User's Guide 11/2002 Edition

sinumerik

Measuring Cycles

SINUMERIK 840D/840Di/810D

SIEMENS

SIEMENS

SINUMERIK 840D/840Di/810D

Measuring Cycles

User's Guide

Valid for

Control	*Software version*
SINUMERIK 840D	6
SINUMERIK 840DE (export version)	6
SINUMERIK 840D powerline	6
SINUMERIK 840DE powerline	6
SINUMERIK 840Di	2
SINUMERIK 840DiE (export version)	2
SINUMERIK 810D	3
SINUMERIK 810DE (export version)	3
SINUMERIK 810D powerline	6
SINUMERIK 810DE powerline	6

11.02 Edition

Part 1: User's Guide

Part 2: Description of Functions

SINUMERIK® Documentation

Printing history

Brief details of this edition and previous editions are listed below.

The status of each edition is shown by the code in the "Remarks" column.

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised edition with new Order No.
C .... Revised edition with new status.
If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks
09.95	6FC5298-3AA01-0BP0	A
03.96	6FC5298-3AA70-0BP1	C
12.97	6FC5298-4AA70-0BP0	C
12.98	6FC5298-5AA70-0BP0	C
08.99	6FC5298-5AA70-0BP1	C
06.00	6FC5298-5AA70-0BP2	C
10.00	6FC5298-6AA70-0BP0	C
09.01	6FC5298-6AA70-0BP1	C
11.02	6FC5298-6AA70-0BP2	C

This manual is included in the documentation available on CD ROM (**DOCONCD**)

Edition	Order No.	Remarks
11.02	6FC5 298-6CA00-0BG3	C

Trademarks

Further information is available on the Internet under:
http:/www.ad.siemens.de/sinumerik

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

We have checked that the contents of this document correspond to the hardware and software described. Nonetheless, differences might exist and therefore we cannot guarantee that they are completely identical. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition. We welcome suggestions for improvement.

Subject to change without prior notice

Order No. 6FC5298-6AA70-0BP2
Printed in Germany

Siemens Aktiengesellschaft

Contents

Part 1: User's Guide

Measuring Cycles for Milling and Machining Centers 5-103

Measuring Cycles for Turning Machines 6-189

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Preface

Organization of documentation

The SINUMERIK documentation is organized on 3 different levels:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

Target group

This manual is aimed at machine tool users. It provides detailed information for operating the SINUMERIK 840D, 810D.

Standard scope

This Operator's Guide describes only the functionality of the standard scope.
A description of add-on features or modifications made by the machine builder are not included in this guide.

For more detailed information on SINUMERIK 840D, 810D publications and other publications covering all SINUMERIK controls (e.g. universal interface, measuring cycles...), please contact your local Siemens office.

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

Validity

This User's Guide is valid for the following controls:
SINUMERIK 810D, 840D, 840Di, MMC 100 and MMC 102/103.
Software versions stated in the User's Guide refer to the 840D and their 810D equivalent, e.g. SW 6 (840D) corresponds to SW 3 (810D).

SINUMERIK 840D powerline

From 09.2001

- SINUMERIK 840D powerline and
- SINUMERIK 840DE powerline

are available, with improved performance. A list of the available **powerline** modules can be found in the hardware description /PHD/ in Section 1.1

SINUMERIK 810D powerline

From 12.2001

- SINUMERIK 810D powerline and
- SINUMERIK 810DE powerline

are available, with improved performance. A list of the available **powerline** modules can be found in the hardware description /PHC/ in Section 1.1

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Hotline

Please address any questions to the following hotline:
A&D Technical Support Phone: ++49-(0)180-5050-222
Fax: ++49-(0)180-5050-223
Email: adsupport@siemens.com

If you have any questions (suggestions, corrections) concerning the documentation, please fax or e-mail them to the following address:
Fax: ++49-(0)0131-98-2176
Email: motioncontrol.docu@erlf.siemens.de
Fax form: See answer form at the end of the document.

Internet address

http://www.ad.siemens.de/sinumerik

Explanation of symbols

Procedure

Ordering option

Explanation

Function

Parameters

Programming example

Programming

Further notes

Cross-reference to other documentation, chapters, sections, or subsections

Notes and indication of danger

Additional notes or background information

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Warnings

The following warnings are used with graded severity.

Danger

Indicates an imminently hazardous situation which, if not avoided, **will** result in death or serious injury or in substantial property damage.

Warning

Indicates a potentially hazardous situation which, if not avoided, **could** result in death or serious injury or in substantial property damage.

Caution

Used with the safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in minor or moderate injury or in property damage.

Caution

Used without safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in property damage.

Notice

Used without the safety alert symbol indicates a potential situation which, if not avoided, **may** result in an undesirable result or state.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Basis

Your SIEMENS SINUMERIK 840D, 804Di, 810D is state of the art and is manufactured in accordance with recognized safety regulations, standards and specifications.

Add-on equipment

Using special add-on equipment and expanded configurations from SIEMENS, SIEMENS controls can be adapted to suit your specific application.

Personnel

Only **authorized and reliable personnel with the relevant training** must be allowed to handle the control. Nobody without the necessary training must be allowed to work on the control, not even for a short time.

The **responsibilities** of the personnel employed for setting, operating and maintenance must be clearly **defined** and **supervised**.

Behavior

Before the control is started up, it must be ensured that the Operator's Guide has been read and understood by the personnel responsible. The operating company is also responsible for **constantly monitoring** the overall technical state of the control (faults and damage apparent from the outside and changes in response).

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Service
Repairs must only be carried out in accordance with the information given in the Service and Maintenance Guide by **personnel trained and qualified** in the relevant field. The relevant safety regulations must be observed.

Note
The following is **contrary to the intended purpose** and **exonerates the manufacturer from any liability**:

Any use whatsoever beyond or deviating from the application stated in the above points.

If the control is **not in perfect technical condition**, or is operated without awareness for safety or the dangers involved or without observing the instructions given in the instruction manual.

If faults that can reduce safety are not remedied **before** the control is started up.

Any **modification, overriding** or **deactivation** of equipment on the control used for the perfect functioning, unrestricted use or active and passive safety.

This can result in **unforeseen dangers** for:
- the health and life of people,
- the control, machine and other property of the operating company and user.

Introduction

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.1 Basics

Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be suitably adapted to the problem at hand by means of parameter settings.

With regard to measurement applications, a distinction must generally be made between **tool measurement** and **workpiece measurement.**

Workpiece measurement

For workpiece measurement, a measuring probe is moved up to the clamped workpiece in the same way as a tool. The flexibility of the measuring cycles makes it possible to perform nearly all measurements which may need to be taken on a milling machine.
An automatic tool offset or an additive ZO can be applied to the result of the tool measurement.
The measurement variants which can be implemented with the measuring cycles available in this configuration are described on the following pages.

Tool measurement

To perform tool measurement, the changed tool, which in the case of a lathe is usually located in the turret, is moved up to the probe which is either permanently fixed or swiveled into the working range. The automatically derived tool geometry is entered in the relevant tool offset data record.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.2 General preconditions

Certain preconditions need to be fulfilled before measuring cycles can be used.

These conditions are described in greater detail in Part 2 Description of Functions (from Chapter 8 onwards).

The following checklist is useful in determining whether all such preconditions are fulfilled:

Machine

- All machine axes are designed in accordance with DIN 66217

Availability of cycles

- The data blocks:
  GUD5.DEF and
  GUD6.DEF
  have been loaded into the control ("Definitions" directory in file system) and
- the measuring cycles have been loaded into the standard cycle directory of the control followed by a power ON operation.

Initial position

- The reference points have been approached.
- All axes are positioned prior to the cycle call in such a way that the setpoint position can be approached without a change in direction.
- The start position can be reached without collisions by means of linear interpolation.

Displaying measuring result screens
It is only possible to display measurement result screens with an MMC/PCU.

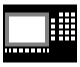
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

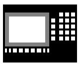
Programming

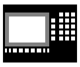
- The inch/metric units system selected in the machine data for the basic setting is active.
- The milling radius compensation and the programmable frame are deselected prior to the cycle call.
- All parameters for the cycle call have been defined beforehand.
- The cycle is called no later than at the 5th program level.
- Neither of the operating modes "Block search" or "Dry run" is active since these are automatically skipped by the measuring cycles.
- The specified default setting of the supplied data blocks is required to ensure that all example programs run correctly.
- With measuring cycles SW 4.4 and higher, measurement in a programmed measurement system that differs from the basic system is possible, i.e. in a metric basic system with active G70 and in an inch basic system with active G71.
- With measuring cycles SW 4.4 and higher, measurement in a programmed measurement system that differs from the basic system is possible with technology data switched over. This means in a metric basic system with active G700 and in an inch basic system with active G710.

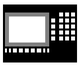
Software status ID

In the delivery status of the measuring cycles, the current software status of the control is entered in parameter _SI[1] in the GUD6 block, i. e. 5 for SW 5. This parameter must be changed to match the measuring cycles to older software releases.
Example:
When using measuring cycles status 5.x.x on a control with SW 4, → _SI[1] = 4
Precondition:
In order to use the measuring cycles, the software status of the control must be ≥ 3.


840 D
NCU 571


840 D
NCU 572
NCU 573


810 D

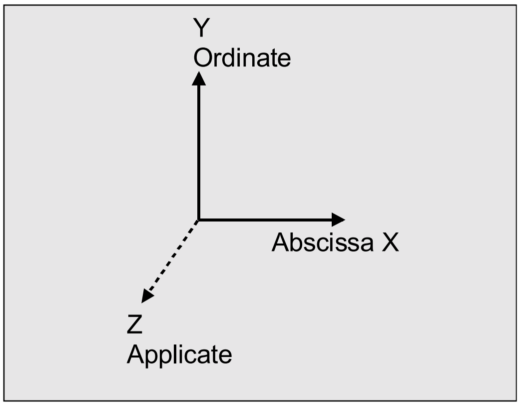

840Di

1.3 Plane definition

Tool radius compensation planes G17, G18 or G19 can be selected. Lengths 1, 2 and 3 are assigned as follows to the axes depending on the tool type used:

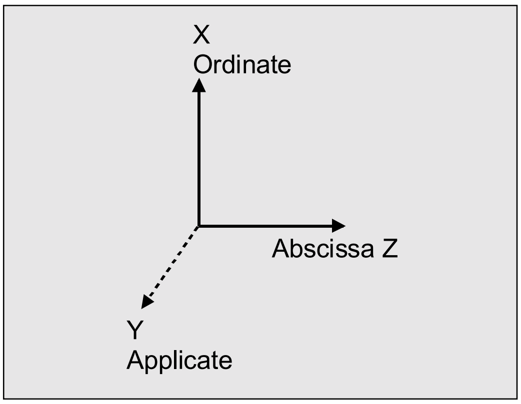
G17 plane

Tool type	100	
Length 1		applies to Z
Length 2		applies to Y
Length 3		applies to X



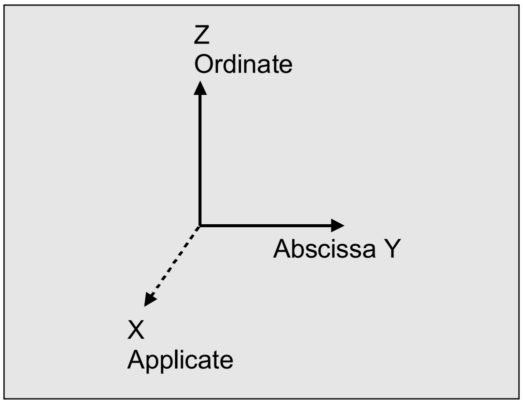
G18 plane

Tool type	100	
Length 1		applies to Y
Length 2		applies to X
Length 3		applies to Z



G19 plane

Tool type	100	
Length 1		applies to X
Length 2		applies to Z
Length 3		applies to Y



840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

1.4 Suitable probes

Function

In order to measure tool and workpiece dimensions, a touch-trigger probe is required that supplies a constant signal (rather than a pulse) when deflected.

The probe must be capable of virtually bounce-free switching. This is normally achieved by adjusting the probe mechanically.

The probe type is defined in the measuring cycles in a parameter.

Various types of probes made by different manufacturers are available on the market. Probes are classified in three groups according to the number of directions in which they can be deflected.

Classification of probe types

Probe type	Turning machines		Milling mach. and mach. centers
	Tool measurement	**Workpiece measurement**	**Workpiece measurement**
Multidirectional	X	X	X
Bidirectional	-	X	X
Monodirectional	-	-	X

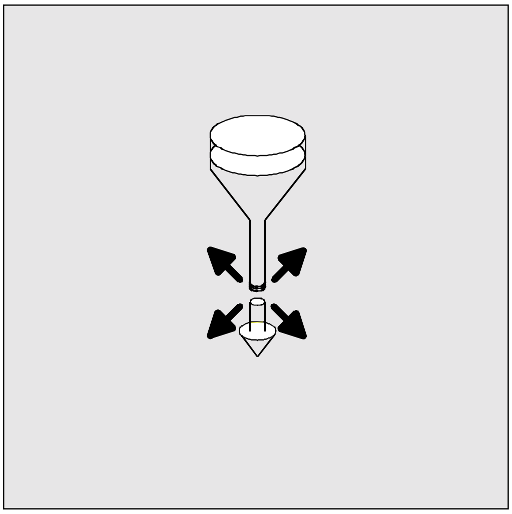
While a bidirectional probe can be used for turning machines, with milling machines and machining centers it is also possible to use a mono probe for workpiece measuring.

The probe is defined in the measuring cycles in a parameter.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Multidirectional probe (3D)

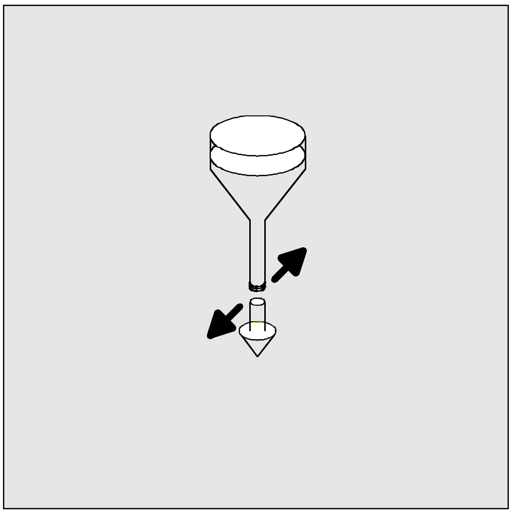
With this type, measuring cycles for workpiece measurement can be used without limitation.



Bidirectional probe

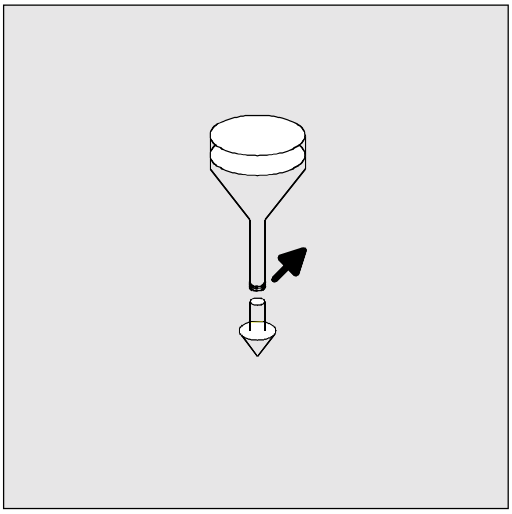
This probe type is used for workpiece measurement on milling machines and machining centers.

This probe type is treated in the same way as a monodirectional probe for workpiece measurement on milling machines and machining centers.



Monodirectional probe

This probe type can only be used for workpiece measurement on milling machines and machining centers with slight limitations; reference is made to this in the cycles concerned.
In order to be able to use this type of probe on milling machines and machining centers, it must be possible to position the spindle with the NC function SPOS and to transmit the switching signal of the probe through 360° to the receiving station (at the machine column).



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

The probe must be mechanically aligned in the spindle in such a way that measurements can be taken in the following directions at the 0 degree position of the spindle.

X-Y plane G17	positive X direction
Z-X plane G18	positive Z direction
Y-Z plane G19	positive Y direction

The measurement will take longer when using a monodirectional probe since the spindle must be positioned in the cycle several times by means of SPOS.

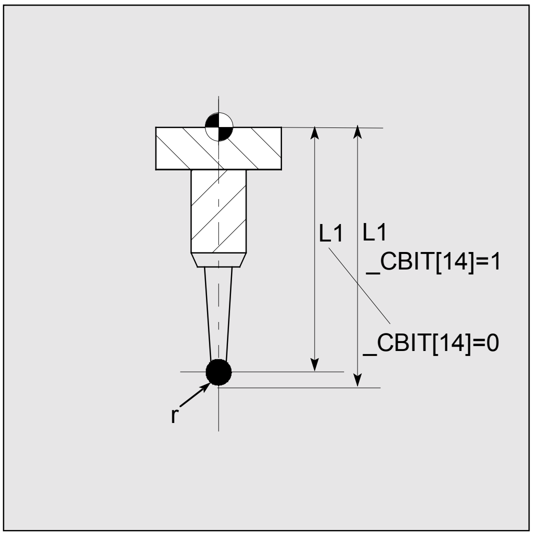
1.5 Workpiece probe, calibration tool in TO memory

1.5.1 Workpiece probe in TO memory for milling machines and machining centers

Workpiece probe
On milling machines and machining centers, the probe is classified as tool type 1x0 and must therefore be entered as such in the TO memory.
In SW 4 and higher, tool type 710 (3D probe) can also be used.

