

NCT[®]

Controls for Milling Machines and Machining Centers

Probing and Digitizing Cycles

Manufactured by **NCT Automation kft.**

H1148 Budapest Fogarasi út 7

☒ Address: H1631 Bp. pf.: 26

☎ Phone: (+36 1) 467 63 00

☎ Fax: (+36 1) 363 6605

E-mail: nct@nct.hu

Home Page: www.nct.hu

Contents

1 Probing Cycles	<u>5</u>
1.1 Of Probing Cycles in General	<u>5</u>
1.2 Protected Positioning with Stylus	<u>7</u>
1.3 Calibrating Radial Offset of Stylus	<u>8</u>
1.3.1 General Considerations	<u>8</u>
1.3.2 Description of macro G102	<u>9</u>
1.4 Calibrating Stylus Ball Radius	<u>11</u>
1.4.1 General Considerations	<u>11</u>
1.5 Calibrating Stylus Length	<u>13</u>
1.5.1 General Considerations	<u>13</u>
1.5.2 Description of macro G104	<u>14</u>
1.6 Radial Measuring of Surface	<u>16</u>
1.7 Axial Measuring of Surface	<u>20</u>
1.8 Internal Corner Position	<u>23</u>
1.9 External Corner Position	<u>25</u>
1.10 Measuring Pocket / Web	<u>27</u>
1.11 Measuring Bore/ Boss	<u>31</u>
2 Automatic Tool Length Measurement	<u>35</u>
2.1 Installation, Applicability	<u>35</u>
2.2 Calibration of the Tool Length Probe	<u>36</u>
2.3 Measurement of the Tool Length	<u>37</u>
3 Digitizing Cycles	<u>39</u>
3.1 Of the Digitizing Cycles in General	<u>39</u>
3.2 Rectangular Digitizing Cycle	<u>40</u>

June 20, 2005

© Copyright NCT June 20, 2005

The Publisher reserves all rights for contents of this Manual. No reprinting, even in extracts, is permissible unless our written consent is obtained.

The text of this Manual has been compiled and checked with utmost care, yet we assume no liability for possible errors or spurious data and for consequential losses or damages.

1 Probing Cycles

1.1 Of Probing Cycles in General

Installation of probing cycles

The below discussed gauging macros and subprograms are from the software for probe system, which can be run on NCT milling machine controls. For these programs are in the operative memory of control, can be loaded and read from computers. When installing the probe system the following macros and subprograms need to be loaded to control:

O9010 (G101 PROTECTED POSITIONING)

O9011 (G102 CAL. RADIAL DATUM)

O9012 (G103 CAL. BALL RADIUS)

O9013 (G104 CAL. AXIAL DATUM)

O9014 (G105 RADIAL SURFACE)

O9015 (G106 AXIAL SURFACE)

O9016 (G107 INTERNAL CORNER)

O9017 (G108 EXTERNAL CORNER)

O9018 (G109 WEB/POCKET)

O9019 (G110 BORE/BOSS)

O9983 (INCH METRIC CONVERTER)

O9984 (PROBE STATUS)

O9985 (IN POSITION CHECK)

O9986 (TIMER)

O9987 (Z MEASURE)

O9988 (Y MEASURE)

O9989 (X MEASURE)

The first 10 macros from O9010 to O9019 can be called by the user, the detailed description and the method of usage of which are in the below chapters.

The programs from O9983 to O9989 are subprograms as well as macros called by the previous macros.

After loading the programs parameters G(9010) No. 9041...G(9019) No. 9050 must be set to 101...110, for the macros with the appropriate program number to be called by the appropriate command G.

If the printing service of macros is to be used the format indicated in the description of G105, G106, G109, G110 is given, provided parameter PRNT No. 9143 of MACRO parameter group is set to 1.

The security and other functions in connection with the stylus must be written into the PLC program of the given machine. In order to switch the stylus on and off by the centrifuge switch functions the following commands must be formed:

M59: function switch-on with orientation, and

M55: function stylus switch-off.

The stylus must be cabled to the control by applying the appropriate interface electronics.

The minimal requirement of the tool machine is that the spindle must be oriented. The probe

cycles cannot be used without spindle orientation.

Macro variable #510 must be set to 1, if the spindle cannot be indexed, otherwise to the code of the M function, as a result of which the spindle rotates with 180 deg.

Use of global variables when applying probe system

In order to execute the appropriate calculation the probe system use temporarily the following global variables not remained:

#100...#130

After the probe cycle have been executed the above variables can be used freely, further on there is no need for them.

Of the global macro variables retained upon power-off the following ones are used by the probe cycles:

#500...#510

These variables contain the calibration values of the stylus, the usage of these variables is from than on forbidden.

Internal clock #3001 is used in the probe cycles.

Inch - metric programming

The probe cycles can be run either in inch or metric mode without alterations to the program inputs. The calibrated values of the stylus are automatically converted from one measure system to the other. Be sure, that the entire calibration of the stylus occurs in the same measure system.

Stylus cycle

Stylus cycle O9987...O9989 execute a movement of three steps. As the first step it senses the surface to be measured at high rate, than if the stylus have sent a signal it stops. Afterwards it changes direction and leaves the surface at medium rate. Than it once more changes direction and measures the position of the surface accurately at a very low rate.

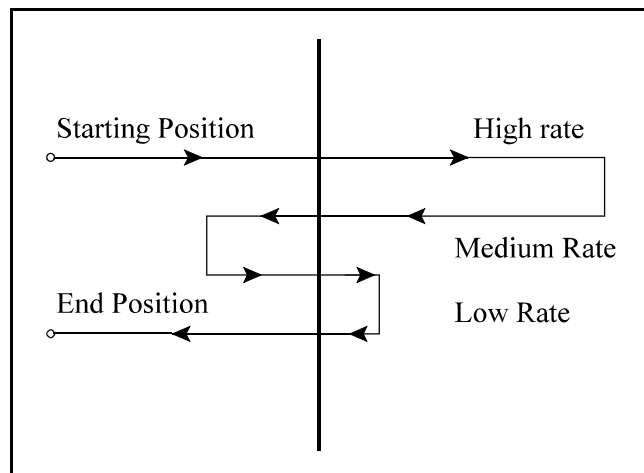


Fig. 1.1-1

THE DOCUMENTATION AND PROBE PROGRAMS ARE THE EXCLUSIVE PROPERTY OF NCT KFT, MAY NOT BE REPRODUCED, STORED OR TRANSMITTED, WHOLLY OR IN PART WITHOUT THE CONSENT OF NCT KFT. ALL CONSEQUENCES OF THIS ARE TAKEN BY THE ILLEGAL USER.

1.2 Protected Positioning with Stylus

With macro block

G101 X Y Z F

protected positioning can be executed, provided the stylus is in the spindle.

On coordinates

X, Y, Z target position values can be given at incremental or absolute values. At address

F the feed rate of probe travel can be entered. If no value is written at address F the stylus moves at 1500 mm/min, or 60 inch/min in the function of the applied measure system.

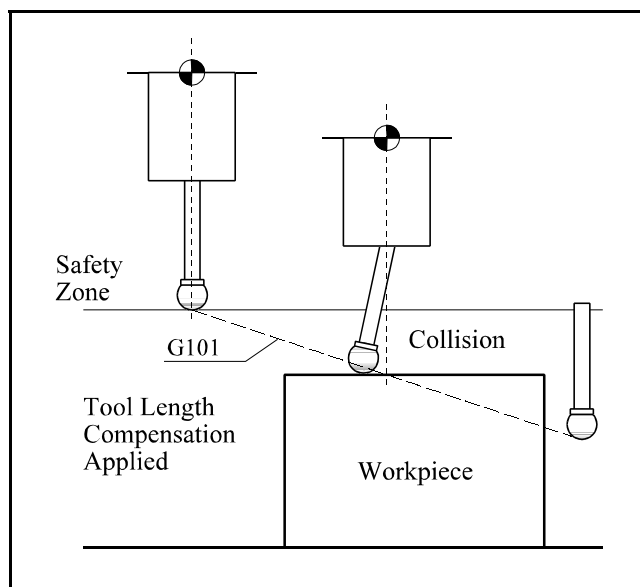


Fig. 1.2-1

Before the start of probe cycle the stylus is set to the desired position by means of this command. Before the movement is started the length offset of the stylus must be applied. The positioning macro examines, whether tool offset is applied, or not. If not it sends an error message. It also examines before the movement start, whether the stylus is in contact, or not. If yes, it displays an error message.

In the course of movement the macro is monitoring the stylus for any type of trigger. If the stylus collides on such obstruction it is stopped and the control sends an error message. If the macro fails to reach its target coordinates the macro sets a variable which causes subsequent gauging macros to fail with an OUT OF POSITION error.

Error messages appearing during the execution of macro G101:

NO TOOL LENGTH:	an attempt was made to use protected positioning without a tool length being applied
PATH OBSTRUCTED:	the stylus hit an obstruction in the course of movement
STYLUS IN CONTACT:	the stylus is in contact with the surface before the macro start
FALSE TRIGGER:	false trigger has been detected
OUT OF POSITION:	the macro fails to reach its target coordinates

Program number and name of macro:	O9010(G101 PROTECTED POSITIONING)
Parameter to be set:	G(9010) No. 9041=101
Subprograms called by the macro:	O9984(PROBE STATUS) O9985(IN POSITION CHECK) O9986(TIMER)

Level of macros called by the current macro: none.

Level of subprograms called by the current macro: 2.

1.3 Calibrating Radial Offset of Stylus

1.3.1 General Considerations

In order to measure accurately with the stylus the distance between the center of stylus ball and the center line of the spindle of the machine has to be known. This divergence is called the radial offset or excentricity of stylus. The excentricity established is taken into account by the different probe cycles in case of different lengths and positions. The macro calibrating the radial offset of stylus can be called by command G102. The calibration can be done in two ways.

