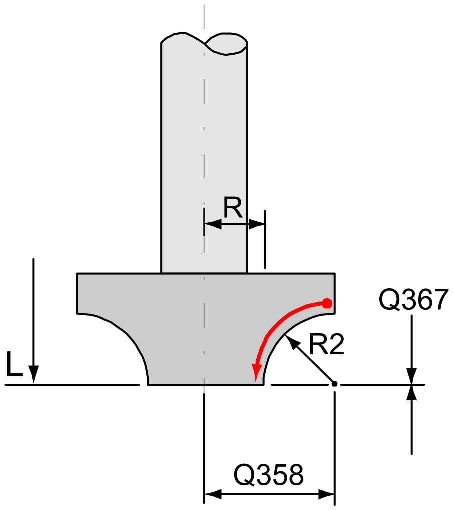
Blum cycle call:

```
...
TOOL CALL 7 Z S3000 FMAX
TCH PROBE 587 CUTTING EDGE CONTROL KF
 Q358=0       ; RADIAL OFFSET
 Q363>=10     ; STARTING ANGLE
 Q364<=120    ; TARGET ANGLE
 Q353=0       ; TRAVEL
 Q354=100     ; FEED
 Q366=2       ; EXTENDED FUNCTION
 Q367=0       ; AXIAL OFFSET
 Q368=0       ; NUMBER OF CUTTERS
 Q375=0       ; APPROACH STRATEGY
 Q376=50      ; SAFETY DISTANCE
...
```

Notice: spindle speed is calculated automatically

Predefined parameter in TOOL.T:

L, DL ; Nominal tool length (L_{CC})
R, DR ; Nominal tool radius
R2>0, DR2 ; Nominal edge radius e.g. R2=+5.0mm
CUT ; Number of cutters
RBREAK ≈0.05...0.2mm



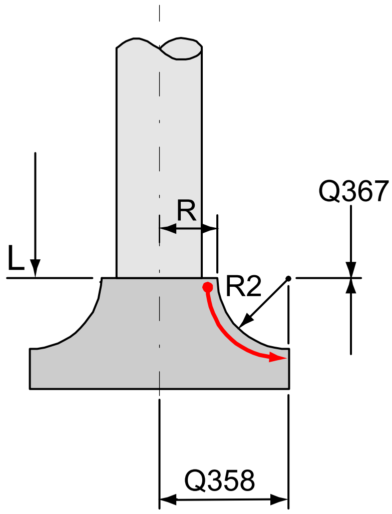
Blum cycle call:

```
...
TOOL CALL 7 Z S3000 FMAX
TCH PROBE 587 CUTTING EDGE CONTROL KF
 Q358=R+ABS(R2) ; RADIAL OFFSET
 Q363>=180    ; STARTING ANGLE
 Q364<=270    ; TARGET ANGLE
 Q353=0       ; TRAVEL
 Q354=100     ; FEED
 Q366=2       ; EXTENDED FUNCTION
 Q367=0       ; AXIAL OFFSET
 Q368=0       ; NUMBER OF CUTTERS
 Q375=0       ; APPROACH STRATEGY
 Q376=50      ; SAFETY DISTANCE
...
```

Notice: spindle speed is calculated automatically

Predefined parameter in TOOL.T:

L, DL ; Nominal tool length (L_{CC})
R, DR ; Nominal tool radius
R2<0, DR2 ; Nominal edge radius e.g. R2= -5.0mm
CUT ; Number of cutters
RBREAK ≈0.05...0.2mm



Blum cycle call:

```
...
TOOL CALL 7 Z S3000 FMAX
TCH PROBE 587 CUTTING EDGE CONTROL KF
 Q358=R+ABS(R2) ; RADIAL OFFSET
 Q363>=270    ; STARTING ANGLE
 Q364<=360    ; TARGET ANGLE
 Q353=0       ; TRAVEL
 Q354=100     ; FEED
 Q366=2       ; EXTENDED FUNCTION
 Q367=0       ; AXIAL OFFSET
 Q368=0       ; NUMBER OF CUTTERS
 Q375=0       ; APPROACH STRATEGY
 Q376=50      ; SAFETY DISTANCE
...
```

Notice: spindle speed is calculated automatically

Predefined parameter in TOOL.T:

L, DL ; Nominal tool length (L_{CC})
R, DR ; Nominal tool radius
R2<0, DR2 ; Nominal edge radius e.g. R2= -5.0mm
CUT ; Number of cutters
RBREAK ≈0.05...0.2mm

2.8 Cycle 588 – Tool Setting in Length, Radius and Edge Radius

Cycle 588 is the program for the measurement of length, radius and angle radius of tools with circular cutting edge geometry with the BLUM laser system. The program relies on a tool with known/unknown tool data (length L≠0/radius R≠0/angle radius R2≠0 mm), which are read from the tool memory via the valid tool no. T.