Entry in TO memory

P1	710	Tool type
P3	L1	Geometry
P6	r	Geometry
P21	L1	Tool base dimension



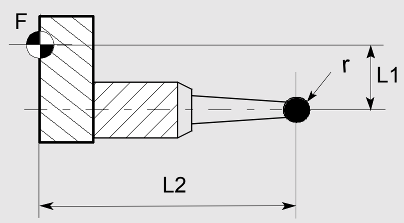
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.5.2 Workpiece probe, calibration tool in TO memory on turning machines

On turning machines, the probes are treated as tool type 500 with the permissible tool edge positions 5 to 8 and must therefore be entered like this in the TO memory. Measuring cycle SW 6.2 and higher also allows you to enter probe type 580 with tool edge positions 5 to 8. Due to their spatial positions, the probes are divided into the following types:

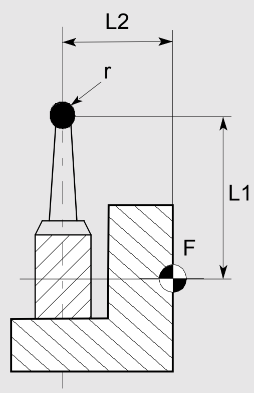
Workpiece probe SL 5
Entry in TO memory

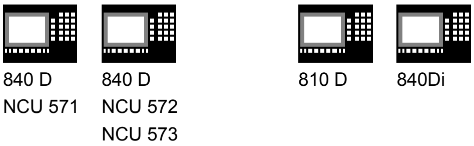
P1	500	Tool type
P2	5	Tool edge position
P3	L1	Geometry
P4	L2	Geometry
P6	r	Geometry
P12	L1	Wear
P13	L2	Wear
P15	r	Wear
P21	L1	Tool base dimension
P22	L2	Tool base dimension



Workpiece probe SL 6 (8)
(data in brackets is in front of turning center)
Entry in TO memory

P1	500	Tool type
P2	6 (8)	Tool edge position
P3	L1	Geometry
P4	L2	Geometry
P6	r	Geometry
P12	L1	Wear
P13	L2	Wear
P15	r	Wear
P21	L1	Tool base dimension
P22	L2	Tool base dimension

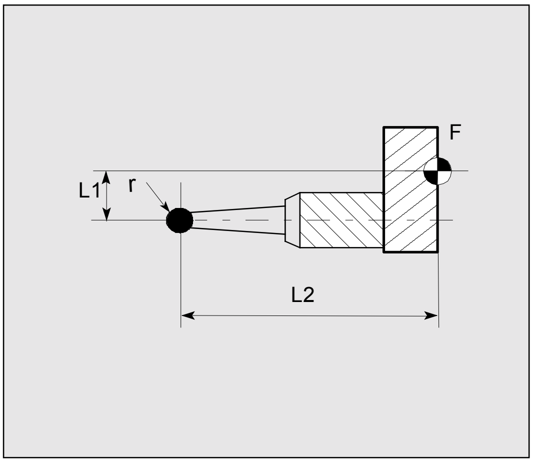




Workpiece probe SL 7

Entry in TO memory

P1	500	Tool type
P2	7	Tool edge position
P3	L1	Geometry
P4	L2	Geometry
P6	r	Geometry
P12	L1	Wear
P13	L2	Wear
P15	r	Wear
P21	L1	Tool base dimension
P22	L2	Tool base dimension

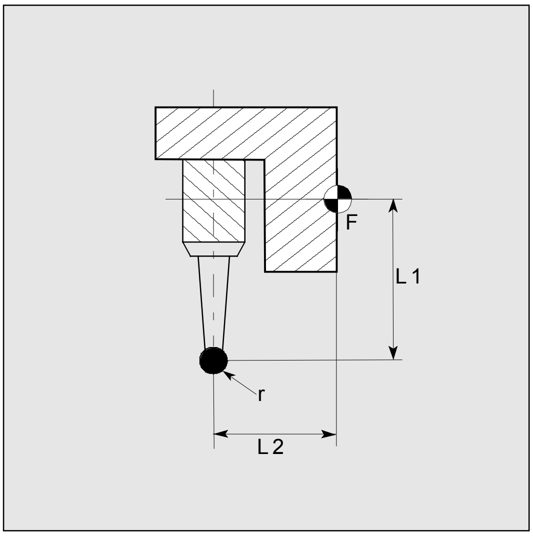


Workpiece probe SL 8 (6)

(data in brackets is in front of turning center)

Entry in TO memory

P1	500	Tool type
P2	8 (6)	Tool edge position
P3	L1	Geometry
P4	L2	Geometry
P6	r	Geometry
P12	L1	Wear
P13	L2	Wear
P15	r	Wear
P21	L1	Tool base dimension
P22	L2	Tool base dimension

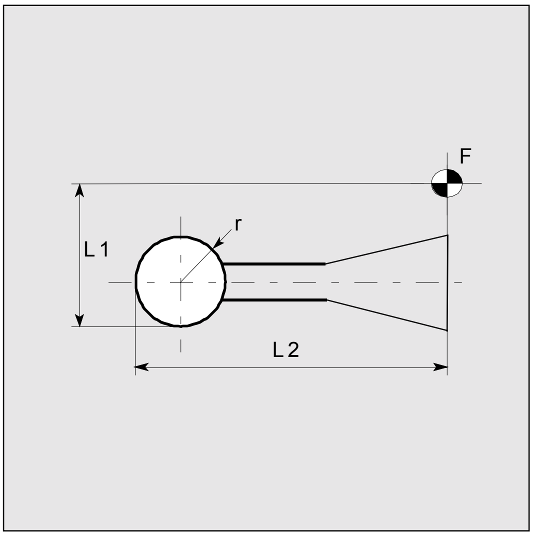


Calibration tool

On turning machines, the calibration tool is classified as a tool with tool edge position 3 and must therefore be entered as such in the TO memory.

Entry in TO memory

P1	500	Tool type
P2	3	Tool edge position
P3	L1	Geometry
P4	L2	Geometry
P6	r	Geometry
P12	L1	Wear
P13	L2	Wear
P15	r	Wear
P21	L1	Tool base dimension
P22	L2	Tool base dimension



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

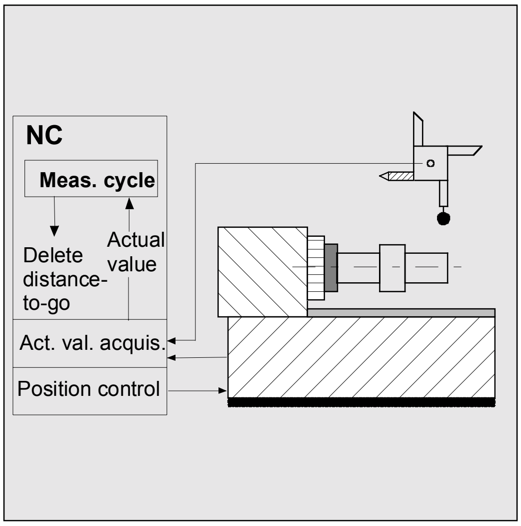
1.6 Measuring principle

Two inputs for the connection of touch trigger probes are provided on the I/O device interface of the SINUMERIK 840D and the FM-NC control systems.

Function

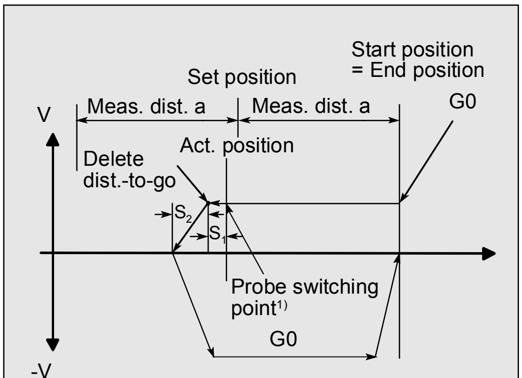
Evaluation of the measuring probe signal

If a measuring point is to be approached, a traverse command is transmitted to the position control loop and the probe is moved towards the measuring point. A point behind the expected measuring point is defined as setpoint position. As soon as the probe makes contact, the actual axis value at the time the switching position is reached is measured and the drive is stopped. The remaining "distance-to-go" is deleted.



"On-the-fly" measurement

The principle of "on-the-fly" measurement is implemented in the control. The advantage of this method is that the probe signal is processed directly in the NC.



S_1=Traversing path by signal processing
S_2=Following error

1) Actual value loaded with probe signal

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

Start position/setpoint position
In the measuring procedure used, a position is specified as setpoint value for the cycle at which the signal of the touch-trigger probe is expected.

Since it is unlikely that the probe will respond at precisely this point, the start position is approached by the control in rapid traverse mode or at a defined positioning velocity. The set position is then approached at the feedrate specified in the parameter for measurement speed. The switching signal is then anticipated over a distance of a maximum length of 2a from the start position.

Load actual value/delete distance-to-go
At the instant the switching signal is output by the probe, the current position is stored internally "on-the-fly" as the actual value followed by execution of the "Delete distance-to-go" function.

Measuring path a/measuring speed
The path increment a is normally 1 mm, but can be increased with a parameter when measuring cycles are called.

The approach speed automatically increases from 150 mm/min to 300 mm/min if the value for a is defined as greater than 1.

The maximum approach speed (measurement speed) is thus dependent upon

- the permissible deflection path of the probe used
- the delay until "delete distance to go" is executed and
- the deceleration behavior of the axis.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

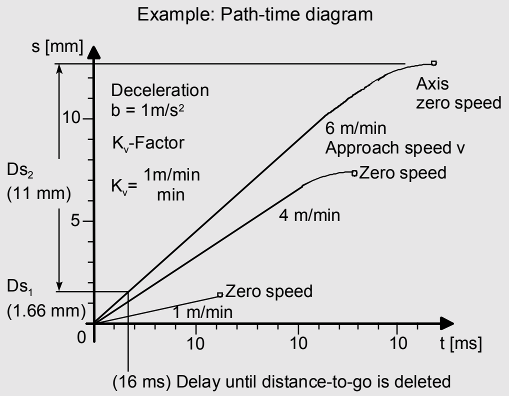
840Di

Calculation of the deceleration path

Since an optimal measurement speed can be set for measuring cycles via a parameter, it must be ensured that safe deceleration can take place within the deflection path of the probe.
The required deceleration path can be calculated as follows:

$$s_b = \underbrace{v \cdot t}_{\Delta s_1} + \underbrace{\frac{v^2}{2a} + \Delta s}_{\Delta s_2}$$

s_b	Deceleration path	in m
v	Approach speed	in m/s
t	Delay	in s
b	Deceleration	in m/s^2
s	Following error	in m



The deflection of the probe up to zero speed of the axis is approximately 12.6 mm with an approach speed of 6 m/min and a delay of 1 m/s²!

Measuring accuracy

The repeat accuracy of the 840D and FM-NC controls for "on-the-fly measurement" is ±1 µm.

The measuring accuracy which can be obtained is thus dependent on the following factors:

- Repeat accuracy of the machine
- Repeat accuracy of the probe
- Resolution of the measuring system

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

1.7 Measuring strategy and compensation value calculation for tools with automatic tool offset

The actual workpiece dimensions must be measured exactly in order to be able to determine and compensate the actual dimensional deviations on the workpiece.

Function

When taking measurements on the machine, the actual dimensions are derived from the path measuring systems of the position-controlled feed axes. For each dimensional deviation determined from the set and actual workpiece dimensions there are many causes which essentially can be classified in 3 categories:

- **Dimensional deviations with causes that are n o t subject to a particular trend,** e.g. positioning scatter of the feedforward axes or differences in measurement between the internal measurement (measuring probe) and the external measuring device (micrometer, measuring equipment, etc.).

  In this case, it is possible to apply so-called **empirical values**, which are stored in separate memories. The set/actual difference determined is automatically compensated by the empirical value.

- **Dimensional deviations with causes that a r e subject to a particular trend**, e.g. tool wear or thermal expansion of the leadscrew.

  These deviations are compensated by specifying fixed threshold values.

- **Accidental dimensional deviations,** e.g. due to temperature fluctuations, coolant or slightly soiled measuring points.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Assuming the ideal case, only those dimensional deviations which are subject to a trend can be taken into account for compensation value calculation. Since, however, it is hardly ever known to what extent and in which direction accidental dimensional deviations influence the measurement result, a strategy (floating average value generation) is needed which derives a compensation value from the actual/set difference measured.

Mean value calculation

Mean value calculation in combination with a higher-order measurement weighting has proved a suitable means to do this.

The formula of the mean value generation chosen is:

$$Mv_{new} = Mv_{old} - \frac{Mv_{old} - D_i}{k}$$

Mv_{new} Mean value new = amount of compensation
Mv_{old} Mean value prior to last measurement
k Weighting factor for average value calculation
D_i Actual/set difference measured
(minus empirical value, if any)

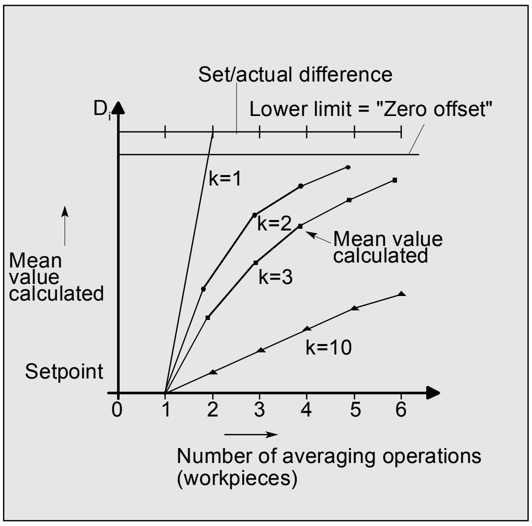
The mean value calculation takes account of the trend of the dimensional deviations of a machining series, where **weighting factor k** from which the mean value is derived is selectable.

A new measurement result affected by accidental dimensional deviations only influences the new tool offset to some extent, depending on the weighting factor.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

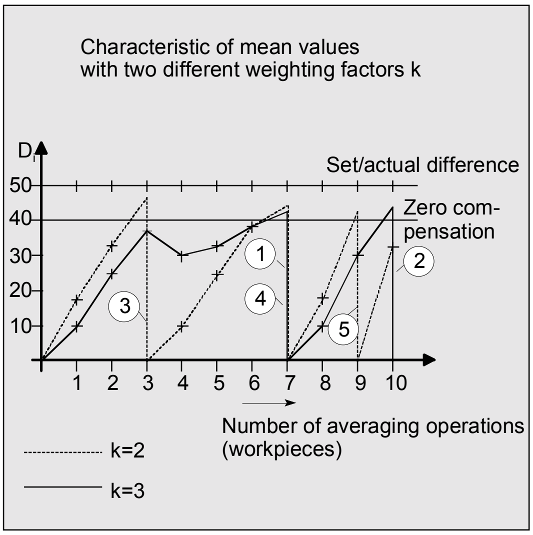
Computational characteristic of the mean value with different weightings k (effects)

- The greater the value of k, the slower the formula will respond when major deviations occur in computation or counter compensation. At the same time, however, accidental scatter will be reduced as k increases.
- The lower the value of k, the faster the formula will react when major deviations occur in computation or counter compensation. However, the effect of accidental variations will be that much greater.
- The mean value Mv is calculated starting at 0 over the number of workpieces i, until the calculated average value exceeds the range of "zero compensation". From this limit on, the calculated average value is applied for compensation.



Example of mean value generation

	Lower limit = 40 µm		
	D_i [µm]	Mean value k=3 [µm]	Mean value k=2 [µm]
1st measurement	30	10	15
2nd measurement	50	23.3	32.5
3rd measurement	60	35.5	46.2 ③
4th measurement	20	30.3	10
5th measurement	40	32.6	25
6th measurement	50	38.4	37.5
7th measurement	50	42.3 ①	43.75 ④
8th measurement	30	10	15
9th measurement	70	30	42.5 ⑤
10th measurement	70	43.3 ②	35



840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

1.8 Parameters for checking the dim. deviation and compensation

Explanation

For constant deviations **not subject to a trend** the dimensional deviation measured can be compensated by an empirical value for certain measurement variants. For other compensations resulting from dimensional deviations, symmetrical tolerance bands are assigned to the set dimension which result in different responses.

Empirical value _EVNUM
The empirical values are used to suppress dimensional deviations **that are not subject to a trend**.

The empirical values are stored in the GUD field
_EV empirical value.