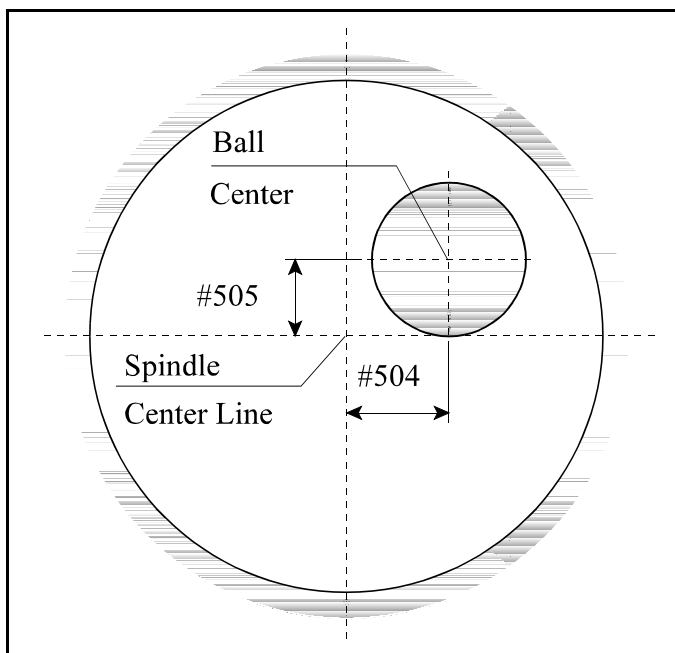


Fig. 1.3-1

Calibration if the spindle cannot be indexed

Macro variable #510 must be set to 0 on SETTING, #500-#599 screen. (#510=0)

Machine a bored hole on the machine, the diameter of which is larger than 20 mm. In the final phase of the machining a hole must be bored by means a single point tool to ensure that the bored hole centerline and spindle centerline are concentric. The datuming cycle must be carried out after the circular hole has been produced in order to avoid thermal movement of machine. The program automatically determines the offset value for the stylus ball relative to spindle.

Calibration if the spindle can be indexed

Macro variable #510 must be set to the value of M code, as the effect of which the spindle is turned away with 180 deg.

The setting can be done with a ring gauge of an accurately known size. The cycle establishes the center of the ring gauge in spindle oriented state, than turns away the spindle with 180 deg and measures it again. The radial offset of the stylus is calculated from the two measurings.

Radial calibration of the stylus must be executed in the following cases:

- when installing the stylus,
- after replacing the stylus or stylus ball,
- after breaking,
- for periodic controls,
- if the direction of spindle changes on the machine (e.g.: switch over from horizontal to vertical machining, vica versa).

The values measured are stored by the calibrating program in the following retained macro variables:

#504: radial offset (excentricity) of the stylus on the first axis of the selected plane (G17: X, G18: Z, G19: Y)

#505: radial offset (excentricity) of stylus on the second axis of the selected plane (G17: Y, G18: X, G19: Z)

1.3.2 Description of macro G102

Command

G102 D

starts the macro calibrating the radial offset of the stylus. At the end of cycle the stylus returns to its start position.

D the approximate diameter of bored hole or ring gauge. The exact value for the calibration of radial offset is not necessary.

It must be filled in, otherwise an error message is sent. The cycle adds 10 mm or 0,4 inch to the size of the programmed D and this value is regarded as limit. If no wall is found an error message is displayed.

In axial direction the length of stylus must be known with such accuracy, so that the stylus ball sinks entirely into the interior of bored hole or ring gauge. The macro executes the calibration in the plane selected by one of commands G17, G18, G19. The result of calibration appears at macro variables #504, #505.

Error messages when executing macro G102

FORMAT ERROR:	if address D is not filled in
NO TOOL LENGTH:	an attempt was made to use protected positioning without a tool length being applied
STYLUS IN CONTACT:	contact with the surface before the macro start
PROBE FAIL:	the expected surface was not contacted within the overtravel of probe
NO MOVEMENT:	no movement results from the programmed data
FALSE TRIGGER:	false trigger has been detected
OUT OF POSITION:	an attempt to run the cycle was made following an error in protected positioning cycle G101

Program number and name of macro: O9011(G102 CAL. RADIAL DATUM)

Parameter to be set: 9042 G(9011)=102

Subprograms called by the macro: O9984(PROBE STATUS)
O9985(IN POSITION CHECK)
O9986(TIMER)

Macros called by the macro: O9987(Z MEASURE)
O9988(Y MEASURE)
O9989(X MEASURE)

Level of macros called by the current macro: 1.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

If the spindle cannot be indexed:

```
%O0102 (CALIBRATING RADIAL OFFSET)
G54 G17 G90 G40 G0 X20 Y30      drilling and calibration in plane X-Y
T1                               Applying 1st drill
G43 Z10 H1
.....
Machining bored hole in position X20 Y30
.....
M19                               Spindle orientation
T98                               Applying stylus with reference to address T98
G43 Z100 H98                     Applying the tool offset of stylus at address
                                H98
G101 X20 Y30                     Protected positioning of stylus to the center
                                of bored hole
G101 Z-10                        Protected positioning of stylus into bored hole
G102 D25                         Calling calibrating macro
G0 Z100                          Retraction of stylus from bored hole
M30
%
```

If the spindle can be indexed

The zero position of coordinate system G54 must be set to the center of the ring gauge in plane Z-X with an accuracy of 2-3 mm and the zero position in direction Y must be set to the top of the ring gauge.

```
%O0102 (CALIBRATING RADIAL OFFSET)
G54 G18 G90 G40 G0 X0 Z0        Calibration in plane Z-X
M19                               Spindle orientation
T98                               Applying stylus with reference to address T98
G43 Y100 H98                     Applying the tool offset of stylus at address
                                H98
G101 Y-10                        Protected positioning of stylus into ring gauge
G102 D25                         Calling calibrating macro
G0 Y100                          Retraction of stylus from ring gauge
M30
%
```

1.4 Calibrating Stylus Ball Radius

1.4.1 General Considerations

For the sake of the accurate calibration the radius of the stylus ball has to be registered precisely in positive and negative directions on both radial-direction axes. The measuring must be done with the help of a ring gauge. The ring gauge for calibration should have an accurately known size and should be as large as is feasible, but more than 25 mm diameter in any case. The macro must be run after the calibration of the radial offset of the stylus, for this macro already takes the radial offset into account.

Before the calibration the probe stylus must be positioned to within ± 10 mm (± 0.4 inch) of the center of the ring gauge and at the correct height. In the course of calibration the macro first determines the center of the ring gauge, then starting from the center measures the effective size of the stylus to each direction against the known ring gauge diameters. The calibration is executed in the plane selected by one of commands G17, G18, G19.

Calibration of stylus ball radius must be executed in the following cases:

- when installing the stylus,
- after replacing the stylus or stylus ball,
- after breaking,
- for periodic controls,
- if the direction of spindle changes in the machine (e.g.: switch over from horizontal to vertical machining, vica versa).

The values measured are stored by the calibrating macro in the following retained global macro variables:

- #500: the stylus ball radius on the 1st axis of the selected plane in direction + (G17: X+, G18: Z+, G19: Y+)
- #501: the stylus ball radius on the 1st axis of the selected plane in direction - (G17: X-, G18: Z-, G19: Y-)
- #502: the stylus ball radius on the 2nd axis of the selected plane in direction + (G17: Y+, G18: X+, G19: Z+)
- #503: the stylus ball radius on the 2nd axis of the selected plane in direction - (G17: Y-, G18: X-, G19: Z-)

1.4.2 Description of macro G103

Command

G103 D

starts the macro calibrating the stylus ball radius. At the end of cycle the stylus returns to its start position.

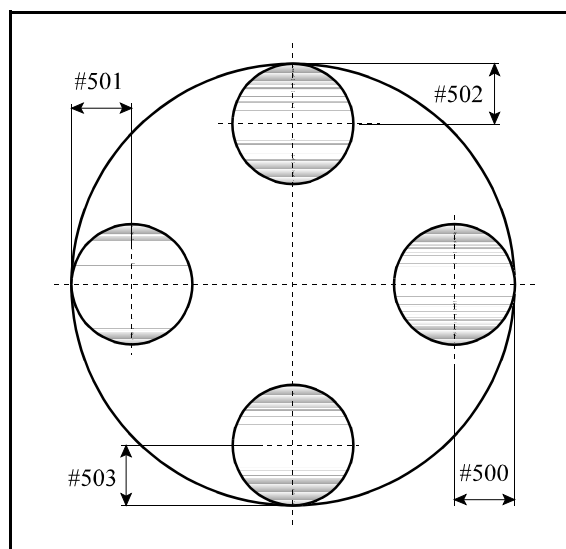


Fig. 1.4-1

D the approximate diameter of bored hole or ring gauge. The accurate value for the calibration of stylus ball radius is not necessary.

It must be filled in, otherwise an error message is sent. The cycle adds 10 mm or 0,4 inch to the size of the programmed D and the value is regarded as scanning limit. If no wall is found an error message is displayed.

In axial direction the length of stylus must be known with an accuracy, so that the stylus ball sinks entirely into the interior of bored hole or ring gauge. The macro executes the calibration in the plane selected by one of commands G17, G18, G19. The result of calibration appears at macro variables #500, #501, #502, #503.

Error messages when executing macro G103

FORMAT ERROR:	if address D is not filled in
NO TOOL LENGTH:	an attempt was made to use protected positioning without a tool length being applied
STYLUS IN CONTACT:	contact with the surface before the macro start
PROBE FAIL:	the expected surface was not contacted within the overtravel of probe
NO MOVEMENT:	no movement results from the programmed data
FALSE TRIGGER:	false trigger has been detected
OUT OF POSITION:	an attempt to run the cycle was made following an error in protected positioning cycle G101

Program number and name of macro: O9012(G103 CAL. BALL RADIUS)

Parameter to be set: 9043 G(9012)=103

Subprograms called by the macro: O9984(PROBE STATUS)
O9985(IN POSITION CHECK)
O9986(TIMER)

Macros called by the macro: O9987(Z MEASURE)
O9988(Y MEASURE)
O9989(X MEASURE)

Level of macros called by the current macro: 1.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

%O0103(CALIBRATING STYLUS BALL RADIUS)	
G54 G17 G90 G40 G0 X0 Z0	Calibration in plane X-Y
M19	Spindle orientation
T98	Applying stylus with reference to address T98
G43 Z100 H98	Applying the tool offset of stylus at address H98
G101 Z-10	Protected positioning of stylus into bored hole
G103 D50	Calling calibrating macro
G0 Z100	Retraction of stylus from bored hole
M30	
%	

1.5 Calibrating Stylus Length

1.5.1 General Considerations

Before the start of calibration the precise length of stylus must be established. First a tool offset register should be assigned to the probe and used whenever the probe is required. Then each time the stylus is used in the part program this length offset must be applied. When calibrating the length of stylus it should be acknowledged, whether the tool length offsets used in the given machine are equal to the part of tool outside the spindle, or contain the axial coordinate of work offset.

If the length offset is the part outside the spindle

The accurate location of a surface, e.g. the top of the ring gauge mentioned by macro G103 in the direction of the axis of the spindle must be determined. For the length of stylus can be defined with the ring contacted, it is important to measure the location of the surface to be contacted accurately. The location must be told the control on OFFSETS, WORK OFFSET MEASURE screen by choosing the appropriate coordinate system. Thereafter the appropriate length of probe stylus should be set manually with an accuracy of ± 5 mm and this is followed by the running of calibrating macro. The probe stylus then contacts the surface and the exact offset value is determined and will be automatically updated.