Caution: For unknown tool data (L=0, R=0) there is always the risk of collision with the Blum laser system. Therefore we recommend basically the input of rough tool geometries into the tool table TOOL.T (input accuracy/max. deviation for L and R < ±5 mm, max. deviation for Q358, Q367, R2 < ±0.5 mm).

Following Q-parameters are used for program call:

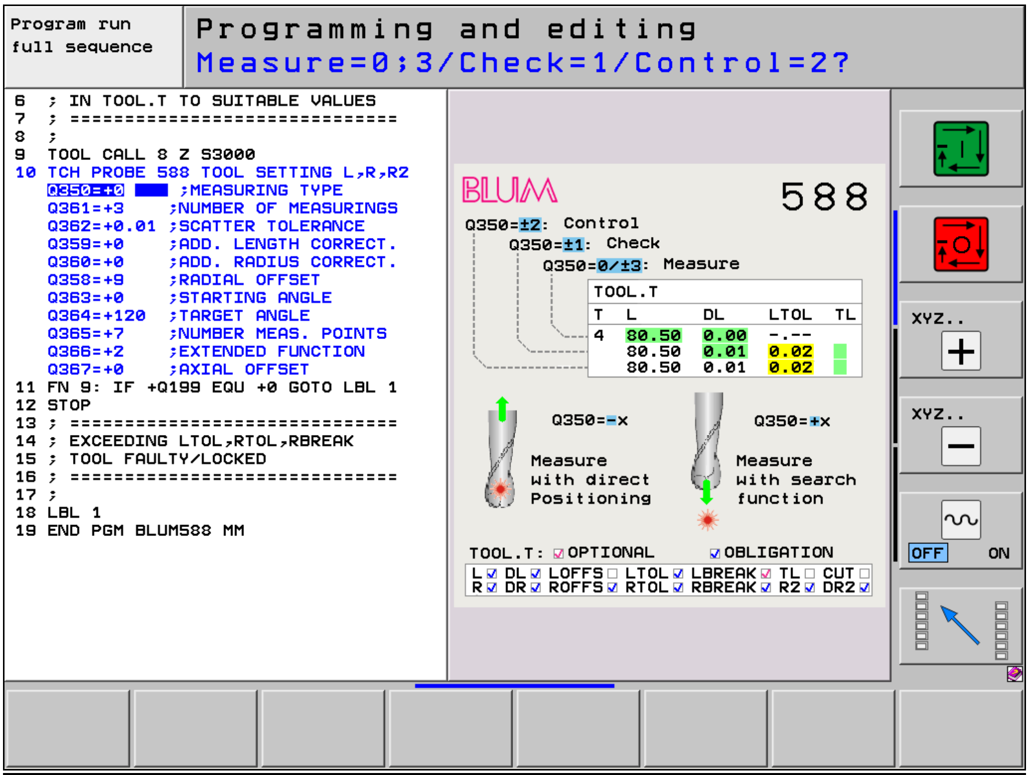
Q350	Measuring mode (note description of Q350 on page 9)
Q358	Radial offset to the angle radius
Q359	Additive length correction
Q360	Additive radius correction
Q361	Number of repetitive measurements for averaging
Q362	Permitted scattering out of tolerance of individual measuring values for repetitive measurement
Q363	Starting angle Alpha to the symmetry axis
Q364	Target angle Beta to the symmetry axis
Q365	Number of measuring points
Q366 optional	Extended functionality (0=inactive / 1,2=active) *Q366 = 0 → Determination $L_{CC\ Reg}$, R, $R2_{Reg}$, $DR2_{Reg}$ refer. to regression radius (only pos. R2) no result file created* *Q366 = 1 → Determination $L_{CC\ Reg}$, R, $R2_{Reg}$, $DR2_{Reg}$ refer. to regression radius (pos. / neg. R2) result stored in file TNC:\BLUM\E_588.TAB* *Q366 = 2 → Determination $L_{CC\ Nom}$, R, $R2_{Nom}$, $DR2_{Nom}$ refer. to nominal radius (pos. / neg. R2) result stored in file TNC:\BLUM\E_588.TAB*
Q367 optional	Axial offset (signed axial distance from the actual measured tool length to the circle centre R2) (only at Q366≠0)

Q388... Q399	The Q-parameters are used in cycle 588 as variables for temporary results and are overwritten.