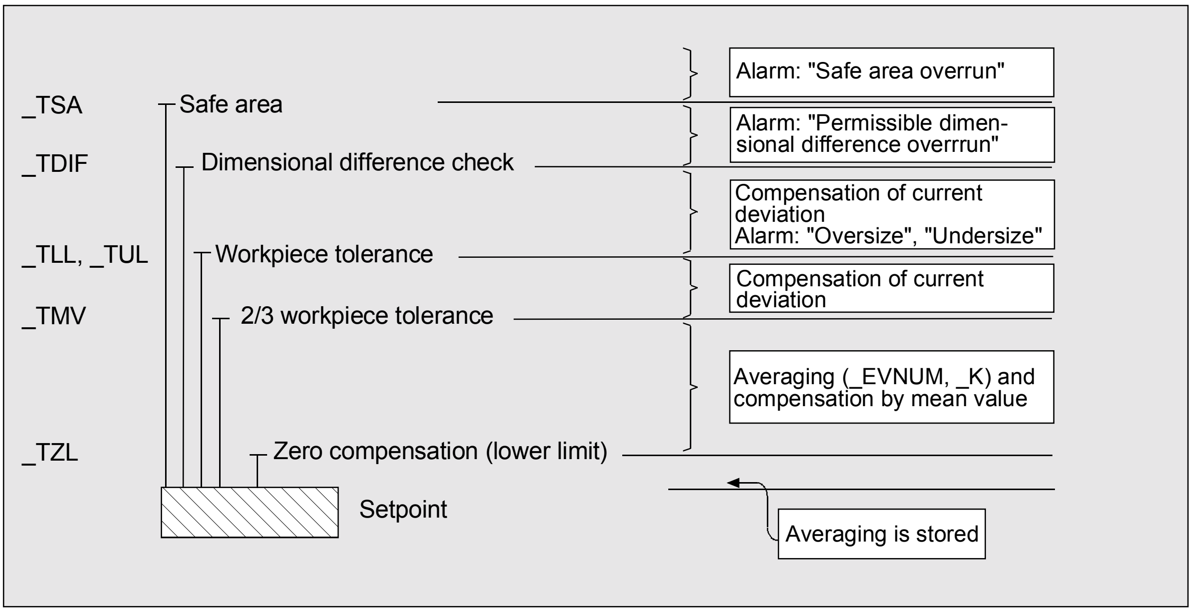
_EVNUM specifies the number of the empirical value memory. The actual/set difference determined by the measuring cycle is corrected by this value **before** any further correction measures are taken.
This is the case

- for workpiece measurement with automatic tool offset
- for tool measurement
- for single-point measurement with automatic ZO compensation

The tolerance bands (range of permissible dimensional tolerance) and the responses derived from them have been specified as follows:

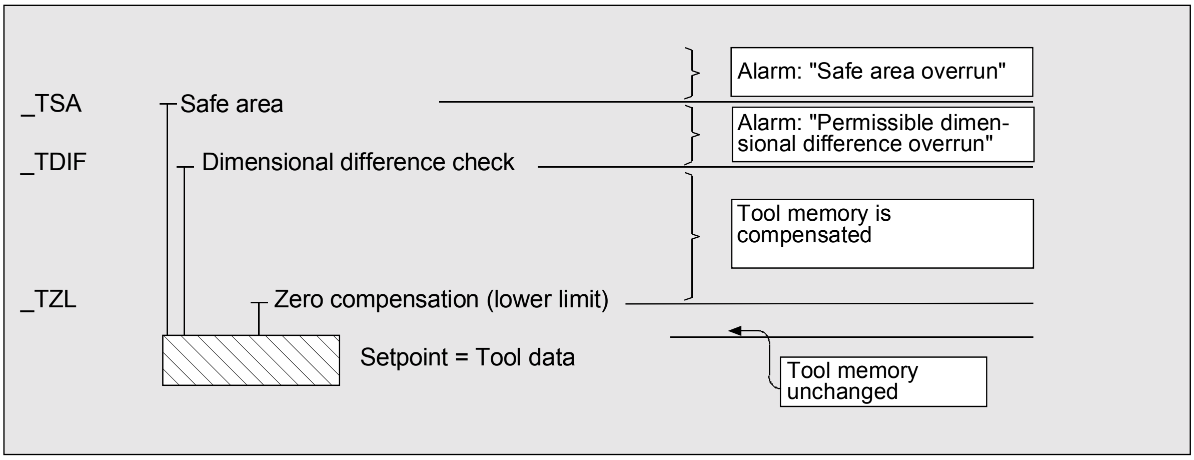
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

- **For workpiece measurement with automatic tool offset**



The workpiece set dimension is placed in the center of the permissible ± tolerance limit applied.

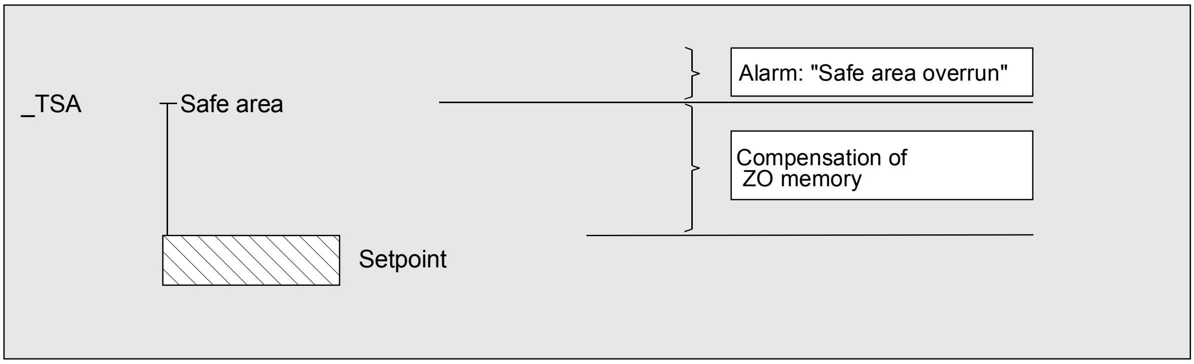
- **For tool measurement**



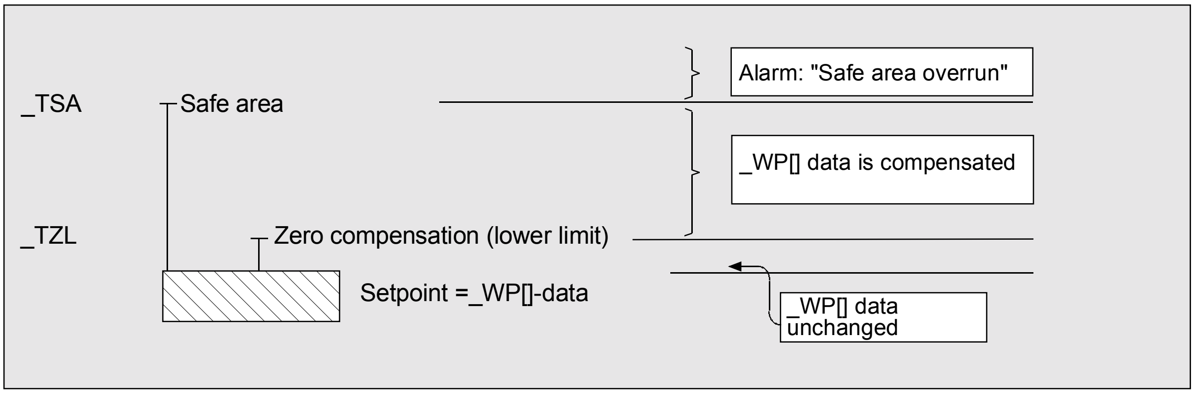
1.8 Parameters for checking the dim. deviation and compensation

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

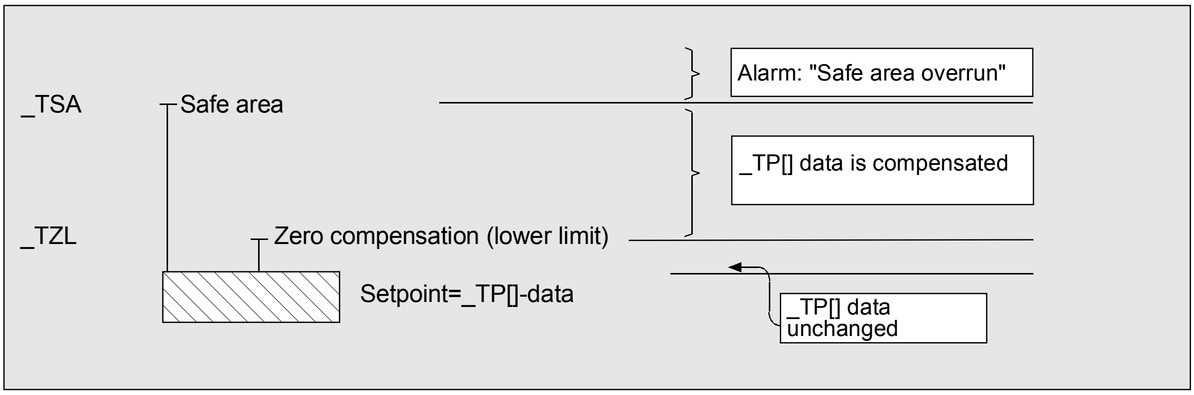
- **For workpiece measurement with zero offset**



- **For workpiece probe calibration**



- **For tool probe calibration**



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Safe area _TSA

The safe area is active for all measurement variants and does not affect the offset value; it is used for diagnosis.

If this value is reached,

- a defect in the probe,
- an incorrect setpoint position or
- an illegal deviation from the setpoint position

may be the cause.

AUTOMATIC operation is interrupted and the program cannot continue. An alarm text appears to warn the user.

Dimensional difference control _TDIF

_TDIF is active only for workpiece measurement with automatic tool offset and for tool measurement.

This limit has no effect on generation of the compensation value either. When it is reached, the tool is probably worn and needs to be replaced.

An alarm text is displayed to warn the operator and the program can be continued by means of an NC start.

This tolerance limit is generally used by the PLC for tool management purposes (twin tools, wear monitoring).

Tolerance of the workpiece _TLL, _TUL

Both parameters are active only for tool measurement with automatic tool offset.

When measuring a dimensional deviation ranging between "2/3 tolerance of workpiece" and "Dimensional difference control", this is regarded 100% as tool compensation. The previous average value is erased.

It is therefore possible to effect fast counteraction if major dimensional deviations occur.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

AUTOMATIC operation is interrupted when the tolerance limit of the workpiece is exceeded. "Oversize" or "undersize" is displayed to the operator depending on the tolerance zone position. Machining can be continued by means of NC start.

2/3 workpiece tolerance _TMV
_TMV is active only for workpiece measurement with automatic tool offset.
Within the range of "Lower limit" and "2/3 workpiece tolerance" the mean value is calculated according to the formula described in Section "Measuring strategy".

Mv_{new} is compared with the zero compensation range:

- If Mv_{new} **is greater** than this range, compensation is corrected by Mv_{new} and the associated mean value memory is cleared.
- If Mv_{new} **is less** than this range, no compensation is carried out to prevent excessively abrupt compensations from being made.

Mean value_EVNUM
_EVNUM is active only for workpiece measurement with automatic tool offset.
When calculating the mean value in a series of machining operations, the mean value determined by the measurement at the same measurement location on the previous workpiece can be taken into account (_CHBIT[4]=1).
The mean values are stored in the GUD field **_MV mean values**. **_EVNUM** also specifies the number of the mean value memory in this GUD field.

Weighting factor for mean value calculation _K
_K is active only workpiece measurement with automatic tool offset. The weighting factor k can be applied to allow different weighting to be given to an individual measurement.

A new measurement result thus has only a limited effect on the new tool offset as a function of _K.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Bottom limit (zero compensation area) _TZL

_TZL active for

- Workpiece measurement with automatic tool offset
- Tool measurement and calibration for milling tools and tool probes

This tolerance range corresponds to the amount of maximum accidental dimensional deviations. It has to be determined for each machine.

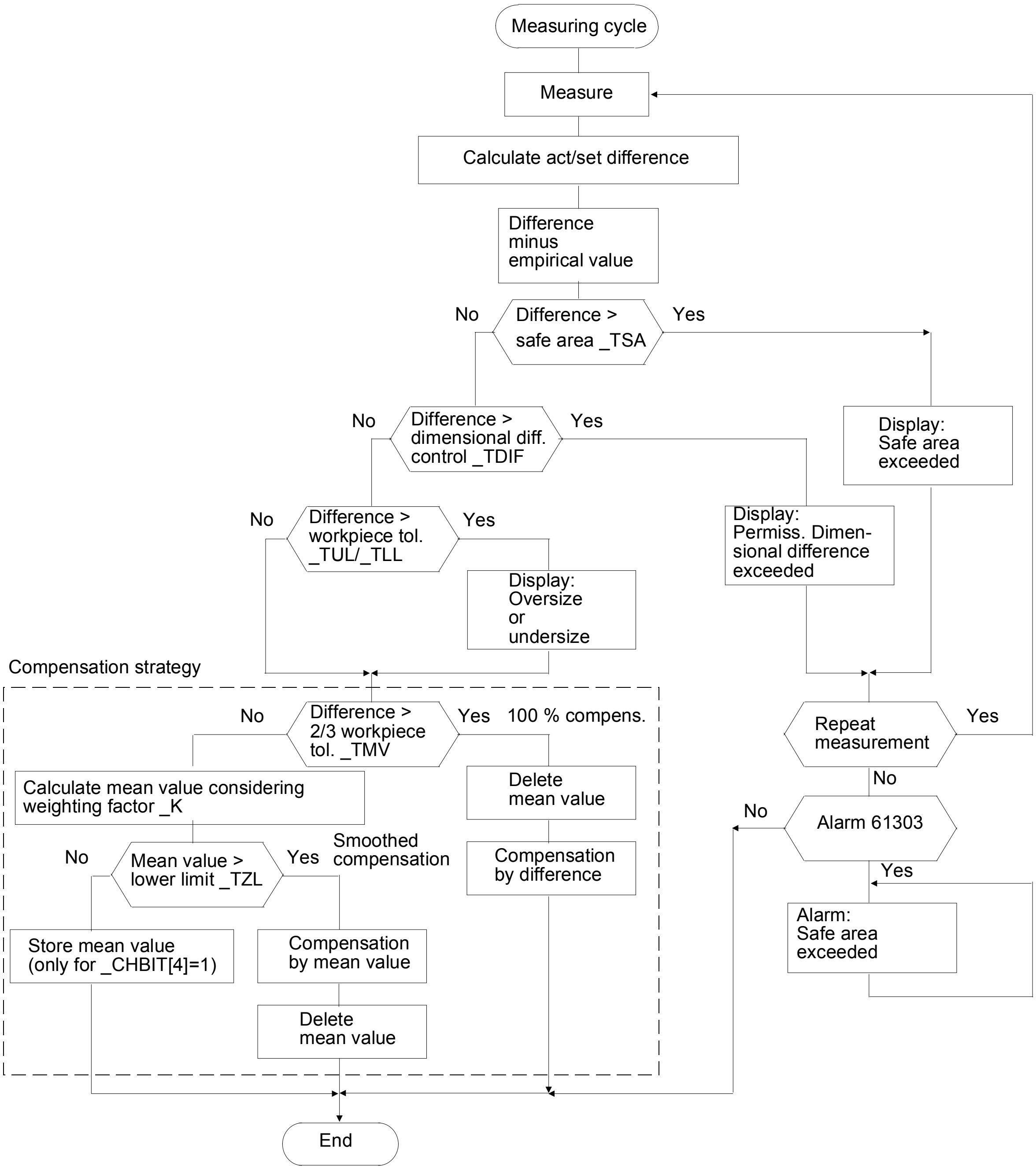
No tool compensation is made within these limits.

However, the average value of this measuring point is updated and re-stored with the actual/set difference measured for workpiece measurement with automatic tool offset, compensated by an empirical value if necessary.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.9 Effect of empirical value, mean value and tolerance parameters

The following flowchart shows the effect of empirical value, mean value and tolerance parameters by way of a workpiece measurement with automatic tool offset.

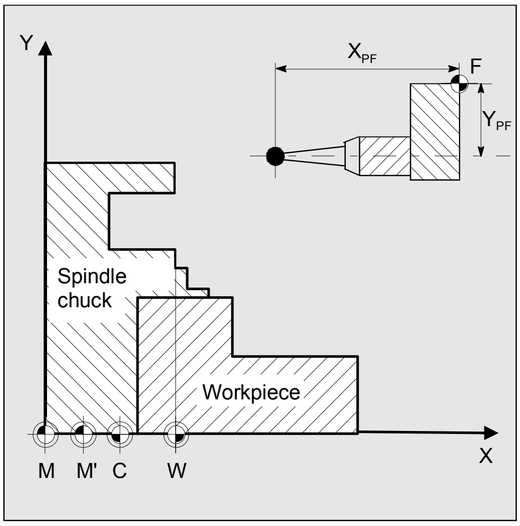


840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.10 Reference points on the machine and workpiece

Function

The actual axis values of different actual value systems must be measured depending on the measuring process applied. While, for example, the machine actual value can be used to advantage to calculate the tool length, the workpiece zero is important for measuring workpiece dimensions and calculating the tool wear compensation. The machine actual value is the dimension between the machine zero and the tool reference point.



M = Machine zero
M' = Machine zero offset by DRF
C = Control zero resulting from PRESET offset
W = Workpiece zero
F = Tool reference point

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

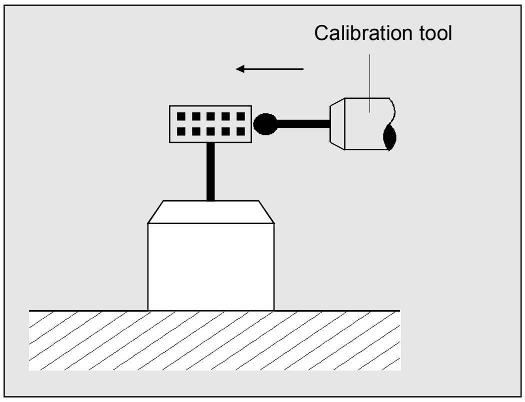
1.11 Measurement variants for milling machines & machining centers

The measurement variants which can be implemented with measuring cycles for milling machines and machining centers are illustrated in diagrams below.