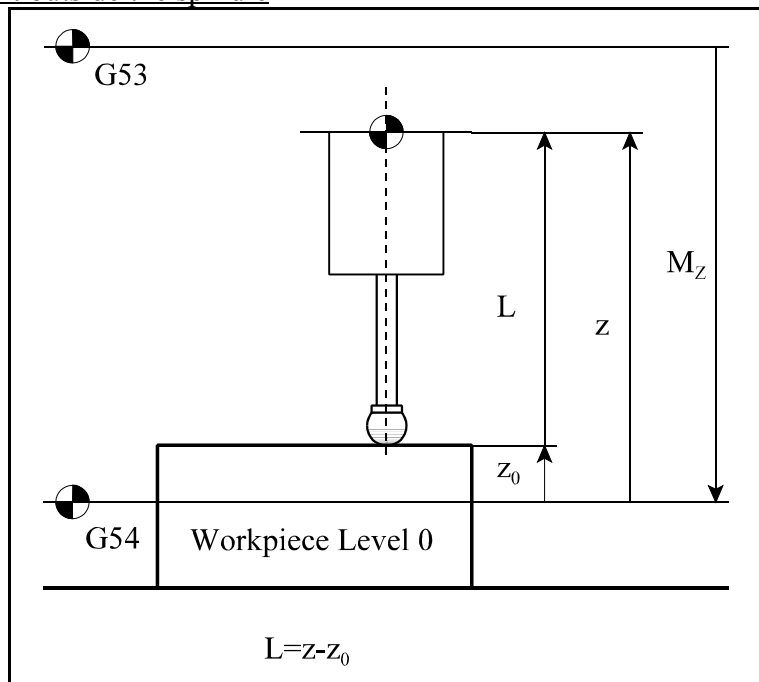


Fig. 1.5-1

If the length offset equals to the axial coordinate of work offset

The surface with known coordinates has to be contacted with the stylus on OFFSETS, TOOL LENGTH MEASURE screen and this value must be written to the selected coordinate for the control to calculate the length offset. Hereupon the calibrating macro can be run. Each time the tools are re-measured the length of stylus also has to be re-calibrated, naturally by referring to the surface with known size, to which the other tools have been measured.

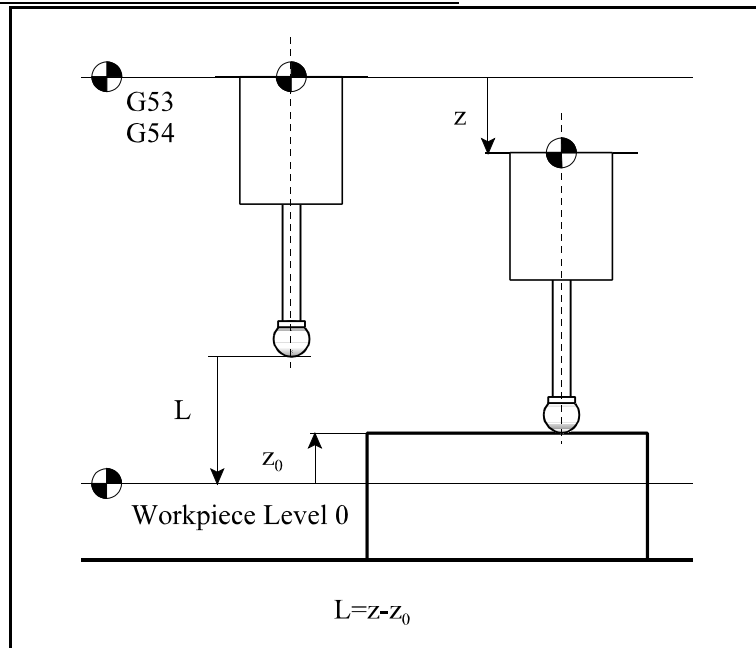


Fig. 1.5-2

The calibration is executed in the plane selected by one of commands G17, G18, G19.

The calibration of the length of stylus has to be done in the following cases:

- when installing the stylus,
- after replacing the stylus or stylus ball,
- after breaking,
- for periodic controls,
- if the direction of spindle changes in the machine (e.g.: switch over from horizontal to vertical machining, vica versa).

1.5.2 Description of macro G104

Command

G17

G104 Z T

or

G18

G104 Y T

or

G19

G104 X T

start the macro calibrating the length of stylus. At the end of cycle the stylus returns to its start position.

At address

Z, Y, X the precise position of the surface with known size is given, at address

T the offset number of the probe register, which is updated by the calibrating cycle.

If value has not been given at an address of an axis perpendicular to the selected plane or at T the control sends an error message. It also gives an error message if no surface is found 10 mm (0,4

inch) under the given level in the course of the cycle.

The result of calibration appears in the length offset register given at address T.

Error messages when executing macro G104

FORMAT ERROR:	if address D or address of the axis perpendicular to the selected plane is not filled in
NO TOOL LENGTH:	an attempt was made to use protected positioning without a tool length being applied
STYLUS IN CONTACT:	contact with the surface before the macro start
PROBE FAIL:	the expected surface was not contacted within the overtravel of probe
NO MOVEMENT:	no movement results from the programmed data
FALSE TRIGGER:	false trigger has been detected
OUT OF POSITION:	an attempt to run the cycle was made following an error in protected positioning cycle G101

Program number and name of macro: O9013(G104 CAL. AXIAL DATUM)

Parameter to be set: 9044 G(9013)=104

Subprograms called by the macro: O9984(PROBE STATUS)
O9985(IN POSITION CHECK)
O9986(TIMER)

Macros called by the macro: O9987(Z MEASURE)
O9988(Y MEASURE)
O9989(X MEASURE)

Level of macros called by the current macro: 1.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

%O0104(CALIBRATION OF STYLUS LENGTH)	
G54 G17 G90 G40 G0 X0 Y0	Calibration along axis Z
M19	Spindle orientation
T98	Applying stylus by referring to address T98
G43 Z100 H98	Acknowledging the length offset of stylus at address H98
G104 Z20 T98	Calling calibrating macro
M30	
%	

1.6 Radial Measuring of Surface

This macro can be applied for gauging the surface perpendicular to the selected plane, parallel to one of the axes of the plane and to the stylus centerline. The cycle can be used for:

- measuring the work coordinate system e.g. before starting the machining,
- determining the exact position of surface after machining.

With the measured data the following offsets can be modified:

- arbitrary work offset,
- tool radius offset.

The divergence between the nominal and current size:

- can be stored into an offset register and loaded by the operator,
- can be sent to computer with or without header bar,
- if tolerance limits are specified for the size deviation the cycle executes an inspection regarding the tolerance limits and sends a message to the operator should a limit be exceeded.

Command

G17

G105 X or Y Z R Q S or T M H U W

or

G18

G105 Z or X Y R Q S or T M H U W

or

G19

G105 Y or Z X R Q S or T M H U W

calls the macro serving for the radial gauging of surface. At the end of cycle the stylus returns to its start position.

X, Y, Z: One axis of the selected plane, on which the nominal value of surface is given. The size must be the arithmetic mean of the upper and lower tolerance limit if address U or H (tolerance limit inspection) is given. It is obligatory to give value to one of the axes of the selected plane.

Z, Y, X: axis of stylus. The filling in of this address indicates that before gauging the radial clearance of distance R from the given surface must be defined, than the stylus must be moved axially with the value at the axial address. If omitted, i.e. the stylus is not moved axially it is moved immediately to the given surface in radial direction

R: the clearance value from the surface to which the stylus ball is to be moved. A default value of 5 mm (0.2 inch) is used if no other value is programmed.

Q: the maximum probe stylus overtravel past the nominal surface. If omitted a value of 10 mm (0.4 inch) is assumed. If the expected surface was not contacted within the overtravel of probe an error message is sent.

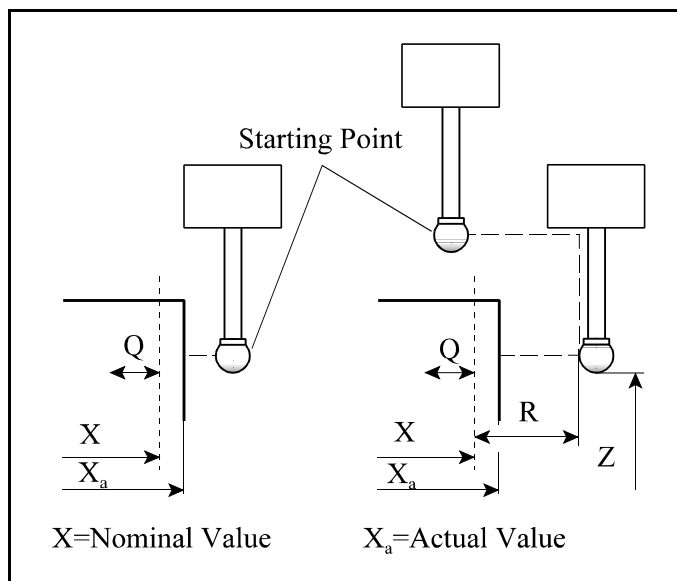


Fig. 1.6-1

- S: indicates the number of the work coordinate system the offset of which has to be modified after the measuring, so that it would coincide with the programmed size:
 S=0 common work coordinate system
 S=1 G54
 S=2 G55
 ...
 S=6 G59
 If filled in together with address T an error message is displayed, because it is senseless to modify tool radius offset and work offset at the same time.
- M: the number of an unused tool offset register to which the difference between nominal and the real size is stored should address M be filled in. The value in the selected register: is positive, provided the Metal condition of surface is on, is negative, provided the Metal condition of surface is off.
- T: the number of a tool offset used to cut the surface with the side of the cutter, the value of which is to be updated with the difference of the given and the measured value, provided the resulted surface was machined with that tool offset.
 If filled in together with address S an error message is displayed, because it is senseless to modify tool radius offset and work offset at the same time.
- H: the working tolerance of the surface position. The value H is half the total tolerance of the surface. E.g.: if the nominal position is 20.0 +0.1/-0.3 then a nominal of X19.9 H0.2 should be programmed. If the real size is out of tolerance an error message is sent.
- U: the upper tolerance limit. If the upper tolerance limit is defined and the deviation between the nominal size and the real size is out of this value neither the tool radius offset, nor the work offset is updated, but an error message is sent to the operator. This function can be used e.g. for tool break detection.
- W: W=1 and W=2 activate the data transfer function.
 If it is set to 1 only the data line,
 if it is set to 2 the header bar and the data line are printed.

Format of data output:

WORK OFFSET NO	NOMINAL POSITION	DEVIATION FROM NOMINAL
3	- 15.0000	.7810

Outputs of the probe cycle:

- T. The tool offset programmed is updated with the surface error,
- M. The value entered into the tool offset register is the Metal condition of surface. No sign means Metal On, while negative sign indicates Metal Off,
- S. A work offset (0-6) is updated to the measured surface,
- W. The measured positions are printed or stored in computer in this format,
- warning to the operator if the error exceeds the tolerance defined at address H,
- warning to the operator if the error exceeds the tolerance defined at address U.