Following parameters must be defined in tool table TOOL.T:

L, DL	Tool length, tool length correction (necessary at Q350=±1/±2/-3) *(referring to the circle centre)*
R, DR	Tool radius, tool radius correction (necessary at Q350=±1/±2/-3)
R2, DR2	Tool radius R2: angle radius / cutting edge radius / ball-ended radius
ROFFS	Radial measuring position for length measurement
LTOL	Wear tolerance: length (only at Q350=±1/±2)
RTOL	Wear tolerance: radius (only at Q350=±1/±2)
RBREAK	Wear tolerance: single measuring value on edge radius R2 (only at Q350=±1/±2, Q366≠0)

Example for the call of the measuring cycle in the CNC program:



T	1
L	+0.0
DL	+0.0
R	+15.0
DR	+0.0
R2	+6.0
DR2	+0.0
TL	
CUT	xxx
ROFFS	+8.5
LOFFS	xxx
LTOL	+0.02
RTOL	+0.02
LBREAK	xxx
RBREAK	+0.05

Explanation:

The tool with T-No.=3 is loaded into the spindle before call of cycle 588.

First the tool length is measured three times (Q361) 8.5 mm (ROFFS) outside the tool axis via multiple approaching cycle, and the scattering out of the individual measuring values concerning specified limits of 0.01 mm (Q362) is controlled.

Then the cutting edge radius R2 is determined by defined measuring points along the circular tool cutting edges. Condition is an accurate set value of R2 in the tool table TOOL.T. The first measuring point is determined by a measuring block in axial direction and is in a distance of 9.0 mm (Q358) from the tool axis at the starting angle 0° (Q363). The L/R coordinates of totally seven (Q365) measuring points on the circular cutting edge contour are determined one after the other with an angular step width of 20°. Each measuring point is mea-sured three times (Q361) and the scattering of the individual measuring values is controlled concerning the specified limits of 0.01 mm (Q362). The last measuring point is determined with a measuring block in radial direction and is above the maximum tool radius at the target angle 120° (Q364).

By means of a circle regression the L/R coordinates of the centre point and the cutting edge radius R2 are determined from the measuring points. The preliminary max. tool length L is corrected by the calculated centre point coordinate ΔL_{CC}, the max. tool radius R is calculated from the centre point coordinate ΔR_{CC} and the cutting edge radius R2. The additive length correction value of 0.0 mm (Q359) is added to the tool correction DL, the additive radius correction value of 0.0 mm (Q360) is added to the tool correction DR.

At **Q350=0/±3 (measure)** the measured tool geometries are written in memory L, R and R2 and the tool compensation set to DL=0, DR=0 und DR2=0 (DR=0 only if R>=R2, DR2=0 only if Q366=0).

At **Q350=±1 (check)** the theoretical tool geometries in memory L, R and R2 are not changed. The deviation between really measured actual tool geometry and theoretical tool geometry is calculated and written in memory DL, DR and DR2. Then the tool corrections DL, DR and DR2 are checked concerning the specified limits of wear LTOL, RTOL and RBREAK and tool status TL is set.

At **Q350=±2 (control)** the theoretical tool geometries in memory L, DL resp. R, DR resp. R2, DR2 are not changed. The deviation between really measured actual tool geometry and theoretical tool geometry is calculated and checked concerning specified limits of wear LTOL, RTOL and RBREAK and tool status TL is set.

When exceeding the wear tolerance LTOL, RTOL, RBREAK the measurement is not repeated, because the measuring results L,DL, R,DR, R2,DR2 don’t depend on single measuring values, but are calculated as mean value over all single measuring points. Thus reduces the cycle execution time to a minimum.

When exceeding the scattering tolerance Q362 the corresponding measurement is repeated. So, it is possible to avoid to the greatest possible extent unique sources of error or sources of error by chance, e.g. falling coolant drops or adhering chips, which could be responsible for faulty results. The measuring value is again determined under slightly changed environmental conditions, e.g. repeated blow-off of the tool, and the result is either revised or confirmed.

⇨ For faultless tool the error marker/flag is set to Q199=0 and M1205=0. The machining of the parts can be continued without interruption.

⇨ For faulty tool the error marker/flag is set to Q199=1 or Q199=2 and M1205=1 and the tool is locked in tool table TOOL.T. All further actions, e.g. machine stop or loading of a sister tool, must be decided in the main program.

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

<u>Following parameters in TOOL.T are set in the program flow:</u>

L, DL	Tool length referring to circle centre, tool length correction (only at Q350=0/±1/±3) *Q366 = 0* $\rightarrow L = L_{CC\ R2\text{-}Reg} = L_{Tip} - R2_{Reg},\ DL = L_{Act} - L_{Nom}$ *Q366 = 1* $\rightarrow L = L_{CC\ R2\text{-}Reg} = L_{Tip} - R2_{Reg},\ DL = L_{Act} - L_{Nom}$ *Q366 = 2* $\rightarrow L = L_{CC\ R2\text{-}Nom} = L_{Tip} - R2_{Nom},\ DL = L_{Act} - L_{Nom}$
R, DR	Tool radius, tool radius correction (only at Q350=0/±1/±3)
R2, DR2	Tool cutting edge radius (only at Q350=0/±1/±3) *Q366 = 0* $\rightarrow R2 = R2_{Reg}$, *DR2 = deviation of regression measurements* $R2_{RegAct} - R2_{RegNom}$ *Q366 = 1* $\rightarrow R2 = R2_{Reg}$, *DR2 = max. deviation of single measurement values* $R2_{Act} - R2_{RegNom}$ *Q366 = 2* $\rightarrow R2 = R2_{Nom}$, *DR2 = max. deviation of single measurement values* $R2_{Act} - R2_{Nom}$
TL	Tool locking (only at Q350=±1/±2)