1.11.1 Workpiece measurement for milling machines

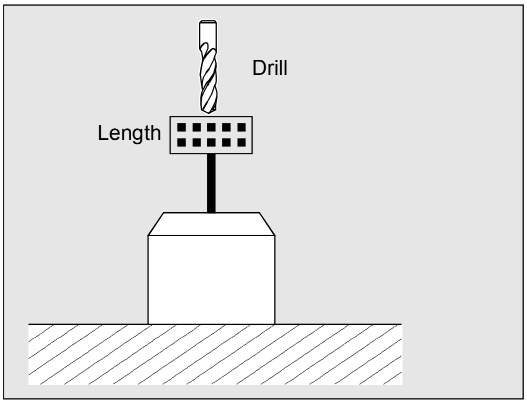
Tool probe calibration

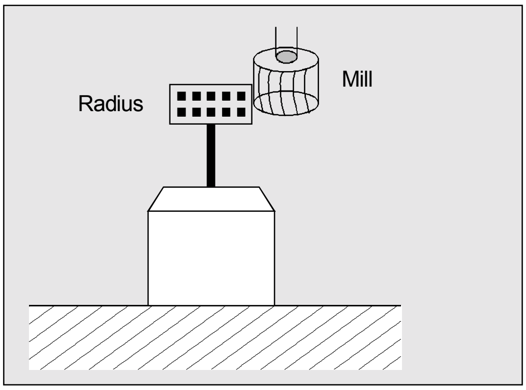
Result:
Probe switching point with reference to machine zero



Measuring the tool

Result:
Tool length
Tool radius





840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.11.2 Measurement variants for fast measurement at a single point

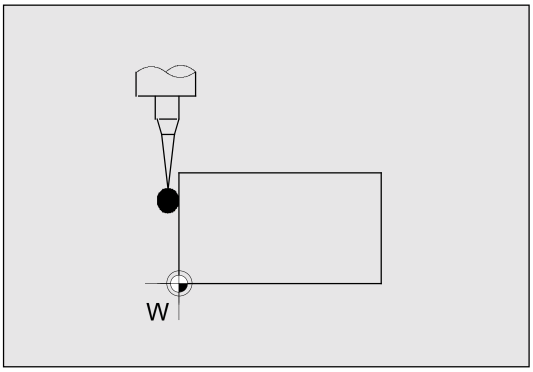
Function

CYCLE978 makes it easy to take a measurement at one point of a surface.

The measuring point is approached paraxially.

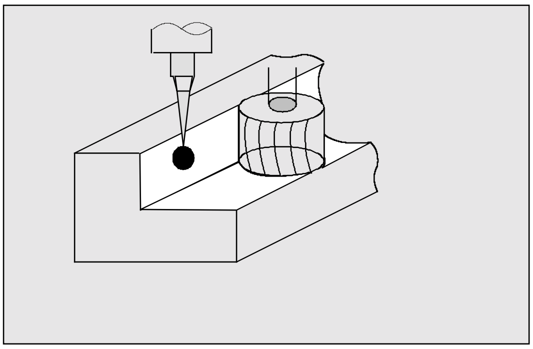
Depending on the measurement variant, the result may influence the selected tool offset or zero offset.

Workpiece measurement, blank measurement



Result:
Position, deviation,
Zero offset

Workpiece measurement, single-point measurement



Result:
Actual dimension, deviation,
tool offset

1.11.3 Measurement variants for workpiece measurement paraxial

Function

The following measurement variants are provided for the paraxial measurement of a hole, shaft, groove or web. They are executed by the cycle CYCLE977.

1.11 Measurement variants for milling machines & machining centers

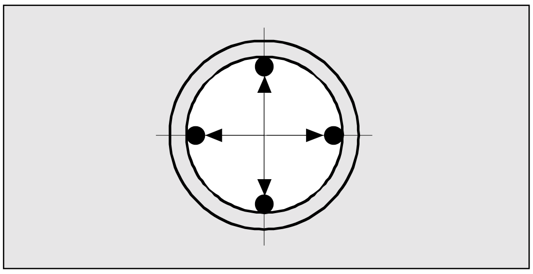
840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

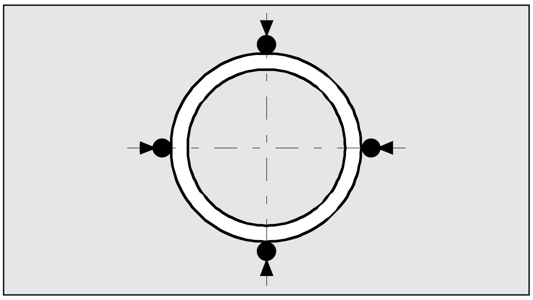
840Di

Workpiece measurement, measuring the hole



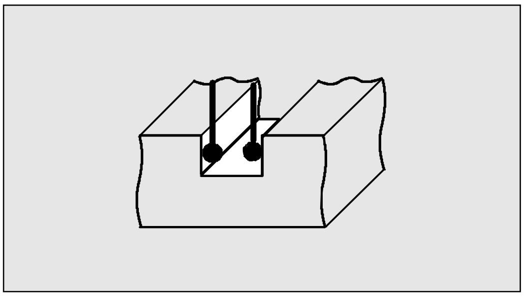
Result:
Actual dimension (diameter),
deviation, center point,
tool offset,
zero offset

Workpiece measurement, measuring the shaft



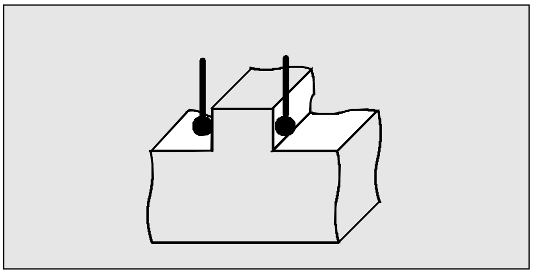
Result:
Actual dimension (diameter),
deviation, center point,
tool offset,
zero offset

Workpiece measurement, measuring the groove



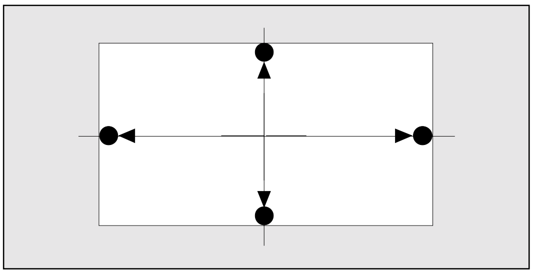
Result:
Actual dimension (groove width),
deviation, groove center,
tool offset,
zero offset

Workpiece measurement, measuring the web



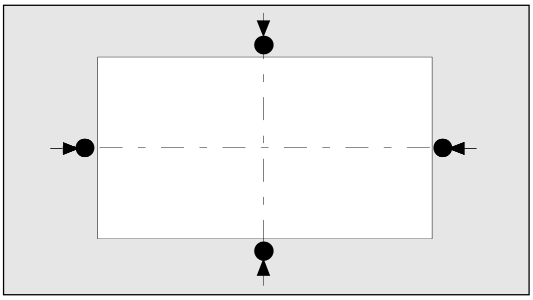
Result:
Actual dimension (web width),
deviation, web center,
tool offset,
zero offset

Workpiece measurement, measuring the inside rectangle



Result:
Actual value rectangle length and width,
actual dimension rectangle center,
deviation rectangle length and width,
deviation rectangle center,
tool offset,
zero offset

Workpiece measurement, measuring the outside rectangle



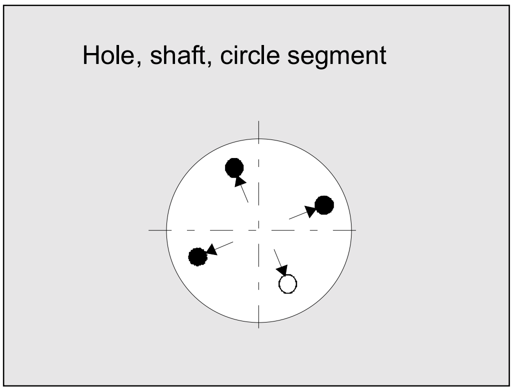
Result:
Actual value rectangle length and width,
actual dimension rectangle center,
deviation rectangle length and width,
deviation rectangle center,
tool offset,
zero offset

1.11.4 Measurement variants for workpiece measurement at random angles

Function

The following measurement variants are provided for the measurement of a bore, shaft, groove or web at random angles. They are executed by CYCLE979.

Triple-point (quadruple-point) measurement at random angles

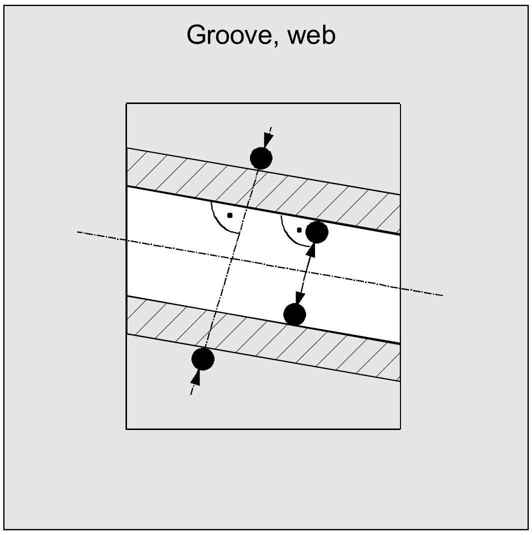


Result:
Actual dimension (diameter),
deviation, center point,
tool offset,
zero offset

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Two-point measurement at random angles

Result:
Actual dimension (groove width, web width), deviation, groove center, web center, zero offset



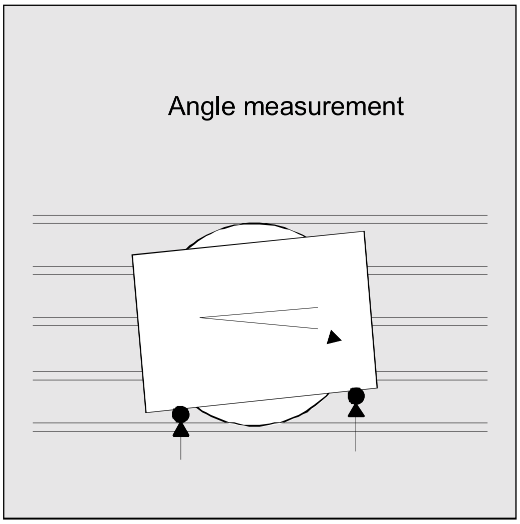
1.11.5 Measuring a surface at a random angle

Function

The zero offset can be compensated after measurement of a surface at a random angle by means of CYCLE998.

Workpiece measurement, angular measurement

Result:
Actual dimension (angle), deviation, zero offset



840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

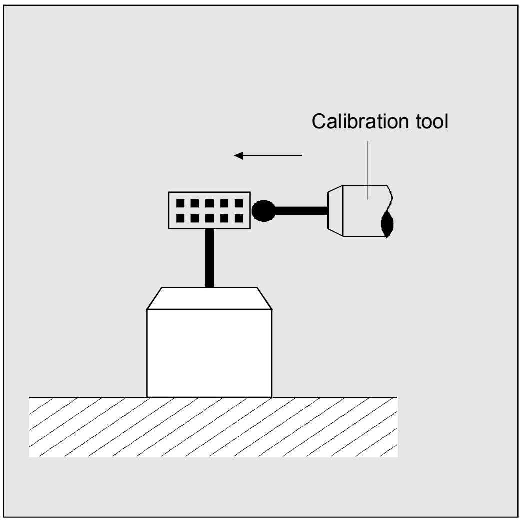
840Di

1.12 Measurement variants for lathes

1.12.1 Tool measurement for lathes

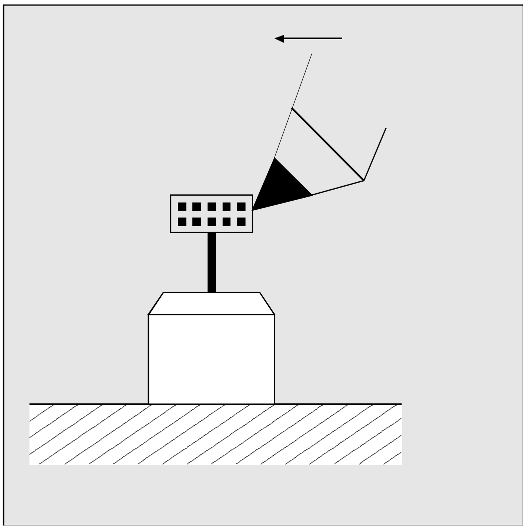
Tool probe calibration

Result:
Probe switching point with reference to machine zero



Measuring the tool

Result:
Tool length (length1, length2)

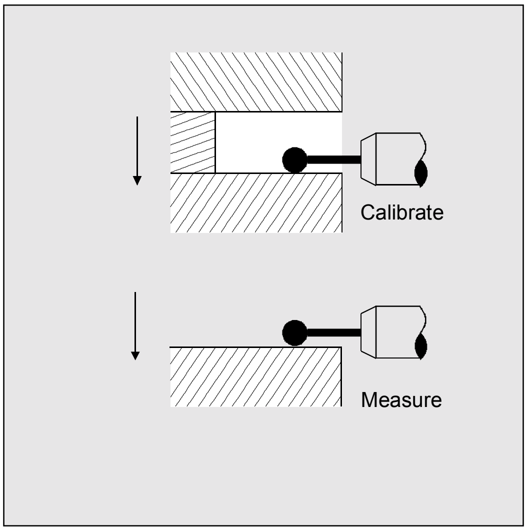


840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.12.2 Workpiece measurement for turning machines: Single-point measurement

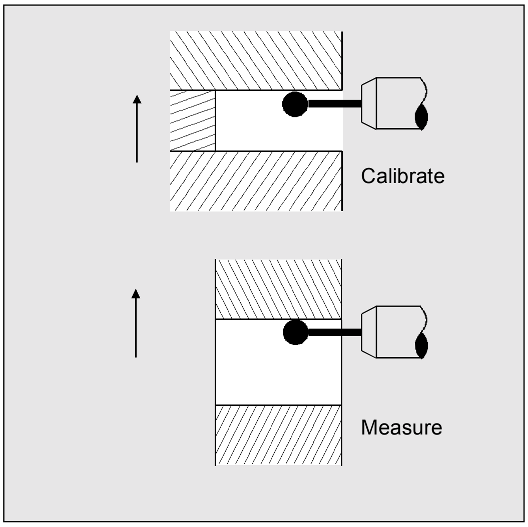
Single-point measurement outside

Result:
Actual dimension (diameter, length),
deviation,
tool offset,
zero offset



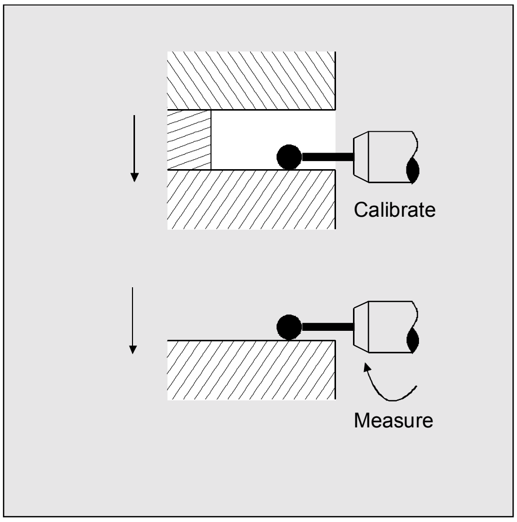
Single-point measurement inside

Result:
Actual dimension (diameter, length),
deviation,
tool offset,
zero offset



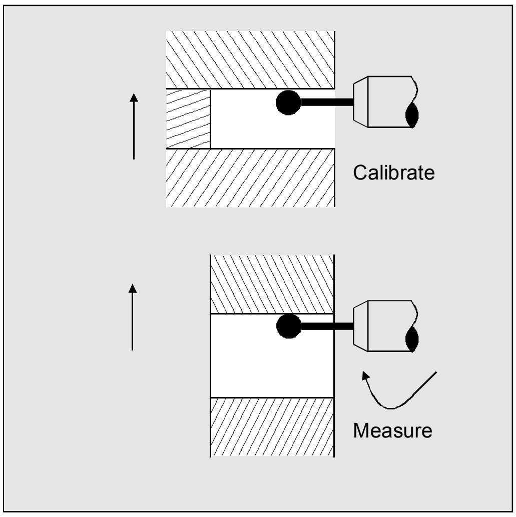
Single-point measurement outside with 180° reversal spindle

Result:
Actual dimension (diameter, length),
deviation,
tool offset



Single-point measurement inside with 180° reversal spindle

Result:
Actual dimension (diameter, length),
deviation,
tool offset

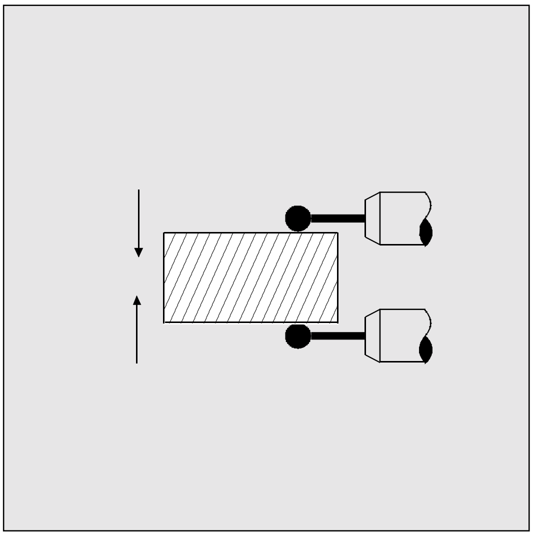


840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.12.3 Workpiece measurement for turning machines: Two-point measurement

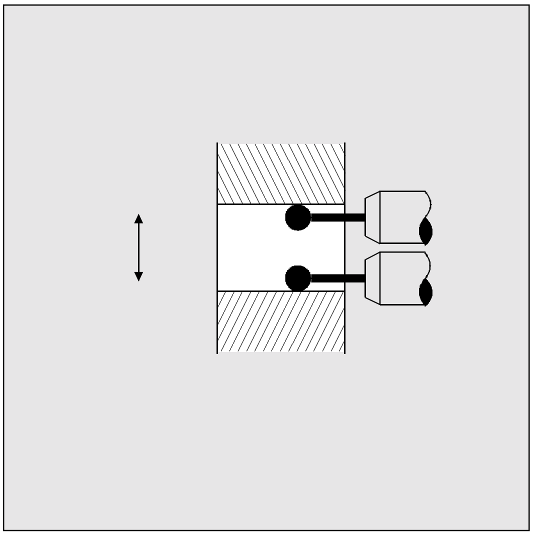
Two-point measurement on outside diameter

Result:
Actual dimension (diameter),
deviation,
tool offset



Two-point measurement on inside diameter

Result:
Actual dimension (diameter),
deviation,
tool offset



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.13 Measuring cycles interface

The measuring cycles provide an interactive function for defining input and output parameters.