Error messages of the cycle

FORMAT ERROR: if neither axis of the selected plane is given value,
If addresses S and T are filled in simultaneously,
If S is less than 0 or larger than 6,
If T or M is less than 6 or larger than 99,
NO TOOL LENGTH: an attempt was made to use protected positioning without a tool
length being applied
STYLUS IN CONTACT: contact with the surface before the macro start
PROBE FAIL: the expected surface was not contacted within the overtravel of
probe
NO MOVEMENT: no movement results from the programmed data
FALSE TRIGGER: false trigger has been detected
OUT OF POSITION: an attempt to run the cycle was made following an error in protected
positioning cycle G101

After the error message has been deleted the appropriate address of the program block must be modified.

Operator messages of the cycle:

OUT OF TOLERANCE: the error exceeds the tolerance specified at address H
UPPER TOL EXCEEDED: the error exceeds the maximum tolerance specified at address U
The program is continued due to START button.

Program number and name of macro: O9014(G105 RADIAL SURFACE)
Parameter to be set: 9045 G(9014)=105
Subprograms called by the macro: O9984(PROBE STATUS)
O9985(IN POSITION CHECK)
O9986(TIMER)
Macros called by the macro: O9010(G101 PROTECTED POSITIONING)
O9987(Z MEASURE)
O9988(Y MEASURE)
O9989(X MEASURE)

Levels of macros called by the current macro: 1.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

```
%O0105 (MILLING)
G54 G17 G90 G40...   Plane X-Y 1st work coordinate system
...                  Program blocks
...
M19                  Spindle orientation
T98                  Applying stylus by referring to address T98
G43 Z100 H98          Applying the length offset of stylus at address H98
G101 Z10              Protected positioning of stylus
G105 X50 Z-10 S1       Exact definition of G54 zero position in direction X
G0 Z100               Retraction of stylus
G41 D1                Machining
...
...
G40
M19                  Spindle orientation
T98                  Applying stylus by referring to address T98
G43 Z100 H98          Applying the length offset of stylus at address H98
G101 Z10              Protected positioning of stylus
G105 X50 Z-10 T1 H0.3 Measuring machined surface and modifying offset D1,
                        requesting error message, if the size is out of
                        tolerance
G0 Z100               Retraction of stylus
M30
%
```

1.7 Axial Measuring of Surface

This macro is capable of measuring the position of surface parallel to the selected plane, i.e. perpendicular to the centerline of stylus. The cycle can be applied for:

- measuring the work coordinate system e.g. before starting the machining,
- determining the exact position of surface after machining.

With the measured data the following offsets can be modified:

- arbitrary work offset,
- tool radius offset.

The difference between the nominal and current size:

- can be stored into an offset register and loaded by the operator,
- can be sent to computer with or without header bar,
- if tolerance limits are specified for the size deviation the cycle executes an inspection regarding the tolerance limits and sends a message to the operator should a limit be exceeded.

Command

G17

G106 Z Q S or T M H U W

or

G18

G106 Y Q S or T M H U W

or

G19

G106 X Q S or T M H U W

calls the macro serving for the axial measuring of surface. At the end of cycle the stylus returns to its start position.

Z, Y, X: address of the axis perpendicular to the plane selected by one of commands G17, G18, G19, on which the nominal size of the surface is given. The size must be the arithmetic mean of the upper and lower tolerance limits if address U or H (tolerance limit inspection) is given. It is obligatory to give value to the address of the axis perpendicular to the selected plane.

Q: the maximum probe stylus overtravel past the nominal surface. If omitted a value of 10 mm (0.4 inch) is assumed. If the expected surface was not contacted within the overtravel of probe an error message is sent.

S: indicates the number of the work coordinate system the offset of which has to be modified after the measuring, so that it would coincide with the programmed size:

S=0 common work coordinate system

S=1 G54

S=2 G55

...

S=6 G59

If filled in together with address T an error message is displayed, because it is senseless

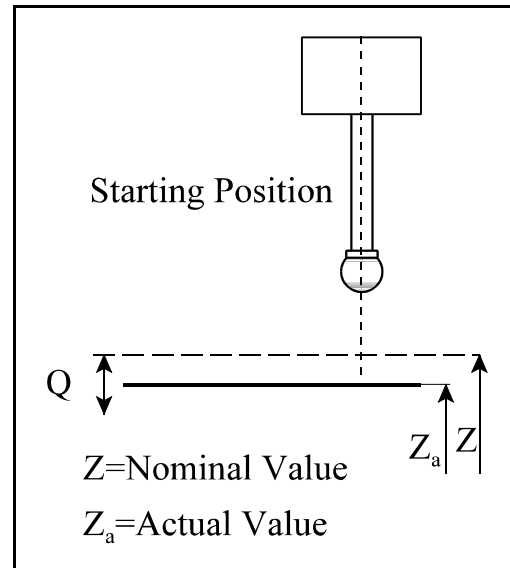


Fig. 1.7-1

- to modify tool radius offset and work offset at the same time.
- M: the number of an unused tool offset register to which the difference between the nominal and real size is stored should address M be filled in. The value in the selected register: is positive, provided the Metal condition of surface is on, is negative, provided the Metal condition of surface is off.
- T: the number of a tool offset used to cut the surface with the side of the cutter, the value of which is to be updated with the difference of the given and the measured value, provided the resulted surface was machined with that tool offset.
If filled in together with address S an error message is displayed, because it is senseless to modify tool radius offset and work offset at the same time.
- H: the working tolerance of the surface position. The value H is half the total tolerance of the surface. E.g.: if the nominal position $20.0 +0.1/-0.3$ then a nominal of $X19.9 H0.2$ should be programmed. If the real size is out of tolerance an error message is sent.
- U: the upper tolerance limit. If the upper tolerance limit is defined and the difference between the nominal size and the real size is out of this value neither the tool radius offset, nor the work offset is updated, but an error message is sent to the operator. This function can be used e.g. for tool break detection.
- W: W=1 and W=2 activate the data transfer function.
If it is set to 1 only the data line,
if it is set to 2 the header bar and the data line are transmitted.

Format of data output:

WORK	NOMINAL	DEVIATION
OFFSET NO	POSITION	FROM NOMINAL
5	- 150.000	.0530

Outputs of the probe cycle:

- T. The tool offset programmed is updated with the surface error,
- M. The value entered into the tool offset register is the Metal condition of surface. No sign means Metal On, while negative sign indicates Metal Off,
- S. A work offset (0-6) is updated to the measured surface,
- W. The measured positions are printed or stored in computer in this format,
- warning to the operator if the error exceeds the tolerance defined at address H,
- warning to the operator if the error exceeds the tolerance defined at address U.

Error messages of the cycle

FORMAT ERROR: if no value is given to the axis perpendicular to the selected plane,
If addresses S and T are filled in simultaneously,
If S is less than 0 or larger than 6,
If T or M is less than 6 or larger than 99,
NO TOOL LENGTH: an attempt was made to use protected positioning without a tool
length being applied
STYLUS IN CONTACT: contact with the surface before the macro start
PROBE FAIL: the expected surface was not contacted within the overtravel of
probe
NO MOVEMENT: no movement results from the programmed data
FALSE TRIGGER: false trigger has been detected
OUT OF POSITION: an attempt to run the cycle was made following an error in protected
positioning cycle G101

After the error message has been deleted the appropriate address of the program block has to be modified.

Operator messages of the cycle:

OUT OF TOLERANCE: the error exceeds the tolerance specified at address H
UPPER TOL EXCEEDED: the error exceeds the maximum tolerance specified at address U
The program is continued due to START button.

Program number and name of macro: O9015(G106 AXIAL SURFACE)
Parameter to be set: 9046 G(9015)=106
Subprograms called by the macro: O9984(PROBE STATUS)
O9985(IN POSITION CHECK)
O9986(TIMER)
Macros called by the macro: O9987(Z MEASURE)
O9988(Y MEASURE)
O9989(X MEASURE)

Level of macros called by the current macro: 1.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

```
%O0106 (MILLING)
G54 G18 G90 G40...   Plane Z-X 1st work coordinate system
...                  Program blocks
M19                  Spindle orientation
T98                  Applying stylus by referring to address T98
G43 Y100 H98          Applying length offset of stylus at address H98
G106 Y0 S1            Exact definition of G54 zero position in direction Y
T1                    Machining
G43 Y100 H1
...
M19                  Spindle orientation
T98                  Applying stylus by referring to address T98
G43 Y100 H98          Applying length offset of stylus at address H98
G106 Y0 T1 H0.3       Measuring the machined surface and modifying offset
                        H1, requesting error message if the size is out of
                        tolerance

M30
%
```

1.8 Internal Corner Position

This macro determines the position of internal corners made up of planes perpendicular to the selected plane and parallel to their axes. It adjusts the appropriate position of the given coordinate system to the position of the corner. The programmer positions the probe stylus to the required depth at a convenient point away from the expected surface. The cycle then automatically gauges the corner position. The cycle works in any plane selected by one of commands G17, G18, G19.

Command

G17
G107 X Y Q S

or

G18
G107 Z X Q S

or

G19
G107 Y Z Q S

calls the macro determining the internal corner position. At the end of cycle the stylus returns to its start position.

X, Y, Z: address of the axes of the plane selected by one of commands G17, G18, G19, on which the nominal coordinates of the corner is given. It is obligatory to give value to the axis addresses of the selected plane.

Q: the maximum probe stylus overtravel past the nominal surface. If omitted a value of 10 mm (0.4 inch) is assumed. If the expected surface was not contacted within the overtravel of probe an error message is sent.

S: indicates the number of the work coordinate system the offset of which has to be modified after the measuring, so that it would coincide with the programmed size:

S=0 common work coordinate system

S=1 G54

S=2 G55

...

S=6 G59

If address S is omitted the offsets of the current work coordinate system is adjusted to the corner position.

Output of the probe cycle:

- S. Adjusting the offset of work coordinate system to the measured corner, If S is not programmed the current work coordinate system is set.