<u>Caution:</u> The tool length is determined referring to the circle centre and not to the tool tip!

If the cutting edge geometry is not exactly circular, e.g. for tools with cutting edge wear, tools with several staggered circular cutting plates (toroidal milling cutter) or for tools with large radial run-out, a cutting edge radius R2 larger than the tool radius R can be calculated from the circle regression, so that the condition R >= R2 is not fulfilled (DR and DR2 are not taken into consideration at this comparison!).

In this case the result is automatically set in the tool table, e.g. acc. to following scheme:

- Tool radius R2 and DR2 from circle regression calculation are set in TOOL.T
- Modification of tool radius correction: $DR = \Delta R = R - R2 < 0$ e.g. $\Delta R = -0.050$ mm
- Modification of tool radius: $R = R2$

<u>Notice:</u> The resulting tool radius R+DR remains constantly by this correction and has the correct value for the radius correction RL/RR for path contours (e.g. L X+10 Y+30 RL F200 M3) !

If the tool is re-measured with cycle 584 and R2,DR2 are predefined in TOOL.T and R < R2 is determined, the result is automatically corrected in the tool table, e.g. acc. to following scheme:

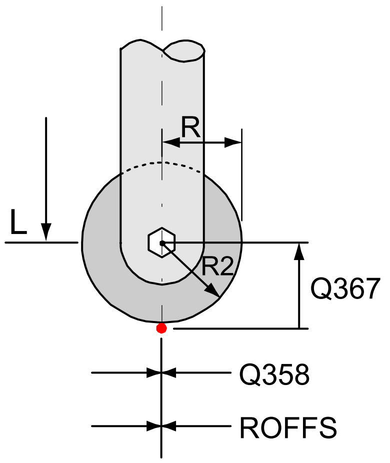
- Tool radius R and DR from cycle 584 are set in TOOL.T
- Modification of edge radius: $R2 = R = R2 + \Delta R$ with $\Delta R = R - R2 < 0$ e.g. $\Delta R = -0.050$ mm
- Modification of edge radius correction: $DR2 = DR2 + \Delta R$

<u>Notice:</u> The resulting edge radius R2 - DR2 remains constantly by this correction !

<u>Following Q-parameters are set in the program flow:</u>

Q199	Tool status parameter for tool breakage, tool wear and cutting edge control *Q199 = 0 → Tool dimension / cutting edge ok, continue work piece machining* *Q199 = 1 → Tool dimension outside the single wear tolerance (LTOL<DL<2*LTOL or RTOL<DR<2*RTOL or RBREAK<DR2<2*RBREAK)* *Q199 = 2 → Tool dimension outside the double wear tolerance, tool / cutting edge broken (DL>2*LTOL or DR>2*RTOL or DR2>2*RBREAK)*

Influence of the extended function Q366 to the measuring result, example:



Blum cycle call:

```
...
TOOL CALL 8 Z S3000 FMAX
TCH PROBE 588 TOOL SETTING L,R,R2
 Q350=0        ; MEASURING MODE
 Q361=3        ; NUMBER OF MEASUREMENTS
 Q362=0.01     ; SCATTERING TOLERANCE
 Q359=0        ; ADD. LENGTH COMPENSATION
 Q360=0        ; ADD. RADIUS COMPENSATION
 Q358=0        ; RADIAL OFFSET
 Q363=0        ; STARTING ANGLE
 Q364=90       ; TARGET ANGLE
 Q365=10       ; NUMBER OF MEASURING POINT
 Q366=2        ; EXTENDED FUNCTION
 Q367=-6       ; AXIAL OFFSET
...
```

Predefined parameter in TOOL.T:

L ; Nominal tool length: L_{Tip}=88,600mm
R ; Nominal tool radius: R=5.994mm
R2>0 ; Nominal edge radius: R2=6.0mm
ROFFS =0mm
LTOL ≈0.03...0.1mm
RTOL ≈0.03...0.1mm

Result at Q366=1:

At **Q366=1** the tool data L_{CC}, R, DR, R2, DR2 are determined and written to the tool table TOOL.T referring to the calculated regression edge radius $R2_{Reg}$:

Tool data in TOOL.T after the measurement:

T	L	DL	R	DR	R2	DR2	TL	CUT	ROFFS	LOFFS	LTOL	RTOL	LBREAK	RBREAK
8	82.558	0.0	6.042	-0.048	6.042	-0.008		xxx	0	Xxx	+0.03	+0.03	xxx	+0.05

The result of the single point measurement is additionally set in the result file TNC:\BLUM\E_588.TAB, e.g. acc. to following scheme: (R2=NOM-R2=$R2_{Reg}$, DR2=NOM-DR2):

NR	ANGLE	X-VALUE	Z-VALUE	ACT-R2	NOM-R2	NOM-DR2	REG-R2	REG-DR2
0	0.454	0.047	-6.039	6.039	6.042	-0.003	6.042	-0.003
1	15.342	1.600	-5.834	6.049	6.042	0.007	6.042	0.007
2	30.299	3.047	-5.215	6.041	6.042	-0.001	6.042	-0.001
3	45.320	4.290	-4.242	6.034	**6.042**	**-0.008**	6.042	-0.008
4	59.816	5.226	-3.040	6.046	6.042	0.004	6.042	0.004
5	74.722	5.832	-1.593	6.046	6.042	0.003	6.042	0.003
6	89.619	6.040	-0.040	6.040	6.042	-0.002	6.042	-0.002

Character explanation:

NR	Number of the single measuring point on the circle
ANGLE	Angle between measuring position and tool axis (0° = tool axis Z–)
X-VALUE	Radial measuring value of the single meas. point referring to $L_{CC\ Reg}$ / $R_{CC\ Reg}$
Z-VALUE	Axial measuring value of the single meas. point referring to $L_{CC\ Reg}$ / $R_{CC\ Reg}$
ACT-R2	Measured edge radius $R2_{Act}$ of the single meas. point referring to $L_{CC\ Reg}$ / $R_{CC\ Reg}$
NOM-R2	Calculated edge radius $R2_{Reg}$ (= actual value from circle regression calculation)
NOM-DR2	Deviation of the single meas. point $DR2_{Act}$ = actual value $R2_{Act}$ – regression value $R2_{Reg}$ $DR2_{Act} < 0$ → undersizing the nominal tool contour (smaller than nominal value) $DR2_{Act} > 0$ → oversizing the nominal tool contour (greater than nominal value)
REG-R2	Calculated edge radius $R2_{Reg}$ from the circle regression calculation
REG-DR2	Deviation of the single meas. point $DR2_{Reg}$ = actual value $R2_{Act}$ – regression value $R2_{Reg}$

Result at Q366=2

At **Q366=2** the tool data L_{CC}, R, DR, R2, DR2 are determined and written to the tool table TOOL.T referring to the predefined nominal edge radius $R2_{Nom}$ (result acc. to Heidenhain specification for R2, DR2)

Tool data in TOOL.T after the measurement:

T	L	DL	R	DR	R2	DR2	TL	CUT	ROFFS	LOFFS	LTOL	RTOL	LBREAK	RBREAK
8	82.600	0.0	6.0	-0.006	6.0	-0.026		xxx	0	Xxx	+0.03	+0.03	xxx	+0.05

The result of the single point measurement is additionally set in the result file TNC:\BLUM\E_588.TAB, e.g. acc. to following scheme: ($R2=NOM\text{-}R2=R2_{Soll}$, DR2=NOM-DR2):