Values can be assigned to the input parameters via a help cycle in an input dialog.

The results of measurement can be displayed automatically via another help cycle.

1.13.1 Displaying measuring result screens

Function

Measuring results can be displayed automatically while a measuring cycle is running.

Activation of this function depends on the configuration of the measuring cycle interface in the MMC and the settings in the measuring cycle data.

Observe the specifications of the machine manufacturer.

Depending on the configuration

- the measuring result displays are automatically deselected at the end of a measuring cycle
- the measuring result displays must be acknowledged with the NC Start key;

In this case, the measuring cycle outputs the message:
"Please acknowledge measuring result display with NC Start".

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Explanation

The measuring cycles can display different measuring result screens depending on the measurement variant:

- Tool probe calibration
- Tool measurement
- Workpiece probe calibration
- Workpiece measurement

The result displays contain the following data:

Calibrating the tool probe
– Measuring cycle and measurement variant
– Probe ball diameter and difference
– Trigger values of axis directions and differences
– Positional deviation during calibration on the plane
– Probe number
– Safe area

Tool measurement
– Measuring cycle and measurement variant
– Actual values and differences for tool offsets
– T number and D number

Calibrate tool probe
– Measuring cycle and measurement variant
– Trigger values of axis directions and differences
– Positional deviation during calibration on the plane
– Probe number
– Safe area and permissible dimensional difference

Workpiece measurement
– Measuring cycle and measurement variant
– Setpoints, actual values and their differences
– Upper and lower tolerance limits
– Offset value
– Probe number
– Safe area and permissible dimensional difference
– T number and D number or ZO memory for automatic offset

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

1.13.2 Setting parameters

Function

Values can be assigned to measuring cycle parameters with CYCLE103.

Activation of this function depends on the configuration of the measuring cycle interface in the MMC.

Observe the specifications of the machine manufacturer.

Explanation

When CYCLE103 is selected and started, an input dialog for setting parameters for the measuring cycles is opened.
During the course of this dialog, a series of input screen forms are opened one after the other on top of the current display. Once the values have been entered each display must be concluded by pressing the OK key in the vertical softkey bar.
At the end of the dialog, the message
"Input dialog successfully completed"
is displayed in the dialog line of the control and the display before dialog mode was activated is reconstructed.

It is immediately possible to select and start the last measuring cycle assigned parameters.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Explanation

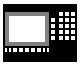
The sequence of the dialog for assigning parameters is as follows:

- Selection of the measuring cycle to which parameters are to be assigned;
- Selection of the measurement variant;
- Assignment of parameters for the measurement variant chosen, this could involve several input screen forms depending on the measuring cycle;
- Input and confirmation of generally applicable measuring cycle data which do not usually change.

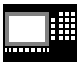
The input values for selecting the measuring cycle and the measurement variant are subjected to a plausibility check and the input screen forms are repeated if necessary.

If the operating area is switched over during the course of the input dialog, the dialog can be selected again at a later stage with "Cycles" softkey in the extended menu.

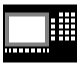
■



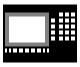
840 D
NCU 571



840 D
NCU 572
NCU 573



810 D



840Di

Notes

Description of Parameters

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

2.1. Parameter concept for measuring cycles

Function

As explained at the beginning, measuring cycles are general subroutines designed to solve specific measuring tasks. They can be adapted for this purpose by means of so-called **defining parameters**.

They also return data such as measurement results. They are stored in **result parameters**.

Furthermore, the measuring cycles also require **internal parameters** for calculations.

Defining parameters

The defining parameters of the measuring cycles are defined as **G**lobal **U**ser **D**ata (abbreviated to GUDs).

They are stored in the nonvolatile storage area of the control such that their setting values remain stored even when the control is switched off and on.

These data are contained in the data definition blocks

- GUD5.DEF and
- GUD6.DEF

which are supplied together with the measuring cycles.

Further notes

Many of the defining parameters have preset values.
See Section 2.2

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

These blocks must be loaded into the control during start-up. They must then be adapted by the machine manufacturer according to the characteristics of the relevant machine (see Part 2 Description of Functions, from Chapter 8 onwards).

Values can be assigned to these GUDs in the program or by means of keyboard inputs.

Result parameters

The results are also stored in specific GUDs.

Internal parameters

Local **U**ser **D**ata (abbreviated to LUDs) are used in the measuring cycles as internal arithmetic parameters.

These are set up in the cycle and thus exist only for the duration of the run-time.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.2 Parameter overview

2.2.1 Input parameters

Explanation

The defining parameters of the measuring cycles can be classified as follows:

- Mandatory parameters
- Additional parameters

Mandatory parameters are parameters that have to be adapted to the measuring task at hand (for example, setpoint axis, measuring axis, etc.) before each measuring cycle call.

Additional parameters can generally be assigned once on a machine. They are then valid **for each measuring cycle call** until they are modified by programming or operation.

All parameters with dimensions (see overview below), except for those marked 1), must be programmed in the unit of measurement of the basic system. The parameters marked 1) must be programmed in the unit of the active system of units.

Mandatory parameters

Parameters	*Type*	*Validity*	*Default:*	*Meaning*
_SETVAL[1)]	REAL	CHAN	-	Setpoint
_SETV[3] [1)]	REAL	CHAN	-	Measure setpoint values on rectangle
_ID[1)]	REAL	CHAN	-	Incremental infeed depth/offset
_CPA[1)]	REAL	CHAN	-	Center point abscissa for measuring at angle
_CPO[1)]	REAL	CHAN	-	Center point ordinate for measuring at angle
_SZA[1)]	REAL	CHAN	-	Protection zone in abscissa
_SZO[1)]	REAL	CHAN	-	Protection zone in ordinate
_STA1	REAL	CHAN	0	Initial angle
_INCA	REAL	CHAN	-	Indexing angle
_MVAR	INT	CHAN	-	Measurement variant
_MA	INT	CHAN	-	Measuring axis
_MD	INT	CHAN	-	Measuring direction
_TNUM	INT	CHAN	-	T number
_TNAME	STRING[32]	CHAN	-	Tool name (alternative to _TNUM in tool management)
_KNUM	INT	CHAN	0	Correction number (D No. or ZO No.)
_RA	INT	CHAN	-	Number of rotary axis at angle measurement

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Auxiliary parameters

Parameters	*Type*	*Validity*	*Default*	*Meaning*
_VMS	REAL	CHAN	0	Variable measuring velocity
_RF	REAL	CHAN	1000	Feedrate at circular-path programming
_CORA	REAL	CHAN	0	Compensation angle for mono probe
_TZL	REAL	CHAN	0.001	Zero offset area
_TMV	REAL	CHAN	0.7	Mean value generation with compensation
_TUL[1)]	REAL	CHAN	1.0	Upper tolerance limit
_TLL[1)]	REAL	CHAN	-1.0	Lower tolerance limit
_TDIF	REAL	CHAN	1.2	Dimensions difference check
_TSA	REAL	CHAN	2	Safe area
_FA	REAL	CHAN	2	Measuring path multiplication factor
_CM[8]	REAL	NCK	90, 2000, 1, 0, 0.005, 50, 4, 10	Monitoring parameters at tool measurement with rotating spindle
_PRNUM	INT	CHAN	1	Probe number
_EVNUM	INT	CHAN	0	Empirical value memory number
_CALNUM	INT	CHAN	0	Calibration block number
_NMSP	INT	CHAN	1	Number of measurements at the same location
_K	INT	CHAN	1	Weighting factor for mean value derivation

Parameters for logging only

Parameters	*Type*	*Validity*	*Meaning*
_PROTNAME[2]	STRING[32]	NCK	[0]: Name of main program the log is from [1]: Name of log file
_HEADLINE[6]	STRING[80]	NCK	6 strings for protocol headers
_PROTFORM[6]	INT	NCK	Formatting for protocol
_PROTSYM[2]	CHAR	NCK	Separator in the protocol
_PROTVAL[13]	STRING[100]	NCK	[0, 1]: Protocol header line [2-5]: Specification of the values to be logged [6-12]: Internal
_DIGIT	INT	NCK	Number of decimal places

2.2.2 Result parameters

Parameters	*Type*	*Validity*	*Meaning*
_OVR[32]	REAL	CHAN	Result parameters: Setpoint values, actual values, differences, offset values and others
_OVI[11]	INT	CHAN	Result parameter, integer

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

2.3 Description of the most important defining parameters

2.3.1 Measurement variant: _MVAR

Function

The measurement variant of the individual cycles is defined in parameter **_MVAR**.

Parameters

Values of _MVAR
The parameter can assume certain positive integers for each measuring cycle which are listed individually below.

The setting of parameter _MVAR is subjected to a plausibility check by the cycle. If it does not have a defined value, the following alarm message is output:
"Measurement variant incorrectly defined".
The cycle must be interrupted by an NC RESET.

Measurement and calibration variants for workpiece measurement on milling machines

	Possible values of _MVAR	Measurement variants
CYCLE976	0	Calibration on any surface (applicate)
	1...112101	Calibrate in random hole (plane)
	8...10108	Calibrate workpiece probe in any hole (plane) with unknown position of the hole center
CYCLE977 and CYCLE979	1	Measure hole
	2	Measure shaft
	3	Measure groove
	4	Measure web
	101	ZO calculation in hole
	102	ZO calculation in shaft
	103	ZO calculation in groove
	104	ZO calculation on web
CYCLE977	5	Measure rectangle inside
	6	Measure rectangle outside
	105	ZO calculation in rectangle inside
	106	ZO calculation in rectangle outside

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

CYCLE977	1001	Measure hole with travel around a protection zone
	1002	Measure shaft while accounting for a protection zone
	1003 1)	Measure hole with contouring of a protection zone
	1004 1)	Measure web by including for a protection zone
	1005	Measure rectangle inside with protection zone
	1006	Measure rectangle outside with protection zone
	1101	ZO calculation hole with travel around a protection zone
	1102	ZO calculation of shaft while accounting for a protection zone
	1103 1)	ZO calculation in groove with contouring of a protection zone
	1104 1)	ZO calculation at web by including a protection zone
	1105	ZO calculation in rectangle inside with protection zone
	1106	ZO calculation in rectangle outside with protection zone
CYCLE978	0	Measure surface
	100	ZO calculation on surface
	1000	Measure surface with differential measurement
	1100	ZO calculation on surface with differential measurement
CYCLE998	105	Angular measurement, ZO calculation
	1105	Angular measurement with differential measurement, ZO calculation

Measurement and calibration variants for tool measurement on milling machines

	Possible values of _MVAR	Measurement variants
CYCLE971	1	Measure tool with motionless spindle (Length or radius)
	2	Measure tool with rotating spindle (Length or radius)
	0	Calibration of the tool probe
	10000	Incremental calibration of the tool probe

Further notes

1) Measuring cycles SW 4.5 and higher

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Measurement and calibration variants for workpiece measurement on lathes

	Possible values of _MVAR	Measurement variants
CYCLE973	0	Calibration on any surface (applicate)
	13...12113	Calibration in reference groove (plane)
CYCLE974	0	Single-point measurement
	100	Single-point measurement ZO calculation
	1000	Single-point measurement with reversal
CYCLE994	1	Two-point measurement with protection zone (for inside measurement only)
	2	Two-point measurement with programmed protection zone (for inside measurement without protection zone)

Measurement and calibration variants for tool measurement on lathes

	Possible values of _MVAR	Measurement variants
CYCLE972	0	Tool probe calibration
	1	Tool measurement
CYCLE982 (measuring cycle SW 5.3 and higher)	0	Tool probe calibration
	1	Measuring turning and milling tools
	2	Automatic measurement

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.3.2 Number of measuring axis: _MA

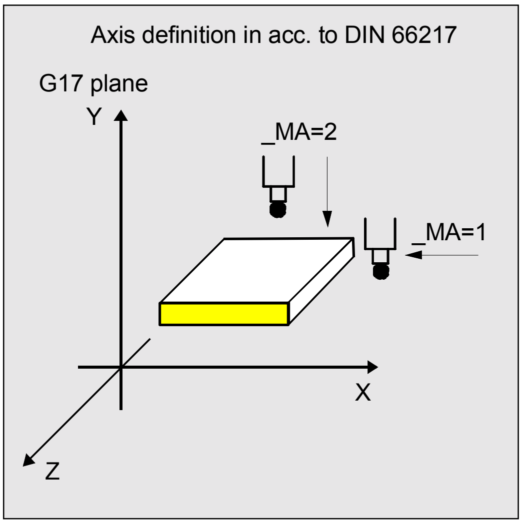
Function

The axis number (1...3) for the measuring axis in the coordinate system must be specified via _MA (not the hardware axis number).

Parameters

Values of _MA

Measuring axis abscissa	_MA = 1
Measuring axis ordinate	_MA = 2
Measuring axis applicate	_MA = 3



_MA must be defined with offset axis /measuring axis for certain measurement variants; in such cases, the first two digits contain the code for the offset axis and the second two digits the code for the measuring axis.

Example:

_MA = 102

⇒ Offset axis: 1 (abscissa)

⇒ Measuring axis: 2 (ordinate)

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.3.3 Tool number and tool name: _TNUM and _TNAME

Function

The tool to be offset is entered during workpiece measurement in the parameters _TNUM and _TNAME.

The parameter _TNAME is only relevant if tool management is active.

Parameters

The parameter _TNUM contains the tool number of the tool to be automatically offset during workpiece measurement.
If tool management is active, the name of the tool can be entered in parameter _TNAME as an alternative.

Example:

- without tool management:
  _TNUM = 12
  _TNAME = " " ⇒ is not assigned;

- with tool management:
  _TNUM = 0   _TNAME = "DRILL"
  ⇒ the tool with the name "DRILL" is offset
  or
  _TNUM = 13 _TNAME = " " or _TNAME="DRILL"
  ⇒ the tool with the internal T number 13 is offset

In SW 4 and higher with spare tools the one is offset which was last used (was in the spindle).
However, the requirement is that only one tool in a group is "active" at on time. Otherwise, the internal tool number of the tool used must be determined and assigned to _TNUM when machining via the system variable $P_TOOLNO.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.3.4 Offset number _KNUM

Parameters

The parameter _KNUM contains the tool offset memory number for workpiece measurement or the specification of the zero offset to be compensated for ZO calculation.

_KNUM setting values

_KNUM can accept integers with up to 6 digits, or 8 digits with flat D number structures. These digits have the following significance:

1. Specification for tool offset:
   Structure of tool offset parameter _KNUM

```
6 5 4 3 2 1
1           D number
3 2         Currently not assigned, i. e. = 0
4           0/1...Length compensation
              2...Radius compensation
5           Normal/inverted offset
            0...Normal
            1...Inverted
6           Offset acc. to 4th digit
            1...Offset of L1
            2...Offset of L2
            3...Offset of L3
            4...Radius compensation
```

In SW 5 and higher the last 3 digits are evaluated as a D number for a value of this MD from 10...999 depending on MD 18102: MM_TYPE_OF_CUTTING_EDGE = 0 and MD 18105: MM_MAX_CUTTING_EDGE_NO. If the value is ≥1000, _KNUM is evaluated as for a flat D number structure.

Example:

_KNUM = 12003
⇒ D3 is corrected,
⇒ calculated as radius offset
⇒ inverted correction

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2. Specification for zero offset:
   _KNUM=1 ... 99 Automatic inclusion of ZO in ZO G54 ... G57 and G505...G599

   In measuring cycle SW 4.4 and higher:
   _KNUM=1000 automatic ZO in basic frame G500 (offset always in the last channel-specific basic frame if there are more than one).

   In measuring cycle SW 6.2 and higher:
   - KNUM=1011...1026 automatic ZO in
     1st to 16th basic frame (channel) ($P_CHBFR[0]...$P_CHBFR[15])
   - KNUM=1051...1066 automatic ZO in
     1st to 16th basic frame (global) ($P_NCBFR[0]...$P_NCBFR[15])
   Note: The remaining active frame chain must be retained. With NCU-global frames, correction for rotation is not possible.
   - _KNUM=2000 automatic ZO in the system frame
     (scratch system frame $P_SETFR)
   - _KNUM=9999 automatic ZO in the active frame:
     settable frame G54...G57, G505...G599, or G500 in the last active basic frame according to $P_CHBFRMASK (most significant bit).
     Note: Only here does a changed frame become active in the cycle immediately, otherwise it is activated by the user writing G500, G54...G5xy.