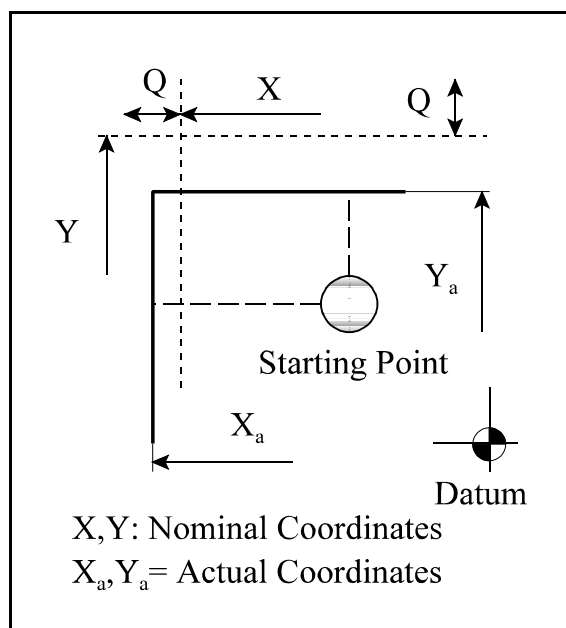


Fig. 1.8-1

Error messages of the cycle

FORMAT ERROR:	if one of the axes of the selected plane is not given a value, if S is less than 0 or larger than 6,
NO TOOL LENGTH:	an attempt was made to use protected positioning without a tool length being applied
STYLUS IN CONTACT:	contact with the surface before the macro start
PROBE FAIL:	the expected surface was not contacted within the overtravel of probe
NO MOVEMENT:	no movement results from the programmed data
FALSE TRIGGER:	false trigger has been detected
OUT OF POSITION:	an attempt to run the cycle was made following an error in protected positioning cycle G101

After the error message has been deleted the appropriate address of the program block has to be modified.

Program number and name of macro:	O9016(G107 INTERNAL CORNER)
Parameter to be set:	9047 G(9016)=107
Subprograms called by the macro:	O9984(PROBE STATUS) O9985(IN POSITION CHECK) O9986(TIMER)
Macros called by the macro:	O9010(G101 PROTECTED POSITIONING) O9015(G105 SINGLE SURFACE) O9987(Z MEASURE) O9988(Y MEASURE) O9989(X MEASURE)

Level of macros called by the current macro: 2.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

%O0107 (INTERNAL CORNER)	
G56 G17 G90 G40 X80 Y60	Plane X-Y 3 rd work coordinate system
M19	Spindle orientation
T98	Applying stylus by referring to address T98
G43 Z100 H98	Applying length offset of stylus at address H98
G101 Z-20	Protected positioning of stylus into corner
G107 X100 Y80 S3	Moving zero position of corner to coordinate X100 Y80 in coordinate system G56
G101 Z100	Retraction of stylus
...	Machining
...	
M30	
%	

1.9 External Corner Position

This macro determines the position of external corners made up of planes perpendicular to the selected plane and parallel to their axes. It adjusts the appropriate position of the given coordinate system to the position of the corner. The programmer positions the probe stylus to the required depth at a convenient point away from the expected surface. The cycle then automatically gauges the corner position. The cycle works in any plane selected by one of commands G17, G18, G19.

Command

G17

G108 X Y Q S

or

G18

G108 Z X Q S

or

G19

G108 Y Z Q S

calls the macro determining the external corner position. At the end of cycle the stylus returns to its start position

X, Y, Z: address of the axes of the plane selected by one of commands G17, G18, G19, on which the nominal coordinates of the corner is given. It is obligatory to give value to the axis addresses of the selected plane.

Q: the maximum probe stylus overtravel past the nominal surface. If omitted a value of 10 mm (0.4 inch) is assumed. If the expected surface was not contacted within the overtravel of probe an error message is sent.

S: indicates the number of the work coordinate system the offset of which has to be modified after the measuring, so that it would coincide with the programmed size:

S=0 common work coordinate system

S=1 G54

S=2 G55

...

S=6 G59

If address S is omitted the offsets of the current work coordinate system is adjusted to the corner position.

Output of the probe cycle:

- S. Adjusting the offset of work coordinate system to the measured corner, If S is not programmed the current work coordinate system is set.

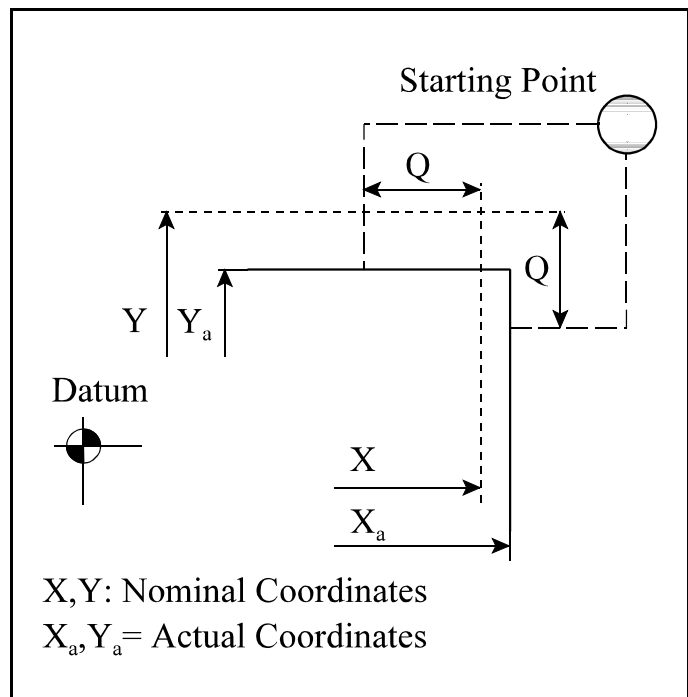


Fig. 1.9-1

Error messages of the cycle

FORMAT ERROR:	if one of the axes of the selected plane is not given a value, if S is less than 0 or larger than 6,
NO TOOL LENGTH:	an attempt was made to use protected positioning without a tool length being applied
STYLUS IN CONTACT:	contact with the surface before the macro start
PROBE FAIL:	the expected surface was not contacted within the overtravel of probe
NO MOVEMENT:	no movement results from the programmed data
FALSE TRIGGER:	false trigger has been detected
OUT OF POSITION:	an attempt to run the cycle was made following an error in protected posi- tioning cycle G101

After the error message has been deleted the appropriate address of the program block has to be modified.

Program number and name of macro:	O9017(G108 EXTERNAL CORNER)
Parameter to be set:	9048 G(9017)=108
Subprograms called by the macro:	O9984(PROBE STATUS) O9985(IN POSITION CHECK) O9986(TIMER)
Macros called by the macro:	O9010(G101 PROTECTED POSITIONING) O9015(G105 SINGLE SURFACE) O9987(Z MEASURE) O9988(Y MEASURE) O9989(X MEASURE)

Level of macros called by the current macro: 2.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

%O0108 (EXTERNAL CORNER)	
G57 G18 G90 G40 X50 Z60	Plane Z-X 4 th work coordinate system
M19	Spindle orientation
T98	Applying stylus by referring to address T98
G43 Y100 H98	Applying length offset of stylus at address H98
G101 Y-20	Protected positioning of stylus into corner
G108 X75 Z85 S4	Moving zero position of corner to coordinate X75 Z85 in coordinate system G57
G101 Y100	Retraction of stylus
...	Machining
...	
M30	
%	

1.10 Measuring Pocket / Web

This macro can be applied for measuring pocket or web parallel to one of the axes of the selected plane. The cycle can be used for:

- adjusting the work coordinate system to the center line of pocket or web for example before starting the machining,
- determining the width of pocket or web for example after machining.

With the measured data the following offsets can be modified:

- arbitrary work offset,
- tool radius offset.

The difference between the nominal and current size:

- can be stored into an offset register and loaded by the operator,
- can be sent to computer with or without header bar,
- if tolerance limits are specified for the size deviance the cycle executes an inspection regarding the tolerance limits and sends a message to the operator should a limit be exceeded.

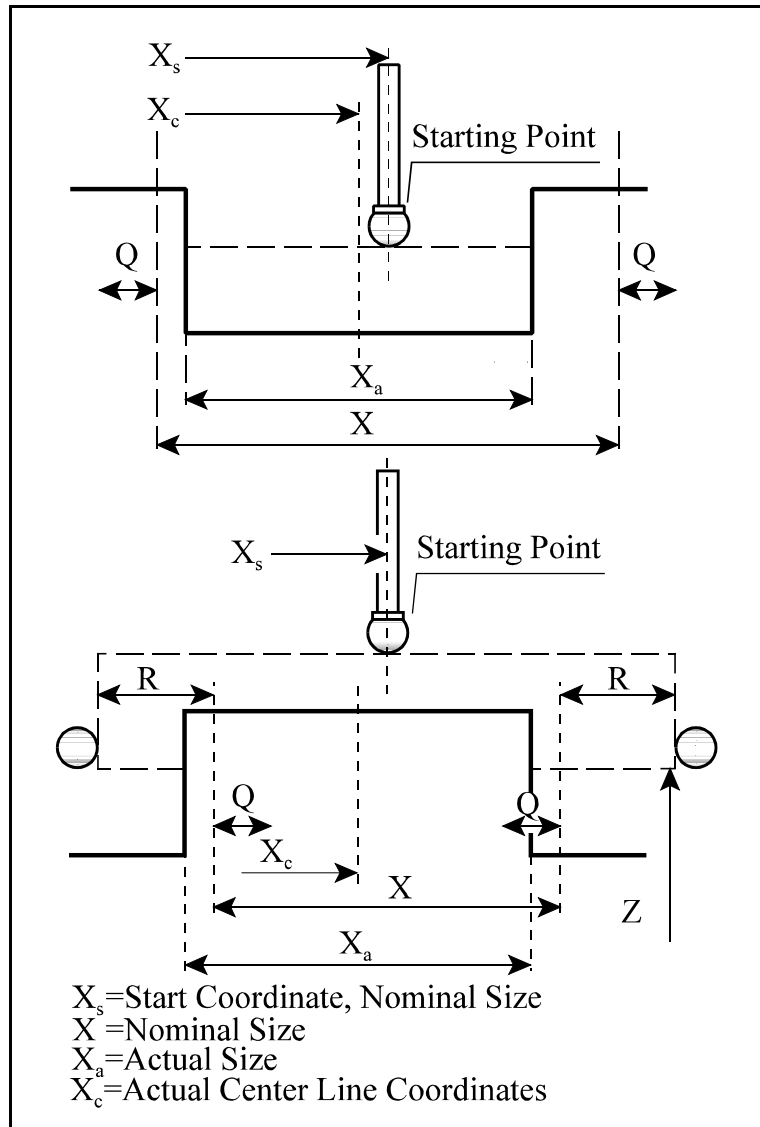


Fig. 1.10-1

Command

G17

G109 X or Y Z R Q S T M H U V W

or

G18

G109 Z or X Y R Q S T M H U V W

or

G19

G109 Y or Z X R Q S T M H U V W

calls the macro measuring pocket or web. At the end of cycle the stylus returns to its start position. In case work offset is programmed the zero position of the programmed axis is adjusted to the nominal position valid when loading the cycle.

X, Y, Z: address of one of the axes of the selected plane, on which the nominal width of the pocket or web is given. The size must be the arithmetic mean of the upper and lower tolerance limits if address U or H (tolerance limit inspection) is given. It is obligatory to

give value to one of the axes of the selected plane.