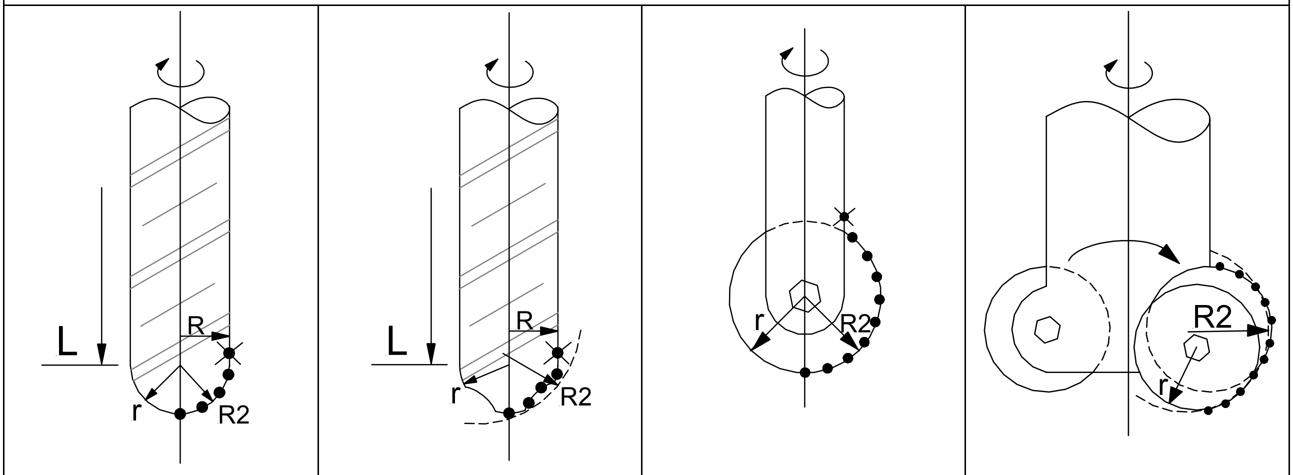
```
NR     ANGLE    X-VALUE    Z-VALUE     ACT-R2     NOM-R2    NOM-DR2     REG-R2    REG-DR2
 0     0.051      0.005     -5.996      5.996      6.000     -0.003      6.042     -0.003
 1    15.121      1.564     -5.789      5.997      6.000     -0.002      6.042      0.007
 2    30.282      3.017     -5.167      5.983      6.000     -0.016      6.042      0.000
 3    45.562      4.264     -4.182      5.973      6.000     -0.026      6.042     -0.008
 4    59.830      5.177     -3.009      5.989      6.000     -0.010      6.042      0.004
 5    74.944      5.788     -1.556      5.993      6.000     -0.006      6.042      0.003
 6    90.021      5.997      0.002      5.997      6.000     -0.002      6.042     -0.002
```

Character explanation:

NR	Number of the single measuring point on the circle
ANGLE	Angle between measuring position and tool axis (0° = tool axis Z–)
X-VALUE	Radial measuring value of the single meas. point referring to $L_{CC\ Nom}$ / $R_{CC\ Nom}$
Z-VALUE	Axial measuring value of the single meas. point referring to $L_{CC\ Nom}$ / $R_{CC\ Nom}$
ACT-R2	Measured edge radius $R2_{Act}$ of the single meas. point referring to $L_{CC\ Nom}$ / $R_{CC\ Nom}$
NOM-R2	Predefined nominal edge radius $R2_{Nom}$ (= nominal value from TOOL.T)
NOM-DR2	Deviation of the single meas. point $DR2_{Act}$ = actual value $R2_{Act}$ – regression value $R2_{Nom}$ $DR2_{Act} < 0$ → undersizing the nominal tool contour (smaller than nominal value) $DR2_{Act} > 0$ → oversizing the nominal tool contour (greater than nominal value)
REG-R2	Calculated edge radius $R2_{Reg}$ from the circle regression calculation
REG-DR2	Deviation of the single meas. point $DR2_{Reg}$ = actual value $R2_{Act}$ – regression value $R2_{Reg}$

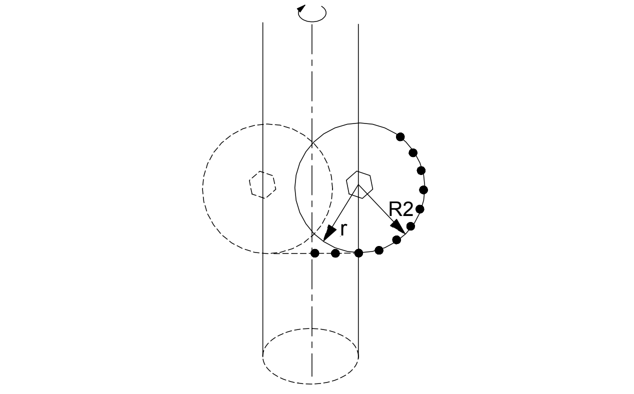
Influences to the edge radius accuracy R2