The following must be set for start-up:
MD 28082: MM_SYSTEM_FRAME_MASK, Bit 0=1 and
Bit 5=1 (system frames for scratching and cycles)

With a _KNUM setting of 0, the automatic tool offset and ZO are deactivated.

In measuring cycle SW 6.2 and higher, CYCLE115 is introduced for the ZO. CYCLE114 is only responsible for the tool offset.

If a fine offset is active (MD 18600: MM_FRAME_FINE_TRANS), the additive ZO will be implemented in it (all measuring cycles with ZO except CYCLE961), otherwise it is implemented in the coarse offset. ZO with CYCLE961 is always in the coarse offset and any fine offset there may be is reset.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

2.3.5 Offset number _KNUM with flat D number structure

Parameters

The flat D number functionality is implemented in SW 4 and higher. Which type of D number management is valid is defined in MD 18102: MM_TYPE_OF_CUTTING_EDGE.

References: /FB/, W1, "Tool offset"

MD 18102:
0: as previously (default setting)
1: flat D number direct programming

With activation of flat D numbers, a five-digit D number is assumed in _KNUM.

```
8 7 6 5 4 3 2 1
| | | └───┬───┘
| | |     └──────── D number
| | └────────────── 0/1...Length compensation
| |                   2...Radius compensation
| └──────────────── Normal / inverted offset
|                   0...Normal
|                   1...Inverted
└────────────────── Offset acc. to 6th digit
                    1...Offset of L1
                    2...Offset of L2
                    3...Offset of L3
                    4...Radius compensation
```

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.3.6 Variable measuring speed: _VMS

Parameters

The measuring speed can be freely selected by means of **_VMS**. It is specified in mm/min or inch/min depending on the basic system.

The maximum measuring speed must be selected such that safe deceleration within the probe deflecting path is ensured.

When _VMS = 0, then the feedrate is preset as standard to 150 mm/min. This value is automatically increased to 300 mm/min if the measuring path a (_FA > 1) is altered via _FA.
If the basic system is in inches, 5.9055 inch/min or 11.811 inch/min takes effect.

2.3.7 Compensation angle position for monodirectional probe: _CORA

Function

When using a monodirectional probe, it may be necessary for machine-specific reasons (e.g. horizontal/vertical millhead) to correct the position of the probe to be able to carry out the measurement.

Parameters

The incorrect position can be corrected by means of parameter **_CORA**. Generally speaking, _CORA is set to 90° or a multiple thereof. If the direction of rotation is altered as a result of swiveling the milling head, then _CORA must be preset to –360° (normally 0°).

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.3.8 Tolerance parameters: _TZL, _TMV, _TUL, _TLL, _TDIF and _TSA

Some information about the tolerance parameters applied in conjunction with measuring cycles is already given in Section 1.8.

Parameters

These parameters contain the following variables:

_TZL	Zero offset[1)2)]
_TMV	Average-value generation with compensation[1)]
_TUL/_TLL	Workpiece tolerance[1)]
_TDIF	Dimension difference check[1)2)]
_TSA	Safe area

1) for workpiece measurement with automatic tool offset only
2) also for tool measurement

Value range

All of these parameters are capable of assuming any value. However, only values increasing from _TZL to _TSA are meaningful. Parameters _TUL/_TLL are specified in mm or inches depending on the active dimension system. All other parameters are programmed in the basic system.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.3.9 Multiplication factor for measurement path 2a: _FA

Parameters

Path increment **a** is 1 mm irrespective of the dimension system, but can be increased with parameter **_FA** when the measuring cycles are called and defines the distance from the expected position at which the probe is triggered.
The maximum value for _FA is calculated as follows:

$$\_FA_{max} = \frac{\text{Axis traversing path}_{max}}{2}$$

The measuring cycles automatically generate a measurement path of 2**a** · _FA, which is traversed at the measuring feedrate, i.e. at a distance of **a** · _FA in front of the specified setpoint position at which the probe is actuated under ideal conditions, up to **a** · _FA after the anticipated setpoint position.
If the probe is triggered during this measurement path the movement is aborted with delete distance-to-go.

Example:_FA=5
→ Irrespective of the system of units, a measurement path of 10 mm is generated, starting at 5 mm before and ending 5 mm after the specified setpoint position.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.3.10 Probe type/Probe number: _PRNUM

Function

The data relating to the workpiece probes are stored in GUD field **_WP Workpiece probe**, the data relating to the tool probes are stored in GUD field **_TP Tool probe**.

The data fields _WP and _TP are configured by the machine manufacturer during start-up.
_PRNUM specifies the number of the selected data field within these fields and the probe type.

Parameters

Values of _PRNUM
PRNUM can assume integers of three digits. In this case, the first digit represents the probe type, i.e.

- 0 = Multidirectional probe
- 1 = Monodirectional probe.

The other two digits contain the code for the probe number.

Digit			*Meaning*
3	**2**	**1**	
	-	-	Probe number (two digits)
0			Multiprobe probe
1			Mono probe

Example of workpiece measurement:

_PRNUM	= 102	
	⇒ Probe type:	Monodirectional probe
	⇒ Data field number:	2

Further notes

The associated field index in _WP = 1, i. e. the data of the _WP[1,0...9] field are considered by the measuring cycle in the calculation of the measuring results.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

2.3.11 Empirical value/mean value: _EVNUM

Function

The empirical values are used to suppress dimensional deviations that are **not subject to a trend**.

The empirical and mean values themselves are stored in GUD fields **_EV Empirical value and _MV Mean values**.

_EVNUM specifies the number of the empirical value memory. The number of the mean value memory is defined at the same time via _EVNUM. The number of empirical and mean values is specified in the GUD field **_EVMVNUM**. The unit of measurement is mm in the metric basic system and inch in the inch basic system, irrespective of the active system of units.

Parameters

Values of _EVNUM
The following values can be set:

- = 0 Without empirical value, without mean value memory
- > 0 Empirical value memory number = mean value memory number

If _EVNUM is defined as < 9999, the first 4 digits of _EVNUM are interpreted as the mean value memory number and the second 4 digits as the empirical value memory number.

Example:

_EVNUM = 90012

⇒ EV memory: 12

⇒ MV memory: 9

Further notes

The corresponding field index in field _EV = 11 and in field _MV = 8.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

2.3.12 Multiple measurement at the same location: _NMSP

Parameters

Parameter **_NMSP** can be used to determine the number of measurements at the same location.

The actual/setpoint value difference D is determined arithmetically.

$$D = \frac{S_1 + S_2 + \ldots S_n}{n}$$

n...number of measurements

2.3.13 Weighting factor k for averaging: _K

Function

The weighting factor k can be applied to allow different weighting to be given to an individual measurement.

A new measurement result thus has only a limited effect on the new tool offset as a function of _K.
A detailed description is given in Section 1.7 "Measuring strategy and compensation value definition".

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840Di

2.4. Description of output parameters

Function

In the same way as their defining parameters, the measuring cycle results are Global User Data of the module GUD5.

In this case, the results are not stored as individual data, but in two fields of the **REAL** (_OVR) and **INTEGER** (_OVI) types.

2.4.1 Measuring cycle results in _OVR

Function

The field _OVR[32] contains the following values:

- Setpoints and actual values for abscissa, ordinate and applicate
- Lower and upper tolerance limits for the three axes
- Setpoint/actual value differences in abscissa, ordinate and applicate
- Safe area
- Dimensional difference
- Empirical value.

The results are described individually with the relevant measuring cycles or measurement variants.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

2.4.2 Measuring cycle results in _OVI

Function

The field _OVI[10] contains the following values:

- D or ZO number
- Machining plane
- Measuring cycle number
- Measurement variants
- Weighting factor
- Probe number
- Mean value memory number
- Empirical value memory number
- Tool number
- Alarm number.

The results are described individually with the measuring cycles.

■

2.4. Description of output parameters

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840Di

Notes

Measuring Cycle Auxiliary Programs

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840Di

3.1 Package structure of measuring cycles

The machine data configuration and the software package version determine which programs can be used. It is also possible to partially define these programs in the global cycle data during start-up.

(Please refer to data supplied by the machine manufacturer and Installation and Start-up Guide.)

Function

The measuring cycle package supplied consists of:

- Data blocks for defining the global measuring cycle data,
- measuring cycles,
- measuring cycle subroutines and
- easy-to-use functions.

To ensure that the measuring cycles can be executed in the control, the data blocks must have been loaded into directory "Definitions" and the measuring cycles and measuring cycle subroutines must be stored in the part program memory.

Please note that the control always requires a Power ON between loading and execution of the measuring cycles!

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840Di

3.2 Measuring cycle subroutines

Function

These measuring cycle subroutines are called directly by the cycles. With the exception of CYCLE116, these subroutines cannot be executed through a direct call.

Programming

Cycle	*Function*	*As from SW 4*	*As from SW 4.5*	*In SW 6.2 and higher*
CYCLE100	Activate logging	X		
CYCLE101	Deactivate logging	X		
CYCLE102	Measured result display			
CYCLE103	Parameter setting in interactive mode			
CYCLE104	Internal subroutine: measuring cycle interface			
CYCLE105	Internal subroutine: logging	X		
CYCLE106	Internal subroutine: logging	X		
CYCLE107	Output of measuring cycle messages			
CYCLE108	Output of measuring cycle alarms			
CYCLE109	Internal subroutine: data transfer		X	
CYCLE110	Internal subroutine: plausibility checks			
CYCLE111	Internal subroutine: measuring functions			
CYCLE112	Internal subroutine: measuring functions			
CYCLE113	Internal subroutine: logging	X		
CYCLE114	Internal subroutine: load ZO memory, load WCS wear			
	Internal subroutine: Load WCS wear			X
CYCLE115	Internal subroutine: Load ZO memory			X
CYCLE116	Calculation of the center point and radius on a circle			
CYCLE117	Internal subroutine: measuring functions			
CYCLE118	Internal subroutine: logging	X		

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840Di

3.2.1 CYCLE103: Parameter definition for measuring cycles

Explanation

This auxiliary cycle controls an input dialog for assigning parameters for the measuring cycles.

It can be either directly selected and started or written in the program before the actual measuring cycle is called.

Several input screen forms are displayed one after the other during the course of this dialog. After entering the values, each display must be concluded with the OK key.

The input values for selecting the measuring cycle and the measurement variant are checked for plausibility. As of measuring cycles SW 4.5, CYCLE103 is no longer supported or developed further. Instead, use the cycle support for measuring cycles to supply the parameter data. Please refer to Chapter 7.2 for a detailed description.

Programming

CYCLE103

Programming example

Calibrate tool probe

CALIBRATION_IN_X_Y	
N10 G54 G17 G0 X100 Y80	Position probe at the center of the hole and select ZO
N15 T9 D1 Z10	Select tool length compensation, position probe in the hole
N20 CYCLE103	The operator can assign the parameters for calibration cycle CYCLE976 in interactive mode
N25 CYCLE976	Measuring cycle call for calibr. in X-Y plane
N50 M30	End of program

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840Di

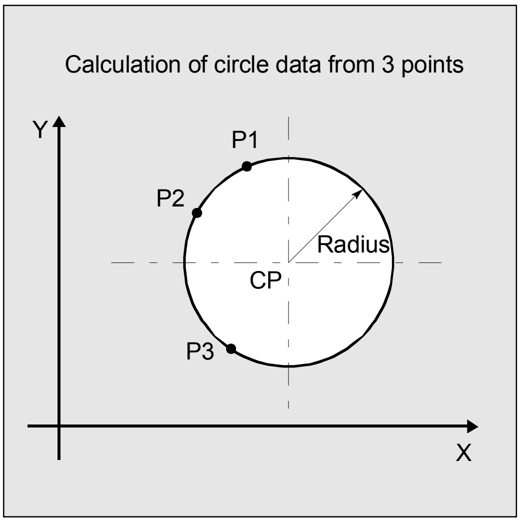
3.2.2 CYCLE116: Calculation of center point and radius of a circle

Explanation

This cycle calculates from three or four points positioned on one plane the circle they inscribe with center point and radius.

To allow this cycle to be used as universally as possible, its data are transferred via a parameter list.

A field of REAL variables of length 13 must be transferred as the parameter.



Programming

CYCLE116 (_DATE, _ALM)

Parameters

Input data

_DATE [0]	Number of points for calculation (3 or 4)
_DATE [1]	Abscissa of first point
_DATE [2]	Ordinate of first point
_DATE [3]	Abscissa of second point
_DATE [4]	Ordinate of second point
_DATE [5]	Abscissa of third point
_DATE [6]	Ordinate of third point
_DATE [7]	Abscissa of fourth point
_DATE [8]	Ordinate of fourth point

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840Di

Output data

The results of the calculation are stored in the last four elements of the same field:

_DATE [9]	Abscissa of circle center point
_DATE [10]	Ordinate of circle center point
_DATE [11]	Circle radius
_DATE [12]	Status for calculation 0 Calculation in progress 1 Error occurred
_ALM	Error number (61316 or 61317 possible)

This cycle is called as a subroutine by measuring cycle CYCLE979.

Example:

Circle.MPF		
`DEF INT _ALM`		
`DEF REAL _DATE[13]= (3,0,10,-10,0,0,-10, 0,0,0,0,0,0)`	; 3 points specified	P1:0,10 P2: -10,0 P3: 0,-10
`CYCLE116(_DATE, _ALM)`	; Result	_DATE[9]=0
`M0`		_DATE[10]=0
`STOPRE`		_DATE[11]=10
`M30`		_DATE[12]=0
		_ALM=0

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840Di

3.3 Measuring cycle user programs

Function

These measuring cycle user programs are called directly by the measuring cycles and can be used to program necessary adaptations before or after a measurement.

3.3.1 CYCLE198: User program prior to calling measuring cycle

Explanation

This cycle is called at the start of each measuring cycle. It can be used to program necessary adaptations prior to starting measurement (e. g. activate probe, position spindle).
As delivered, this cycle contains only one CASE instruction for a jump to a marker that corresponds to the measuring cycle called, followed by command M17.

e. g.:	**_M977**:	prior to call CYCLE**977**
	M17	End of cycle

The user can program the necessary machine adaptations here.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840Di

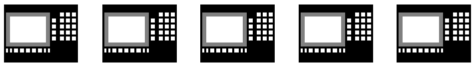
3.3.2 CYCLE199: User program at the end of a measuring cycle

Explanation

This cycle is called at the end of each measuring cycle. It can be used to program necessary actions following completion of a measurement (e. g. deactivate probe).
As delivered, this cycle (just like CYCLE198) contains only one CASE instruction for a jump to a marker that corresponds to the measuring cycle called, followed by command M17.

e. g.: **_M971**: at the end of the CYCLE**971**
M17 End of cycle

The user can program the necessary machine adaptations here.



840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840Di

3.4 Subpackages

Explanation

In many application cases not all the measuring cycles are used on one machine, instead part packages are used.

The following overview shows which part packages are advisable and executable. This allows you to save memory capacity.

Basic package

Milling measuring cycles

Calibrate tool probe + measure tool	Calibrate workpiece probe + measure tool + write ZO or TC	
CYCLE971 CYCLE107 CYCLE108 CYCLE109 CYCLE110 CYCLE111 CYCLE198 CYCLE199	 CYCLE961 CYCLE976 CYCLE977 CYCLE978 CYCLE979 CYCLE998	CYCLE107 CYCLE108 CYCLE109 CYCLE110 CYCLE111 CYCLE112 CYCLE114 CYCLE116 CYCLE198 CYCLE199

Basic package

Turning measuring cycles

Measure tool + calibrate tool probe	Calibrate workpiece probe + measure workpiece + write ZO or TC	
CYCLE972 CYCLE982 CYCLE107 CYCLE108 CYCLE109 CYCLE110 CYCLE111 CYCLE198 CYCLE199	CYCLE973 CYCLE974 CYCLE994	CYCLE107 CYCLE108 CYCLE109 CYCLE110 CYCLE111 CYCLE114 CYCLE117 CYCLE198 CYCLE199

\+

Additional package
Measuring at milling machine in JOG mode

Semi-automatic calibration of tool probe + measure tool		Semi-automatic calibration of tool probe, calculation and setting of reference points
E_MS_CAL E_MS_CAN E_MS_HOL	E_MS_PIN E_MT_CAL E_MT_LEN E_MT_RAD	CYC_JM CYC_JMC

Additional package
Operator interface

Measurement result display selection

Activation: _CHBIT[10]=1

CYCLE102
CYCLE104

Input dialog

Activation: Call in user NC program

CYCLE103
CYCLE104

\+

Additional package
Operator interface

Log in a file of the part program memory

Precondition:
SW 4.3 in NCK

Activation: CYCLE100

CYCLE100
CYCLE101
CYCLE105
CYCLE106
CYCLE113
CYCLE118

840 D
NCU 571

840 D
NCU 572
NCU 573

FM-NC

810 D

840Di

Notes

Measuring in JOG

840 D
NCU 572
NCU 573

810D

840Di

4.1 General preconditions

Certain preconditions must be fulfilled before measuring in JOG can be used.
These conditions are described in greater detail in Part 2 Description of Functions (from Chapter 8 onwards).