- Z, Y, X: centerline of stylus. The inclusion of a Z value in the macro indicates a web, otherwise a pocket will be assumed. The macro program should position the probe stylus over the expected center of web. The R value is radial clearance outside the expected width of the web. If omitted, a pocket is assumed and the probe stylus should be programmed to move the approximate center of the pocket to the depth required.
- R: the clearance value for web measurement. It is the additional distance outside the surface to which the stylus ball is to be positioned. If omitted a default value of 5 mm (0,2 inch) is applied but only when a Z value is included
- Q: the maximum probe stylus overtravel past the nominal surface. If omitted a value of 10 mm (0,4 inch) is assumed. If the expected surface was not contacted within the overtravel of probe an error message is sent.
- S: indicates the number of the work coordinate system the offset of which has to be modified after the measuring, so that it would coincide with the nominal position valid at the cycle start:
S=0 common work coordinate system
S=1 G54
S=2 G55
...
S=6 G59
If filled in together with address T an error message is displayed, because it is senseless to modify tool radius offset and work offset at the same time.
- M: the number of an unused tool offset register to which the difference between the nominal and the real width is stored should address M be filled in. The value in the selected register:
is positive, provided the Metal condition of surface is on,
is negative, provided the Metal condition of surface is off.
- T: the number of the tool offset to be updated. This is used on features where both sides were machined separately with the same tool. If a pocket is measured and the modified offset value does not have enough room in the pocket an error message is sent to the operator.
- H: the value of tolerance on the pocket or web width. The value H is half the total tolerance. E.g.: if the nominal width is 120.0 +0.1/-0.3 then a nominal of X119.9 H0.2 should be programmed. If the real size is out of tolerance an error message is sent.
- U: the upper tolerance limit. If the upper tolerance limit is defined and the difference between the nominal width and the real width is out of this value neither the tool radius offset, nor the work offset is updated, but an error message is sent to the operator. This function can be used e.g. for tool break detection.
- V: V=1 and V=2 activate the print function for dimensions.
If it is set to 1 only the data line,
if it is set to 2 the header bar and the data line are displayed
- W: W=1 and W=2 activate the print function for position.
If it is set to 1 only the data line,
if it is set to 2 the header bar and the data line are transmitted.

Format of data display:

TOOL	NOMINAL	TOLERANCE	DEVIATION
OFFSET NO	DIMENSION		FROM NOMINAL
D 3	30.0000	.2000	- .7700

WORK	NOMINAL	DEVIATION
OFFSET NO	POSITION	FROM NOMINAL
5	.0000	.0110

Outputs of the probe cycle:

- T. The tool offset programmed is updated with the surface error,
- M. The error size is stored into the tool offset register,
- S. Adjusting the offset of work coordinate system to the center line of pocket or web,
- V. Displaying data line of tool offset,
- W. The measured positions are printed or stored in computer in printed format,
- warning to the operator if the error exceeds the tolerance defined at address H,
- warning to the operator if the error exceeds the tolerance defined at address U.
- warning to the operator in case of pocket measuring if the pocket cannot be machined with the modified tool radius offset.

Error messages of the cycle

FORMAT ERROR: if neither axis of the selected plane is given value,
if S is less than 0 or larger than 6,
if T or M is less than 6 or larger than 99,

NO TOOL LENGTH: an attempt was made to use protected positioning without a tool length being applied

STYLUS IN CONTACT: contact with the surface before the macro start

PROBE FAIL: the expected surface was not contacted within the overtravel of probe

NO MOVEMENT: no movement results from the programmed data

FALSE TRIGGER: false trigger has been detected

OUT OF POSITION: an attempt to run the cycle was made following an error in protected positioning cycle G101

After the error message has been deleted the appropriate address of the program block has to be modified.

Operator messages of the cycle:

OUT OF TOLERANCE: the error exceeds the tolerance specified at address H

UPPER TOL EXCEEDED: the error exceeds the maximum tolerance specified at address U

TOOL DIA OFFS TOO LARGE: If the pocket cannot be machined with the resulted tool radius offset

The program is continued due to START button.

Program number and name of macro:	O9018(G109 WEB/POCKET)
Parameter to be set:	9049 G(9018)=108
Subprograms called by the macro:	O9984(PROBE STATUS)
	O9985(IN POSITION CHECK)
	O9986(TIMER)

Macros called by the macro: O9010(G101 PROTECTED POSITIONING)
 O9014(G105 SINGLE SURFACE)
 O9987(Z MEASURE)
 O9988(Y MEASURE)
 O9989(X MEASURE)

Level of macros called by the current macro: 2.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

```
%O0109(WEB)
G54 G17 G90 G40...   Plane X-Y 1st work coordinate system
...                 Program blocks
...
M19                 Spindle orientation
T98                 Applying stylus by referring to address T98
G43 Z100 H98         Applying length offset of stylus at address H98
G101 X120            Protected positioning to the nominal centerline of
                     web in direction X
G101 Z10             Protected positioning of stylus
G109 X80 Z-15 R35 S1 Adjusting the centerline of web to coordinate X120 in
                     coordinate system G54
G0 Z100             Retraction of stylus
G41 D1              Machining of web
...
...
G40
M19                 Spindle orientation
T98                 Applying stylus by referring to address T98
G43 Z100 H98         Applying the length offset of stylus at address H98
G101 X120            Protected positioning on the nominal centerline of web
in direction X
G101 Z10             Protected positioning of stylus
G109 X80 Z-10 T1 H0.3 Measuring the machined surface and modifying offset
                     D1, requesting error message if the size is out of
                     tolerance
G0 Z100             Retraction of stylus
M30
%
```

1.11 Measuring Bore/ Boss

This macro can be applied for measuring bore or boss parallel to one of the axes of the selected plane. The cycle can be used for:

- adjusting the work coordinate system to the center of bore or boss for example before starting the machining,
- determining the diameter of bore or boss for example after machining.

With the measured data the following offsets can be modified:

- arbitrary work offset,
- tool radius offset.

The difference between the nominal and current size:

- can be stored into an offset register and loaded by the operator,
- can be sent to computer with or without header,
- if tolerance limits are specified for the difference in size the cycle executes an inspection regarding the tolerance limits and sends a message to the operator should a limit be exceeded.

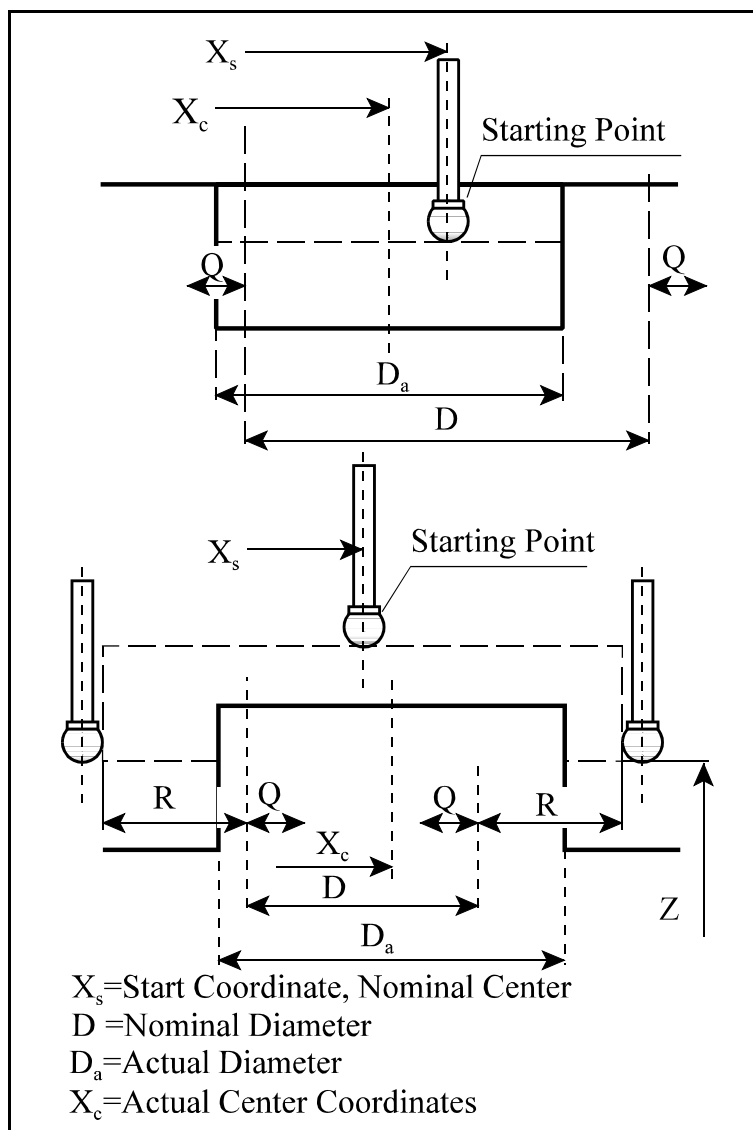


Fig. 1.11-1

Command

G17

G110 D Z R Q S T M H U V W

or

G18

G110 D Y R Q S T M H U V W

or

G19

G110 D X R Q S T M H U V W

calls the macro measuring bore or boss. At the end of cycle the stylus returns to its start position. In case work offset is programmed the zero position of the programmed axis is adjusted to the nominal center position valid when loading the cycle.

- D: Nominal diameter of bore or boss. The size must be the arithmetic mean of the upper and lower tolerance limit if address U or H (tolerance limit inspection) is given. It is obligatory to give value to address D.
- Z, Y, X: centerline of stylus. The inclusion of a Z value in the macro indicates a boss, otherwise a bore will be assumed. The macro program should position the probe stylus over the expected center of boss. The R value is radial clearance outside the expected diameter of the boss. If omitted, a bore is assumed and the probe stylus should be programmed to move the approximate center of the bore to the depth required.
- R: the clearance value for boss measurement. It is the additional distance outside the surface to which the stylus ball is to be positioned. If omitted a default value of 5 mm (0.2 inch) is assumed between the stylus ball and the boss but only when a Z value is included.
- Q: the maximum probe stylus overtravel past the nominal surface. If omitted a value of 10 mm (0.4 inch) is assumed. If the expected surface was not contacted within the overtravel of probe an error message is sent.
- S: indicates the number of the work coordinate system the offset of which has to be modified after the measuring, so that it would coincide with the nominal position valid at the cycle start:
S=0 common work coordinate system
S=1 G54
S=2 G55
...
S=6 G59
If filled in together with address T an error message is displayed, because it is senseless to modify tool radius offset and work offset at the same time.
- M: the number of an unused tool offset register to which the difference between the nominal and the real width is stored should address M be filled in. The value in the selected register:
is positive, the Metal condition of surface is on,
is negative, the Metal condition of surface is off.
- T: the number of a tool offset used to cut the surface with the side of the cutter, the value of which is to be updated with the difference of the given and the measured diameter value, provided the bore or boss was machined with that tool offset. If a bore is measured and the modified offset value does not have enough room in the bore an error message is sent to the operator.
- H: the working tolerance of the bore or boss diameter. The value H is half the total tolerance. E.g.: if the nominal diameter is 100.0 +0.1/-0.3 then a nominal of X99.9 H0.2 should be programmed. If the real size is out of tolerance an error message is sent.
- U: the upper tolerance limit. If the upper tolerance limit is defined and the difference between the nominal diameter and the real diameter is out of this value neither the tool radius offset, nor the work offset is updated, but an error message is sent to the operator. This function can be used e.g. for tool break detection.
- V: V=1 and V=2 activate the print function for dimensions.
If it is set to 1 only the data line,
if it is set to 2 the header bar and the data line are displayed
- W: W=1 and W=2 activate the print function for position.
If it is set to 1 only the data line,
if it is set to 2 the header bar and the data line are displayed.