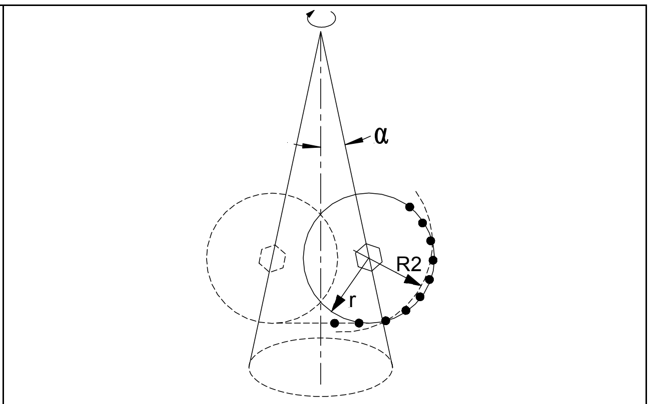
Measuring of tool radius



R2 ≅ r	R2 > r	R2 ≅ r	R2 > r

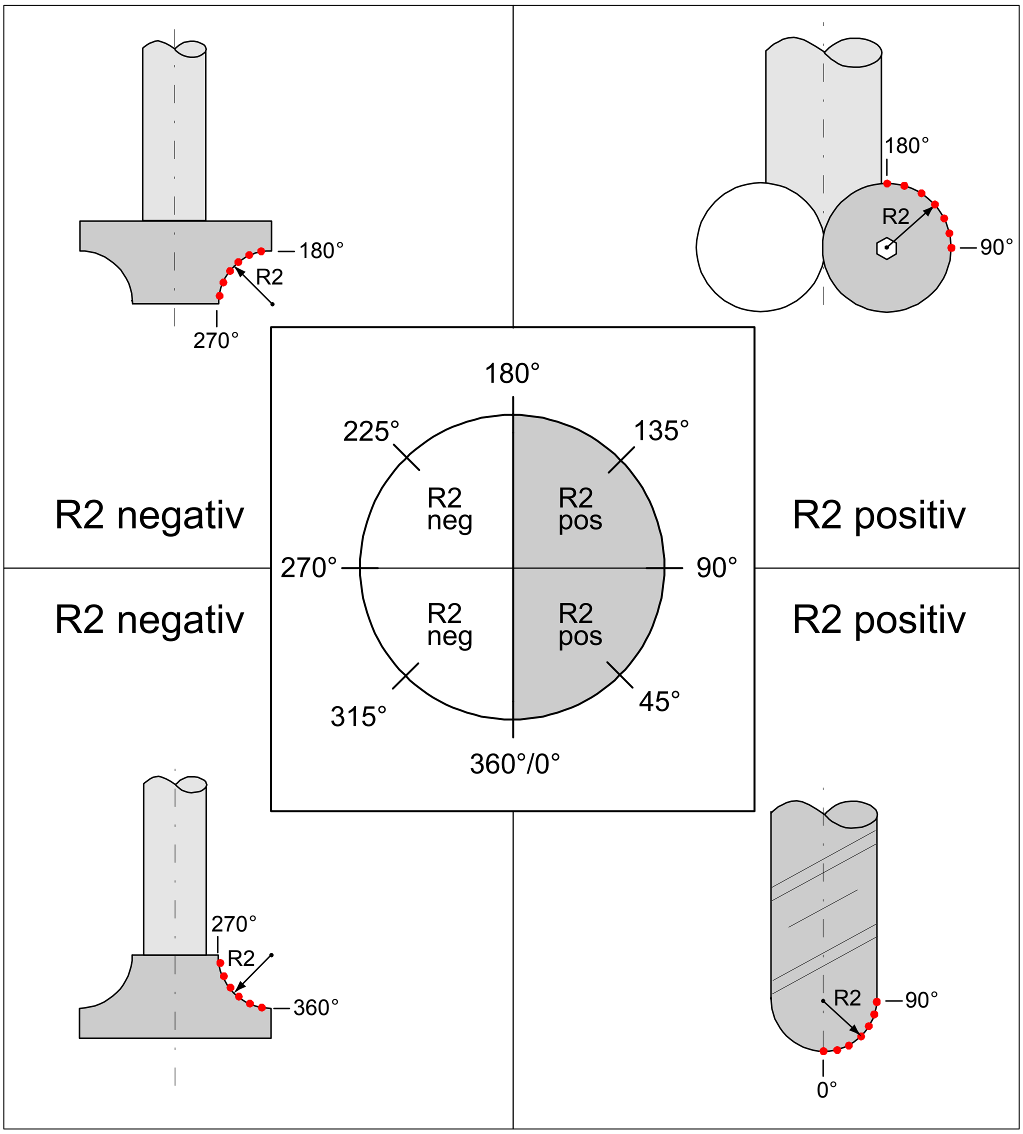
Influences to the edge radius accuracy R2



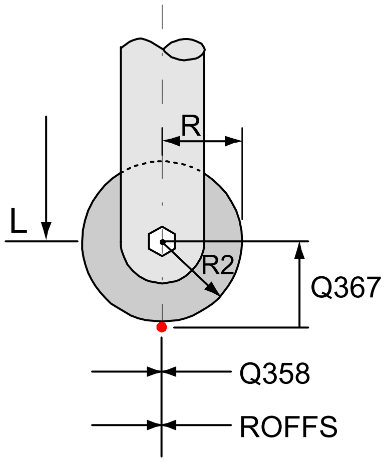


R2 ≅ r Eccentric tool clamping: tool axis describes a cylinder	**R2 > r** Sloped tool clamping: tool axis describes a cone

Setting of the edge radius R2, starting angle Q363, target angle Q364 depending on the tool geometry



Example for the call of the measuring cycle to determine the tool length referring to the circle centre