The following checklist is useful in determining whether all such preconditions are fulfilled:

Machine

- All machine axes are designed in accordance with DIN 66217.
- A touch-trigger probe (3D) is provided for acquiring workpiece dimensions, and a touch-trigger tool probe for acquiring tool dimensions.
  (see also Section 1.4 Suitable probe types)
- The reference points have been approached.

Control

- 840D as of NCU 572 with SW 5.3 and higher,
  810D SW 3.3 and higher
  MMC103 SW 5.3 and higher

Machine data for running machine cycles:

- All machine data listed in Section 10.1 meet the minimum requirements for running measuring cycles.

Machine data for measuring in JOG

- Machine data
  – MD 11602: ASUB_START_MASK
  – MD 11604: ASUB_START_PRIO_LEVEL
  – MD 20110: RESET_MODE_MASK
  – MD 20112: START_MODE_MASK
  are set as specified in the detailed function description (see Subsection 10.3.1).

  Notice: Interrupt number 8 is used to start the ASUBs for measuring in JOG and must therefore not be used by the user.

Availability of measuring cycles

- The data blocks:
  GUD5.DEF and
  GUD6.DEF
  in directory DEFINE on diskette 1 have been loaded in the control (directory "Definitions" in the file system) and
- the measuring cycles in directory CYCLES on diskette 1 have been loaded into the standard cycle directory of the control and then a power-on executed.

Availability of JOG measuring files

- All files in directory JOG_MESS on diskette 2 have been loaded into the control via "Data in" and a power-on then executed.

Adaptation of data block GUD7.DEF:
Data block GUD7.DEF has been adapted to the requirements of measuring in JOG as specified in the detailed function description (see Subsection 10.3.1).

Function

MEASURING IN JOG comprises the following functions:

- Semi-automatic calculation of tool lengths and storage in tool offset memory.
- Semi-automatic calculation and setting of reference points and storage in zero offset memory.

The functions are operated with softkeys and input displays. The measuring operation is canceled with RESET.

Notice
Make sure that you select the correct channel, as the function MEASURING IN JOG operates channel dependently. Selecting the wrong channel when the measuring operation is active could destroy the measuring probe.

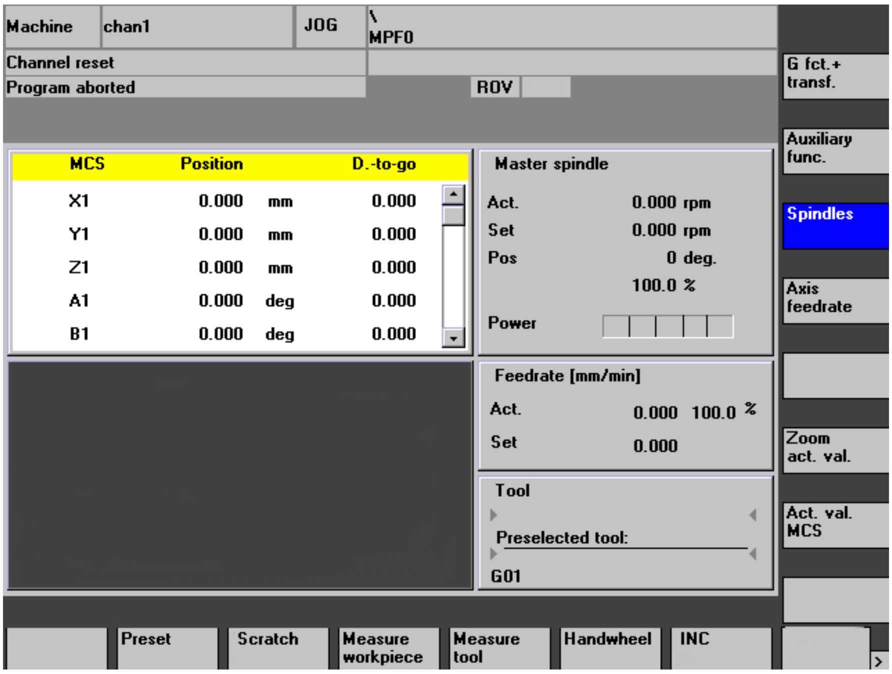
The measuring function is selected via the softkey bar in the JOG basic display.

4.1 General preconditions

840 D
NCU 572
NCU 573

810D

840Di



Measure workpiece

For calculating and setting reference points.

Measure tool

For measuring milling and drilling tools.

840 D
NCU 572
NCU 573

810D

840Di

4.2 Workpiece measurement

Function

With this function you can set reference points on the workpiece using a workpiece probe on the machine.

You call a measuring cycle to set up a workpiece that is clamped on the table. This measuring cycle automatically generates the measurement paths and intermediate positions as a function of the specified setpoints. While the measuring cycle is running, the basic offset defined via GUD6 or a settable ZO, as well as a further working plane G17...G19 set in GUD6 data are effective. The GUD6 data also specifies which data field is assigned to the measuring probe in the spindle and the measuring probe type (multiprobe or monoprobe) (the parameters for switching behavior found by calibrating the measuring probe are also stored in this data field).
All the measuring points required for the measurement task are approached. Prepositioning can either be performed manually or in a program.

When measurement is complete, the result (corner, center point of hole/spigot, edge) is automatically calculated in the measuring cycle according to the type of measurement, and the reference point is set with reference to the basic frame or a settable zero offset according to the selection made by correcting the zero offset memory in question. If "Off" is selected, no correction is made.

Precondition

- The workpiece probe is located in the spindle and has been calibrated.

840 D
NCU 572
NCU 573

810D

840Di

4.2.1 Operation and function sequence of workpiece measurement

Procedure

1. The workpiece is clamped, the probe is positioned in the spindle and calibrated.
2. When you press softkey "Measure workpiece", the following softkey bar is displayed for selection:

Edge >
Corner >
Hole >
Spigot >
Calibrate probe
<< back

3.
- Select zero offset to which the defined setpoint position refers and for which the offset is to apply:
  – Basic frame
  – Settable zero offset G54...
- Enter setpoints if necessary (e.g. approx. diameter of hole/spigot).
- Select the setpoint position in the measuring axis (for edge), the center point (for hole/spigot) or the corner point.
- Select axis and axis direction for edge/corner.

4. On "NC Start", the measuring operation is performed with a measuring feedrate set in the measuring cycle data (GUD6).
   The measuring probe is triggered. When a corner or edge is measured, the probe is automatically retracted in rapid traverse to its starting position for the measuring point in question. When a hole or spigot is measured, all four points are automatically scanned one after the other.
   The translation offset and also an offset for the rotation around the infeed axis in relation to the corner defined for the selected zero offset is determined on the basis of the measuring results and the specified setpoint position.
   When the basic frame is selected, the last channel-specific basic frame is always taken if more than one is available.

840 D
NCU 572
NCU 573

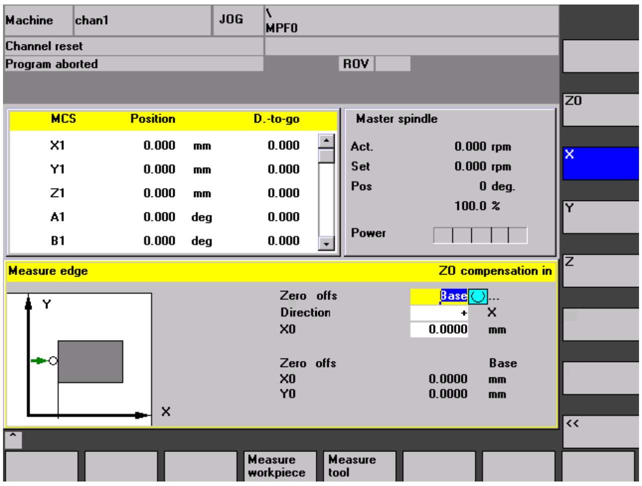
810D

840Di

4.2.2 Measuring an edge

Function

If "Measure edge" is selected, a reference point can be set in any axis of the working plane (G17...G19) defined in a GUD6 data.



Sequence of operations

Precondition

The measuring probe is located in the spindle and has been calibrated.

Approach the workpiece

Position the probe in the required axis direction in front of the workpiece, e.g. in the +X direction.

Select the function with softkey

Measure workpiece	Edge	X	...	Z

Enter details in input form

- Select the zero offset to which the specified setpoint position refers and for which the offset is to apply:
  – Basic frame
  – or zero offset taken from the list of zero offsets
- **Direction:** Set the sampling direction of the selected axis for which the reference point has been set, e.g. +X.
- Enter **set position** of the reference point (edge).

Set the feedrate override switch to the same value as for calibration!

840 D
NCU 572
NCU 573

810D

840Di

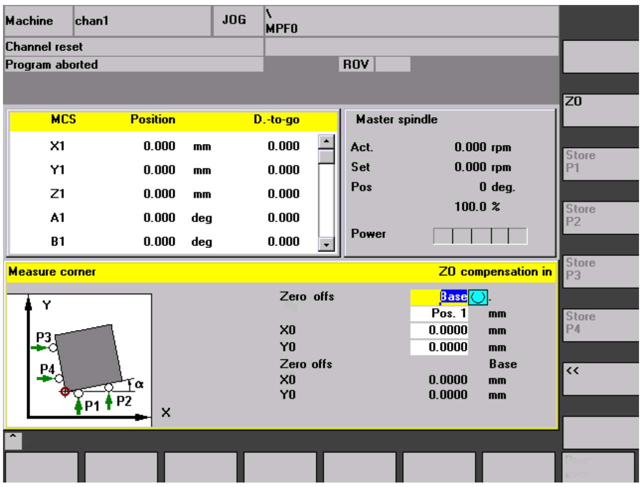
On "NC Start", the measuring operation is automatically performed with a measuring feedrate set via GUD data.

- The measuring probe is triggered.
- Automatic retraction to starting position in rapid traverse.
- The translation offset for the selected zero offset is determined on the basis of the measuring results and the specified setpoint position. On selection of the basic frame the offset is always implemented in the last channel-specific basic frame, if there are more than one. The offset is implemented in the coarse offset and any fine offset there may be is reset.

4.2.3 Measuring a corner

Function

With the selection "Corner", the corner of a workpiece can be measured as the reference point. The probe is positioned at a selected corner of the workpiece.



Sequence of operations

Precondition

The measuring probe is located in the spindle and has been calibrated.

Approach the workpiece

Position the probe at a selected corner of the workpiece.

840 D
NCU 572
NCU 573

810D

840Di

Select the function with softkey

Measure workpiece	Corner

Enter details in input form

- Select the zero offset to which the specified setpoint position for the corner refers and for which the offset is to apply:
  – Basic frame
  – or zero offset taken from the list of zero offsets
- **Position:** Set the corner to be used as the reference point.
- Enter **set position** of the reference point (corner).

Approach sampling point

Position the probe at the first sampling point **P1** of the workpiece edge.

Set the feedrate override switch to the same value as for calibration!

On "NC Start", the measuring operation is automatically performed with a measuring feedrate set via GUD data.

- The measuring probe is triggered.
- Automatic retraction to starting position in rapid traverse.

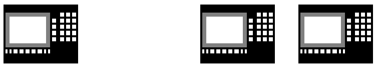
Store the position values of sampling point **P1** by pressing softkey "Save P1". Repeat the procedure "approach sampling points" for sampling points **P2...P4** in the same way.

Calculate corner

Press softkey "Calculate corner" to calculate the translation offset and the rotational offset around the infeed axis for the selected zero offset. On selection of the basic frame the offset is always implemented in the last channel-specific basic frame, if there are more than one. The offset is implemented in the coarse offset and any fine offset there may be is reset.

- The order in which sampling points P1...P4 are approached must be maintained.

On a rectangular workpiece, three sampling points are sufficient for the calculation.



840 D
NCU 572
NCU 573

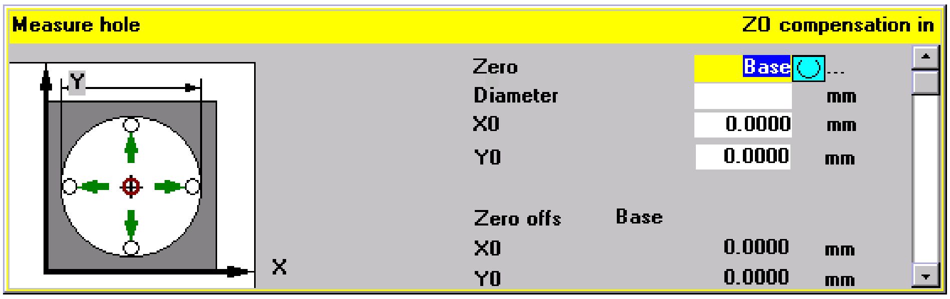
810D

840Di

4.2.4 Measuring a hole

Function

With "Hole", you can set the center of a hole as the reference point. The probe is approximately positioned at the center of the hole and measuring depth.



Sequence of operations

Precondition

The measuring probe is located in the spindle and has been calibrated.

Approach the workpiece

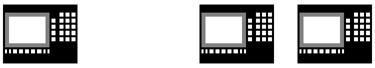
Position the probe approximately in the center of the hole.

Select the function with softkey

Measure workpiece	Hole

Enter details in input form

- Select the zero offset to which the specified setpoint position for the center of the hole refers and for which the offset is to apply:
  – Basic frame
  – or zero offset taken from the list of zero offsets
- **Diameter:** Enter approximate diameter of the hole. If no diameter is entered, sampling is started from the starting point at measurement feedrate.
- Enter **set position** of the hole center.



840 D
NCU 572
NCU 573

810D

840Di

Set the feedrate override switch to the same value as for calibration!

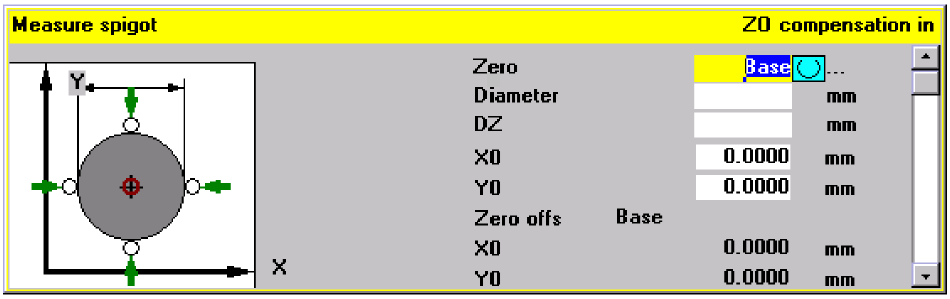
Measurement is performed automatically as soon as you press "NC Start". One after the other, the probe samples four points on the inner surface of the hole.

Once the measurement is complete, the translation offset is determined for the selected zero offset. On selection of the basic frame the offset is always implemented in the last channel-specific basic frame, if there are more than one. The offset is implemented in the coarse offset and any fine offset there may be is reset.

4.2.5 Measuring a spigot

Function

With "Spigot", you can set the center of a spigot (shaft) as the reference point.
The probe is approximately positioned above the center of the spigot.



Sequence of operations

Precondition
The measuring probe is located in the spindle and has been calibrated.

Approach the workpiece
Position the probe approximately above the center of the spigot.

Select the function with softkey

Measure workpiece	Spigot

840 D
NCU 572
NCU 573

810D

840Di

Enter details in input form

- Select the zero offset to which the specified setpoint position for the center of the spigot refers and for which the offset is to apply:
  – Basic frame
  – or zero offset taken from the list of zero offsets
- **Diameter:** Specify the approximate spigot diameter (check diameter>0, safety clearance, include probe offsets).
- **Specify set position** of the center of the spigot.
- Enter **measurement infeed**.

Set the feedrate override switch to the same value as for calibration!

Measurement is performed automatically as soon as you press "NC Start". One after the other, the probe samples four points on the outside of the spigot.

Once the measurement is complete, the translation offset is determined for the selected zero offset. On selection of the basic frame the offset is always implemented in the last channel-specific basic frame, if there are more than one. The offset is implemented in the coarse offset and any fine offset there may be is reset.

4.2.6 Calibrating the measuring probe

Function

With milling machines and machining centers, the probe is usually loaded into the spindle from a tool magazine. This may result in errors when further measurements are taken on account of probe clamping tolerances in the spindle.
In addition, the trigger point must be precisely determined in relation to the spindle center. This is performed by the calibration cycle with which it is possible to calibrate the measuring probe either in any hole or on a surface.
The type of calibration is selected with softkeys "Length" and "Radius".

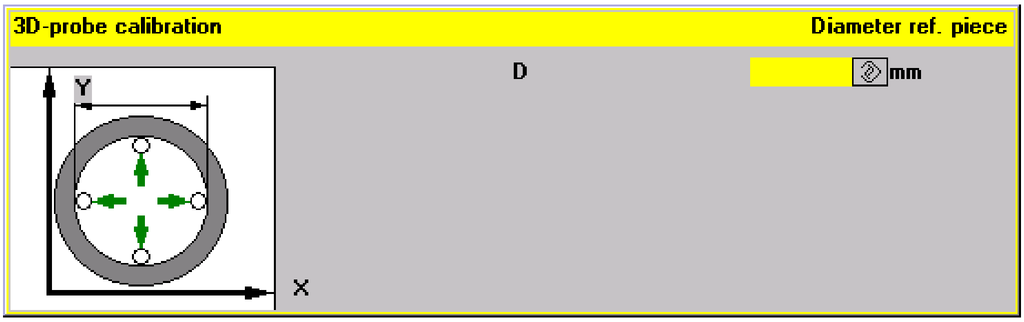
840 D
NCU 572
NCU 573

810D

840Di

Calibrating the workpiece probe in any hole (radius)

With this cycle, the probe can be calibrated in any hole of a reference part, e.g. on a workpiece or in an adjustment ring. The resulting trigger points are automatically loaded in the corresponding data storage area of the GUD6 block.