Format of data display:

TOOL	NOMINAL	TOLERANCE	DEVIATION
OFFSET NO	DIMENSION		FROM NOMINAL
D 4	51.5700	.0200	- .0140

WORK	NOMINAL	DEVIATION
OFFSET NO	POSITION	FROM NOMINAL
6	X .0000	.0016
6	Y .0000	- .0006

Outputs of the probe cycle:

- T. The tool offset programmed is updated with the surface error,
- M. The error size is stored into the tool offset register,
- S. Adjusting the offset of work coordinate system to the center of bore or boss,
- V. Displaying data line of tool offset,
- W. The measured positions are printed or stored in computer in printed format,
- warning to the operator if the error exceeds the tolerance defined at address H,
- warning to the operator if the error exceeds the tolerance defined at address U.
- warning to the operator in case of bore measuring if the bore cannot be machined with the modified tool radius offset.

Error messages of the cycle

FORMAT ERROR:	if no value is given to address D, if S is less than 0 or larger than 6, if T or M is less than 6 or larger than 99,
NO TOOL LENGTH:	an attempt was made to use protected positioning without a tool length being applied
STYLUS IN CONTACT:	contact with the surface before the macro start
PROBE FAIL:	the expected surface was not contacted within the overtravel of probe
NO MOVEMENT:	no movement results from the programmed data
FALSE TRIGGER:	false trigger has been detected
OUT OF POSITION:	an attempt to run the cycle was made following an error in protected positioning cycle G101

After the error message has been deleted the appropriate address of the program block has to be modified.

Operator messages of the cycle:

OUT OF TOLERANCE:	the error exceeds the tolerance specified at address H
UPPER TOL EXCEEDED:	the error exceeds the maximum tolerance specified at address U
TOOL DIA OFFS TOO LARGE:	the bore cannot be machined with the resulted tool radius offset because the new value of T exceeds D in a bore

The program is continued due to START button.

Program number and name of macro:	O9019(G110 BORE/BOSS)
Parameter to be set:	9050 G(9019)=110
Subprograms called by the macro:	O9984(PROBE STATUS) O9985(IN POSITION CHECK) O9986(TIMER)

Macros called by the macro: O9010(G101 PROTECTED POSITIONING)
 O9014(G105 SINGLE SURFACE)
 O9987(Z MEASURE)
 O9988(Y MEASURE)
 O9989(X MEASURE)

Level of macros called by the current macro: 2.

Level of subprograms called by the current macro: 2.

Example for the use of the macro:

```
%O0110 (BORE)
G59 G18 G90 G40...   Plane Z-X 6th work coordinate system
...                 Program blocks
...
M19                 Spindle orientation
T98                 Applying stylus by referring to address T98
G43 Y100 H98         Applying length offset of stylus at address H98
G101 X120 Z200       Protected positioning on the nominal center of bore
                     in direction Z-X
G101 Y-10            Protected positioning of stylus
G110 D50 S6          Adjusting center of bore to coordinate X120 Z200 in
                     coordinate system G59
G0 Y100              Retraction of stylus
G41 D1               Machining of bore
...
...
G40
M19                 Spindle orientation
T98                 Applying stylus by referring to address T98
G43 Y100 H98         Applying length offset of stylus at address H98
G101 X120 Z200       Protected positioning on the nominal center of bore
                     in direction Z-X
G101 Y-10            Protected positioning of stylus
G110 D50 T1 H0.3     Measuring the machined surface and modifying offset
                     D1, requesting error message if the size is out of
                     tolerance
G0 Y100              Retraction of stylus
M30
%
```

2 Automatic Tool Length Measurement

2.1 Installation, Applicability

The 09920.NCT software has to be loaded in the memory of the NCT milling machine control. The software is stored in the memory of the control under the name 09920 TM_z. No further sub-routines are required for its operation. Since the software calls itself as a subroutine for a single level of depth, the program number must not be altered! Furthermore, the program may be used in automatic mode to maximum the second subroutine level!

Using a reference tool of known size, the software is capable of calibrating a tool probe, the axis of which is parallel to axis Z. Once the calibration is completed, it can be used for measuring the length and diameter of any arbitrary tool parallel to axis Z. The measured length compensations are applicable with G43 code. The software uses the following global macro-variables:

- #506 - distance of measuring device from the reference point in direction Z,
- #507 - distance of measuring device from the reference point in direction X,
- #508 - distance of measuring device from the reference point in direction Y,
- #509 - disk (wheel) diameter of the measuring device,
- #511 - default speed (r.p.m.),
- #512 - default feed,
- #513 - length of reference-standard tool.

Variables #506, #507, #508 and #513 are automatically given values at the time of calibration; variable #509 will be changed by the calibration in the knowledge of the diameter of the reference tool. The nominal disk (wheel) diameter of the measuring device is 40 mm with permissible deviations of max. 2 mm. The default values of variables #511 and #512 are 1500 r.p.m. and 750 r.p.m., respectively; they can be altered by the values of S and F specified at the time of calibration or measurement.

Signal TSIG of the measuring device and signal RDY have to be connected to interface input #1002 and input #1003, respectively. Unless the RDY signal is present (the device is connected), the software will return error message MEASURING DEVICE?. Unless the TSIG signal arrives from the measuring device prior to the measurement (the probe is being pressed continuously), the program will abort with error message DEPRESSED?.

Once the reference point has been assumed, first the calibration has to be carried out.

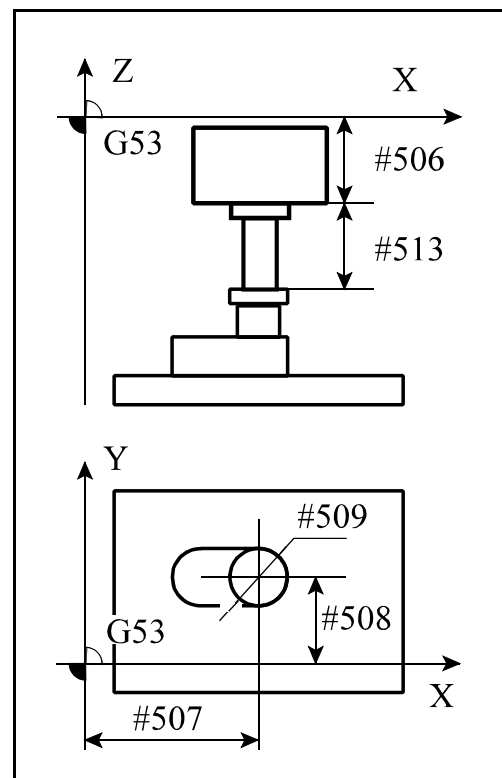


Fig. 2.1-1

2.2 Calibration of the Tool Length Probe

The probe has to be so positioned in the working space of the machine that the reference tool should be able to go around it.

Prior to the start of calibration, the reference tool has to be positioned over the centre of the measuring device, and the block G65 P9920 Zz Dd Hh T0 [Ee] [Ff] [Ss] has to be implemented.

The parameters have the following meanings:

- Zz - depth of immersion (measurement) (invariably a negative value),
- Dd - exact diameter of the reference tool,
- Hh - exact length of the reference tool (to be filled optionally; the default value is 0 mm),
- T0 - mode of calibration,
- Ee - safety clearance (to be filled optionally; the default value is 2 mm),
- Ff - feed (to be filled optionally; the default value is invariably #512),
- Ss - speed (to be filled optionally; the default value is invariably #511).

At the time of calibration the reference tool is moved by $(d+\#509)/2+e$ mm in direction +X and will sink z mm. Should it impact the measuring disk (wheel) during sinking, it will return to the start point with error message D TOO SMALL!. After the sinking, the reference tool moves toward the centre of the disk. If it fails to reach the disk, the reference tool returns to the start point with error message INCORRECT PROBING!. On completion of feeling, the reference tool goes up to the initial elevation and repeats the above operations in directions -X, +Y and -Y. Then it goes over the centre of the disk and measures position Z of the reference tool. At the end the reference tool moves to the centre of the measuring disk at the initial height.

In the course of calibration, variables #506, #507, #508 and #513 are set automatically; variable #509 is altered in the knowledge of the diameter of the reference tool.

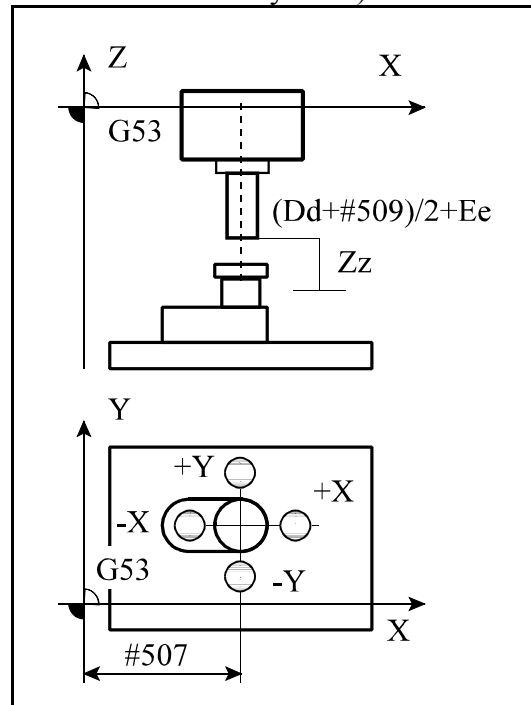


Fig. 2.2-1

2.3 Measurement of the Tool Length

The tool to be tested has to be positioned above the measuring device in direction Z. For the measurement, block G65 P9920 Zz [Kk] [Dd] Tt [Mm] [Ee] [Ff] [Ss] has to be carried out. The parameters have the following meanings:

- Zz - depth of immersion (measurement) (invariably a negative value),
- Kk - in a diameter measurement, the lift elevation (invariably a positive value; unless it is filled, only the tool length will be measured),
- Dd - estimated diameter of the tool under test (to be filled optionally; the default value is taken from the compensation register, the serial number of which is specified in parameter Tt),
- Tt - serial number of the tool-compensation register to be altered,
- Mm - tool rotation in measurement (to be filled optionally; the default condition is the stand-still of spindle; with 3 or 4 filled, the spindle will be rotating in a length and diameter measurement as well; with 13 or 14 filled, it will be rotating in a diameter measurement only),
- Ee - safety clearance (to be filled optionally; the default value is 2 mm),
- Ff - feed (to be filled optionally; the default value is invariably #512),
- Ss - speed (to be filled optionally; the default value is invariably #511).