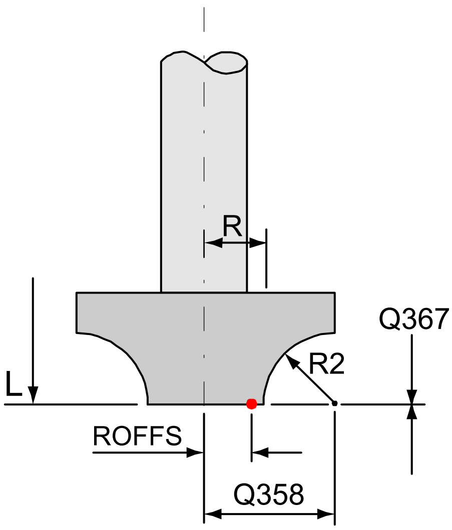


Blum cycle call:

```
...
TOOL CALL 8 Z S3000 FMAX
TCH PROBE 588 TOOL SETTING L,R,R2
 Q350=+3       ; MEASURING MODE
 Q361=3        ; NUMBER OF MEASUREMENTS
 Q362=0.01     ; SCATTERING TOLERANCE
 Q359=0        ; ADD. LENGTH COMPENSATION
 Q360=0        ; ADD. RADIUS COMPENSATION
 Q358=0        ; RADIAL OFFSET
 Q363=0        ; STARTING ANGLE
 Q364<=120     ; TARGET ANGLE
 Q365=10       ; NUMBER OF MEASURING POINT
 Q366=2        ; EXTENDED FUNCTION
 Q367=-R2      ; AXIAL OFFSET
...
```

Predefined parameter in TOOL.T:

L ; Nominal tool length (L_{CC})
R ; Nominal tool radius
R2>0 ; Nominal edge radius e.g. R2=+5.0mm
ROFFS =0mm
LTOL ≈0.03...0.1mm
RTOL ≈0.03...0.1mm

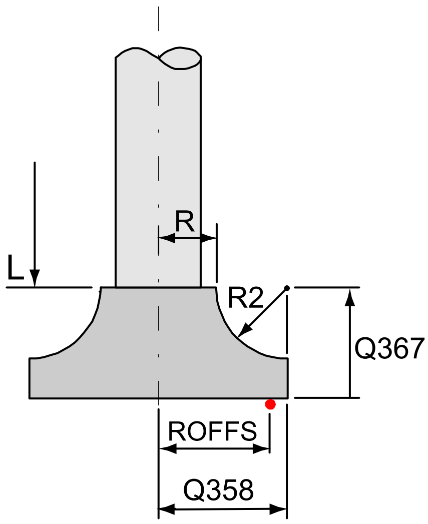


Blum cycle call:

```
...
TOOL CALL 8 Z S3000 FMAX
TCH PROBE 588 TOOL SETTING L,R,R2
 Q350=+3       ; MEASURING MODE
 Q361=3        ; NUMBER OF MEASUREMENTS
 Q362=0.01     ; SCATTERING TOLERANCE
 Q359=0        ; ADD. LENGTH COMPENSATION
 Q360=0        ; ADD. RADIUS COMPENSATION
 Q358=R+ABS(R2) ; RADIAL OFFSET
 Q363>=180     ; STARTING ANGLE
 Q364<=270     ; TARGET ANGLE
 Q365=10       ; NUMBER OF MEASURING POINT
 Q366=2        ; EXTENDED FUNCTION
 Q367=0        ; AXIAL OFFSET
...
```

Predefined parameter in TOOL.T:

L ; Nominal tool length (L_{CC})
R ; Nominal tool radius
R2<0 ; Nominal edge radius e.g. R2= -5.0mm
ROFFS =R-0.5mm
LTOL ≈0.03...0.1mm
RTOL ≈0.03...0.1mm



Blum cycle call:

```
...
TOOL CALL 8 Z S3000 FMAX
TCH PROBE 588 TOOL SETTING L,R,R2
 Q350=+3       ; MEASURING MODE
 Q361=3        ; NUMBER OF MEASUREMENTS
 Q362=0.01     ; SCATTERING TOLERANCE
 Q359=0        ; ADD. LENGTH COMPENSATION
 Q360=0        ; ADD. RADIUS COMPENSATION
 Q358=R+ABS(R2) ; RADIAL OFFSET
 Q363>=270     ; STARTING ANGLE
 Q364<=360     ; TARGET ANGLE
 Q365=10       ; NUMBER OF MEASURING POINT
 Q366=2        ; EXTENDED FUNCTION
 Q367=-ΔL      ; AXIAL OFFSET
...
```

Predefined parameter in TOOL.T:

L ; Nominal tool length (L_{CC})
R ; Nominal tool radius
R2<0 ; Nominal edge radius e.g. R2= -5.0mm
ROFFS =R+ABS(R2)-0.5mm
LTOL ≈0.03...0.1mm
RTOL ≈0.03...0.1mm