During measurement, the tool is kept at distance $d/2$ from the disk centre in direction +X, and sinks from the actual height by the value specified in parameter z to measure the length of tool. Should the tool fail to reach the measuring device to depth z, it will return to the start point, and the software will return error message INCORRECT PROBING!. On completion of feeling, the tool ascends to height k over the point of feeling, moves $(d+\#509)/2+e$ mm in direction +X, and goes below the plane of feeling by k. Should the tool impact the measuring disk while sinking, it will return to the start point with error message D KICSI! (D TOO SMALL!). At the end of sinking the tool moves toward the centre of disk. Unless it reaches the disk, it will go back to the start point, and the software will return error message INCORRECT PROBING!).

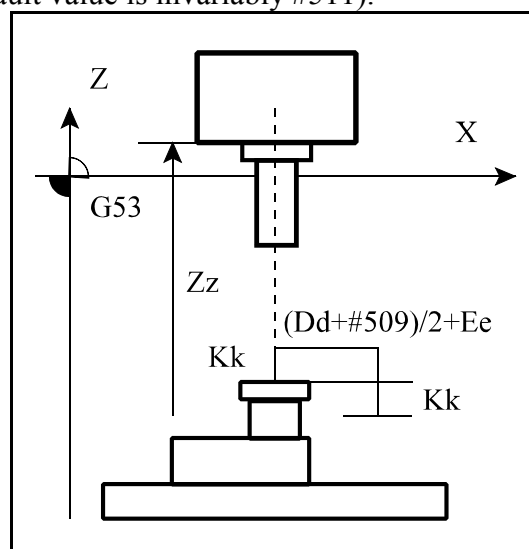


Fig. 2.3-1

Notes:

- In a length or diameter measurement the tool stands still unless address M is filled or its value is 5. Should 3 or 4 be filled at the address of M, the tool will be rotating both in length and in diameter measurements. If 13 or 14 is filled at address M, the spindle will stand still during the length measurement, but the tool will be rotating during the measurement of diameter. A length measurement with spindle turning is extremely dangerous because the tool may tear off the disk of the measuring device!
- The value of K has to be selected so that the tool sinks sufficiently at the time of diameter measurement. The minimum value of K is about 2 mm. If K is smaller than that, the tool will fail to touch the vertical side of the measuring disk. Unless address character K is filled, the diameter measurement will be omitted. (Its use may be justified in the case of

a drill or tap drill tool).

- When the diameter measurement is omitted (K is not filled), address D or E may also be 0 in order for the tool to touch the centre of the measuring disk.
- If the spindle is spinning during a diameter measurement, the error of the first feeling is theoretically smaller than 0.1 mm; the theoretical error of the diameter measurement is less than 1 μm . The error varies with the feed which, in turn, is computed by the program from the tool speed (if the spindle is spinning). A quicker measurement requires a higher r.p.m. to be specified.
- The software sets state G49 and may alter the feed and the speed.

THE TEST PROGRAM IS THE EXCLUSIVE PROPERTY OF NCT LTD. IT MAY NOT BE USED OR DISTRIBUTED UNLESS PERMITTED BY THE NCT KFT. ALL CONSEQUENCES THEREOF MUST BE BORNE BY THE ILLEGAL USER.

3 Digitizing Cycles

3.1 Of the Digitizing Cycles in General

Digitizing means the systematic measuring of positions of a free surface and the storing of the determined coordinates. A free surface is an arbitrary surface, even the one which cannot be expressed mathematically, as e.g. the wood pattern of a die-mold.

The measuring results, i.e. the surface position coordinates are transferred to the computer by the digitizing cycles in NC block format through serial transfer channel (RS-232C). The gathered measuring results are stored in the computer.

For to receive the results of digitizing cycles the appropriate program must be run on the computer. This may be a self-made program or one bought the stored data base can be sent back to the NCT control through DNC channel, for it is an NC program. If the format of the stored data base is a part program the task can be executed with this part program. The machined surface is congruent if the scanning is executed with a ball the diameter of which corresponds to the machining tool diameter.

If the digitizing is executed with a stylus ball the diameter of which differs from the machining tool diameter, the stored data base needs to be transformed.

3.2 Rectangular Digitizing Cycle

The rectangular digitizing cycle scans the free surface on the plane selected by one of commands G17, G18, G19. The cycle scans and digitizes this 3-D surface in a back-and-forth series of parallel line. Select an axis in the working plane. During digitizing the stylus moves in the positive direction of the selected axis, starting at the MIN point of working plane. When it reaches the range limit it moves by the line spacing in the positive direction of the other axis and then travels back along this line. At the other end of this line the stylus moves once again by the line spacing. This process is repeated until the entire rectangular has been scanned. Naturally the digitizing process can be started at any corner of the rectangular.

The limits of the surface to be digitalized must be specified so that the shape of the projection of the surface on the selected plane would be a rectangle.

The digitizing can be started by calling the macro with program number O9969. Before the beginning the start corner of rectangle to be digitalized must be positioned on the selected plane. The stylus ball should be in axial direction in safe distance from the surface. The digitizing is not continued axially under the given deepest position, but hereupon the scanning is done in radial direction. If it is not found till the given end position it finishes the given distance in side direction and goes on.

The macro can be called by means of the following block:

In case of G17 (plane X-Y):

G65 P9969 X Y Z **J** R Q F S:

J: scans in direction X goes in direction Y with J line spacing

or

G65 P9969 X Y Z **I** R Q F S:

I: scans in direction Y goes in direction X with I line spacing

where

X, Y: End positions of the surface to be digitalized (opposite corner to the rectangular start position)

Z: The deepest position of the surface to be digitalized in direction Z.

In case of G18 (plane Z-X):

G65 P9969 X Y Z **I** R Q F S:

I: scans in direction Z goes in direction X with J line spacing

or

G65 P9969 X Y Z **K** R Q F S:

K: scans in direction X goes in direction Z with K line spacing

where

X, Z: End positions of the surface to be digitalized (opposite corner to the rectangular start position)

Y: The deepest position of the surface to be digitalized in direction Y

In case of G19 (plane Y-Z):

G65 P9969 X Y Z **K** R Q F S:

K: scans in direction Y goes in direction Z with K line spacing

or

G65 P9969 X Y Z **J** R Q F S:

J: scans in direction Z goes in direction Y with J line spacing

where

Y, Z: End positions of the surface to be digitalized (opposite corner to the rectangular start position)

X: The deepest position of the surface to be digitalized in direction X.

Input data independent of plane selection:

R: Raster distance of scanning along the tangent of the interested curve. Its value must be less than half the maximum deflection of stylus.

Q: The axial retraction of stylus, when it scans the surface. E.g.: in case of scanning on horizontal plane it retracts R distance in horizontal direction and Q distance in vertical direction before searching for the next position.

F: Rate of scanning. Relation between the rate of scanning and the accuracy of measuring:
– in case of F12 the accuracy is 0.001 mm
– in case of F120 the accuracy is 0.01 mm
– in case of F1200 the accuracy is 0.1 mm

S: the program number at which the data sent to the computer are stored.

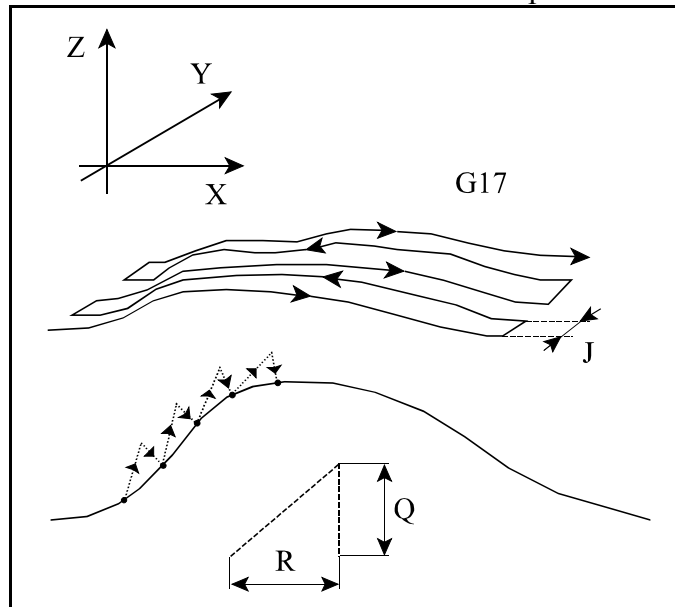


Fig. 3.2-1

Error messages of the cycle:

- FORMAT ERROR: data input error:
 - if of values I, J, K of the selected plane both are filled out,
 - if of addresses X, Y, Z, Q, R one is not filled in,
- NO TOOL LENGTH: no tool offset applied
- OUT OF POSITION: the set of sledges is out of position monitoring limit
- STYLUS IN CONTACT: Stylus is in contact with the surface before the movement start (It can occur in case of fix-distance scroll of lines.)
- SCANNING OBSTRUCTED: stylus stuck in a cavity, must be retracted manually.
- SCANNING FAIL: stylus have not found position after the third attempt.

Global variables used by the cycle:

#126

#150...#190

After the digitizing cycle has been executed the above variables can be freely used, there is no need for them further on.

Subprograms and macros used by the digitizing macro:

The rectangular digitizing cycle is a system of macros and subprograms, which can be run on NCT milling machine controls. For these programs are in the operative memory of the control, they can be loaded and read from the computer. When installing the digitizing system the following macros and subprograms need to be loaded into the control:

O9960 (RAPID WHEN SCANNING)

O9961 (PROBE FOR SCANNING)

O9962 (PRINT 1 FOR SCANNING)

O9963 (PROBE RETRACT)

O9964 (STEP)

O9965 (PRINT 2 FOR SCANNING)

O9966 (PROBE CHECK)

O9967 (AXIAL PROBE)

O9968 (RADIAL PROBE)

O9969 (RECTANGULAR SCANNING)

O9984 (PROBE STATUS)

O9985 (IN POSITION CHECK)

O9986 (TIMER)

Level of macros called by the current macro: 1.

Level of subprograms called by the current macro: 2.