Sequence of operations

Precondition

The measuring probe is located in the spindle. The precise radius of the probe ball must be entered in the tool offset block.
An adjustment ring with a known radius, for example, is used for calibration.

Approaching the reference part

The probe is approximately positioned at the center of and at the calibration depth of the hole.

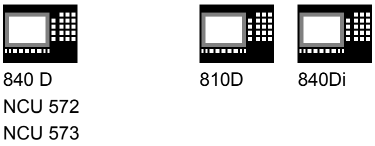
Select the function with softkey

Measure workpiece	Calibrate probe >	Radius

Enter details in input form

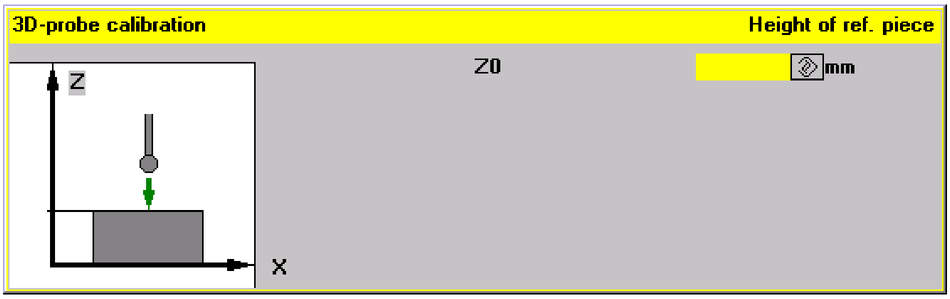
Enter diameter ∅ of the reference part (here: adjustment ring).

Calibration is performed automatically as soon as you press "NC Start". First, the precise position of the center of the adjustment ring is calculated. Then, four trigger points inside the adjustment ring are sampled one after the other.



Calibrating a workpiece probe on any surface

With this measuring cycle you can calibrate the probe on a random surface, e.g. on the workpiece, to determine the length.



Sequence of operations

Precondition
The measuring probe is located in the spindle. The precise radius of the probe ball must be entered in the tool offset block.

Approach the workpiece
The probe must be positioned opposite the calibration surface of the workpiece.

Select the function with softkey

Measure workpiece	Calibrate probe >	Length

Enter details in input form
Known reference Z_0 of the machine table relative to the active zero offset set by GUD6 during measurement.

Calibration is performed automatically as soon as you press "NC Start". The measuring probe is triggered.

The calculated length of the probe is written to the tool offset data block.

840 D
NCU 572
NCU 573

810D

840Di

4.3 Tool measurement

Function

Tools can be measured in the machine with this function.
The tool lengths are automatically written to a tool offset memory and are therefore immediately available for workpiece machining directly after measurement.

General preconditions

- The reference points have been approached.
- The tool measuring probe is swung in or inserted.
- The tool probe has been calibrated.
- The tool to be measured is located in the spindle.
- The tool geometry data (length and radius) have been entered in the tool offset data block as approximate values.
- The tool must be prepositioned in such a way that collision-free approach to the tool measuring probe is possible.

4.3.1 Operation and function sequence of tool measurement

Procedure

1. The tool is replaced or inserted manually.
2. When you press softkey "Measure tool", the following selection appears on the softkey bar:

Length >	Diameter >	Calibrate probe	<< back

3. Enter the measurement type and enter the values in the input form.
4. Position the tool near the tool measuring probe with the JOG direction keys.
5. Start the measuring procedure with "NC-Start".

840 D
NCU 572
NCU 573

810D

840Di

4.3.2 Tool measurement

Function

In tool measurement with a tool measuring probe (table probe system) either the radius or the length of a tool can be measured.

Sequence of operations

Precondition

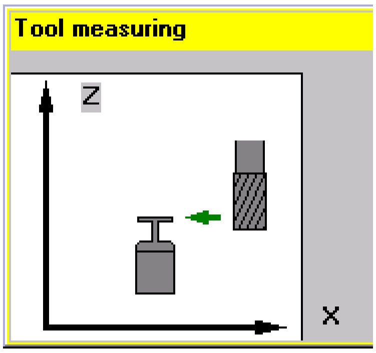
- The tool probe is calibrated.
- The tool geometry data (length and radius/diameter) have been entered in the tool offset data block of the tool list as approximate values.
- The tool to be measured is located in the spindle.
- The data of the tool measuring probe (active width/diameter for length/radius measurement, distance between tool lower edge and tool probe upper edge, permissible axis directions) must be entered in the relevant GUD7 data.

Approaching the tool measurement probe

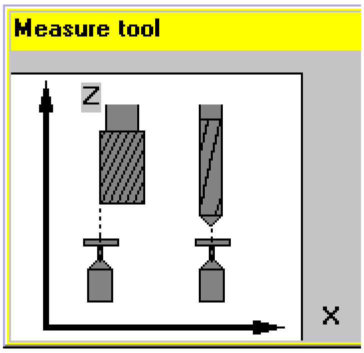
Position the tool near the measuring surface of the tool probe.
Select whether the radius/diameter or the length of the tool is to be measured.

Select the function with softkey

Measure tool	Diameter	or	Length



Radius/diameter



Length

840 D
NCU 572
NCU 573

810D 840Di

Enter details in input form

- Enter length offset **V** (positive value), required, for example, for milling with ballhead cutters or mills with tool inserts.

Measurement is performed automatically as soon as you press "NC Start".
The tool geometry data radius and length are calculated and written to the tool list.

4.3.3 Calibrating the tool measuring probe

Function

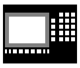
Mechanical tool measuring probes are typically shaped like a cube or a cylindrical disk. The probe is fixed in the machining range of the machine (on the machine table) and must be aligned relative to the machining axes.

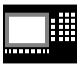
The function "Calibrate tool measuring probe" calculates the current distance between machine zero and the tool measuring probe using the calibration tool and automatically writes them to an internal data storage area. 120 (mill) can be entered as the tool type, there is no special calibration tool type.

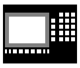
Sequence of operations

Precondition

- The exact length and radius of the calibration tool must be stored in a tool offset data block.
- The calibration tool is located in the spindle.
- The data of the tool measuring probe (active width/diameter for length/radius measurement, distance between tool lower edge and tool probe upper edge, permissible axis directions) must be entered in the relevant GUD7 data.


840 D
NCU 572
NCU 573


810D


840Di

Approaching the tool measuring probe

Traverse the calibration tool approximately to the center of the measuring surface of the tool probe.

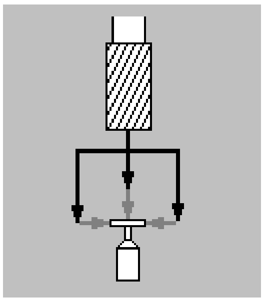
Select the function with softkey

Measure tool	Calibrate probe

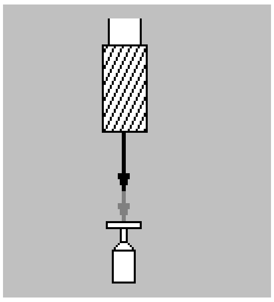
Enter the type of measurement in the input form:

- Compare length only

Compare length and diameter



Compare length and diameter



Compare length only

Calibration at measurement feedrate is performed automatically as soon as you press "NC Start".
The actual distance between machine zero and the tool measurement probe is calculated and stored in an internal data storage area.

■

Measuring Cycles for Milling and Machining Centers

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D

5.1 General preconditions

Function

Measuring cycles are subroutines that have been kept general for solving a certain measuring problem and which are adapted to the specific problem by the input data. The measuring cycles are created as a program package comprising the actual measuring cycles and utilities.

To be able to run the measuring cycles described in this Chapter, the following programs must be stored in the part program memory of the control.

Programming

Overview of the measuring cycles

CYCLE961	Automatic setup inside and outside corner
CYCLE971	Tool measurement for milling tools, calibrate tool probe
CYCLE976	Calibrate workpiece probe in random hole (plane) or on random surface (applicate)
CYCLE977	Paraxial measurement of hole, shaft, groove, web or ZO calculation
CYCLE978	Single-point measurement or ZO calculation on surface
CYCLE979	Measurement of hole, shaft, groove, web of ZO calculation at random angles
CYCLE998	Angular measurement (ZO calculation only)

Overview of the utilities required

CYCLE100	Log ON
CYCLE101	Log OFF
CYCLE102	Measurement result display selection
CYCLE103	Preassignment of input data
CYCLE104	Internal subroutine
CYCLE105	Generate log contents
CYCLE106	Logging the sequential controller
CYCLE107	Output of message texts
CYCLE108	Output of alarm messages
CYCLE110	Internal subroutine
CYCLE111	Internal subroutine
CYCLE112	Internal subroutine
CYCLE113	Read system date and time
CYCLE114	Internal subroutine (tool offset)
CYCLE115	intern subroutine (zero offset, measuring cycle SW 6.2 and higher)
CYCLE116	Calculate circle center point
CYCLE118	Format real values

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

The two data blocks

- **GUD5.DEF**
- **GUD6.DEF**

are needed. All the data required by measuring cycles are defined in these blocks.

Procedure

Call and return conditions

The following general call and return conditions must be observed:

- D compensation containing the probe data must always be activated before the cycle is called (does not apply to tool measurement). Tool type 1x0 or 710 (3D probe) for measuring cycles SW 4 and higher is permitted. No mirroring or scale factors <>1 must be active (up to measuring cycle SW 5.3 and higher).
- As of measuring cycles SW 5.4, workpiece cycles can also be used on turning machines if the following requirements are satisfied:
  - The 3rd geometry axis exists.
  - Probe tool type 500 with tool edge positions 5 to 8
  - Tool length offset is machine-specific (SD TOOL_LENGTH_TYPE=2)
  - With tool edge positions 5 or 7, measurement is carried out in G17 plane; with tool edge positions 6 or 8 in G19 plane.
- Measuring cycles SW 4.4. and higher allows coordinate rotation for the workpiece measuring cycles.
- As of measuring cycles SW 5.4, mirroring of workpiece measuring cycles is permissible, except for calibration (condition: MD 10610=0).
- When using a multidirectional probe the best measurement results are achieved if the probe in the spindle is mechanically aligned during calibration and measurement in such a way that one and the same point on the probe ball, e.g. in the + direction of the abscissa (+X with active G17), is in the active workpiece coordinate system.
- The G functions active before the measuring cycle is called remain active after the measuring cycle call even if they have been changed inside the measuring cycle.
- **Measuring cycles version SW 6.2 and higher can only be used with NCK-SW 6.3 and higher.**

Plane definition

The measuring cycles work internally with the 1st axis (abscissa), 2nd axis (ordinate) and 3rd axis (applicate) of the current plane. Which plane is the current plane is set with G17, G18 or G19 before the measuring cycle is called.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

5.2 CYCLE971 Tool measuring for milling tools

Programming

CYCLE971

Function

Measuring cycle CYCLE971 performs calibration of a tool probe and measures tool lengths and/or radius for milling tools.

Supports the following measuring tasks:

- Measure tool length with motionless and rotating spindle
- Measure tool radius with motionless and rotating spindle
- Calibration of a tool probe

Result parameters

The measuring cycle CYCLE971 returns the following values in the GUD5 module for the measurement variant calibration:

_OVR [8]	REAL	Trigger point in minus direction, actual value, abscissa
_OVR [10]	REAL	Trigger point in plus direction, actual value, abscissa
_OVR [12]	REAL	Trigger point in minus direction, actual value, ordinate
_OVR [14]	REAL	Trigger point in plus direction, actual value, ordinate
_OVR [16]	REAL	Trigger point in minus direction, actual value, applicate
_OVR [18]	REAL	Trigger point in plus direction, actual value, applicate
_OVR [9]	REAL	Trigger point in minus direction, difference, abscissa
_OVR [11]	REAL	Trigger point in plus direction, difference, abscissa
_OVR [13]	REAL	Trigger point in minus direction, difference, ordinate
_OVR [15]	REAL	Trigger point in plus direction, difference, ordinate
_OVR [17]	REAL	Trigger point in minus direction, difference, applicate
_OVR [19]	REAL	Trigger point in plus direction, difference, applicate
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Measuring probe number
_OVI [9]	INTEGER	Alarm number

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Compensation of the tool probe trigger points _TP[x,0...5] is only performed if the measured difference lies in the tolerance band between _TZL and _TSA!

Result parameters

Measuring cycle CYCLE971 returns the following result values in the GUD5 module after tool measurement:

_OVR [8]	REAL	Actual value length L1
_OVR [10]	REAL	Actual value radius R
_OVR [9]	REAL	Difference length L1
_OVR [11]	REAL	Difference radius R
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVR [29]	REAL	Permissible dimension difference
_OVR [30]	REAL	Empirical value
_OVI [0]	INTEGER	D number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Measuring probe number
_OVI [7]	INTEGER	Number of empirical value memory
_OVI [8]	INTEGER	T number
_OVI [9]	INTEGER	Alarm number

Compensation of length 1 or the radius is only performed if the measured difference lies in the tolerance band between _TZL and _TDIF!

Measurement variants

Measuring cycle CYCLE977 permits the following measurement variants which are specified via parameter _MVAR.

Value	*Meaning*
0	Tool probe calibration
1	Measure tool with motionless spindle (length or radius)
2	Measure tool with rotating spindle (length or radius)
10000	Calibrate tool probe incrementally

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

5.2.1 CYCLE971 Measuring strategy

Function

Measure tool

Before the measuring cycle is called, the tool must be prepositioned in such a manner that collision-free approach to the probe is possible. First, the measuring cycle generates traversing paths with a reduced rapid traverse velocity (_SPEED[0]), or with active collision monitoring at the position feedrate set in _SPEED[1] or _SPEED[2].

Measure tool with motionless spindle

With milling tools, measurement through spindle positioning may call for the tool to be rotated such that the measurement is executed on a tool edge. The measurement feedrate is defined by _VMS.

Measure tool with rotating spindle

Typically, measurements of the radius of milling tools are executed with rotating spindle, that is the largest edge determines the measuring result.
A length measurement of milling tools with rotating spindle is advisable if the tool diameter is greater than the wheel diameter valid for the length measurement or edge length of the tool probe.

Points to bear in mind:

- Is the tool probe permissible for measuring with rotating spindle with length and/or radius calculation? (Manufacturer documentation)
- Permissible peripheral speed for the tool to be measured.
- Maximum permissible speed.
- Maximum permissible feedrate for probing.
- Minimum feedrate for probing.
- Selection of the rotation direction depending on the cutting edge geometry with view to preventing hard impacts when probing.
- Required measuring accuracy.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

When measuring with rotating tool the relation between axis feed and spindle speed must be taken into account. Here it is necessary to base the assumptions on a single cutter. (With multiple cutters only the longest edge is used for the measuring result)

The following connections have to be taken into account:

$$n = \frac{S}{2 \cdot \pi \cdot r \cdot 0.001} \qquad F = n \cdot \text{Measuring accuracy}$$

		Basic system	
		metric	inch
n	Speed	rpm	rpm
S	Max. permissible peripheral speed	m/min	feet/min
r	Tool radius	mm	inch
F	Probe feedrate	mm/min	inch/min
	Measuring accuracy	mm	inch

With a grinding wheel surface speed of 90 m/min, milling tools with a radius of between 5 and 100 mm produce speeds between 2865 and 143 rpm. With a specified measuring accuracy of, for example, 0.005 mm, this results in feeds ranging from 14 mm/min to 0.7 mm/min.

Compensation strategy

The tool measuring cycle is provided for various applications:

- Initial measuring of a tool in the machine or
- Subsequent measuring of a tool.

Accordingly, you can either enter the measured value in the parameter for length/radius of tool compensation and delete the corresponding wear data at the same time, or enter the differences to length and radius in the wear data.

Furthermore, for tool measurement, the measured values can be corrected by empirical values.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

5.2.2 CYCLE971 Calibrate tool probe

Function

The cycle uses the calibration tool to ascertain the current distance dimensions between the machine zero and the tool probe trigger points and automatically loads them into the appropriate data area in the GUD6 module. They are always calculated without empirical or mean values.

Precondition

The approximate coordinates of the tool probe regarding the machine zero have to be entered in the data field _TP[_PRNUM-1, 0] to _TP[_PRNUM-1, 5] before starting the calibration.
The exact length and radius of the calibration tool must be stored in a tool offset data block. This tool offset must be active when the measuring cycle is called.
120 can be preset as tool type, there is no separate type of calibration tool.

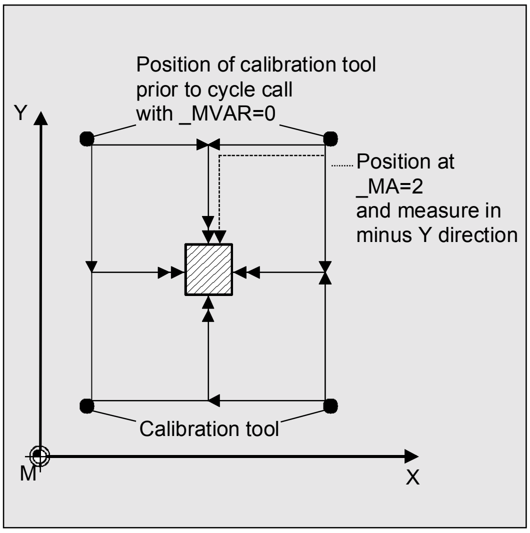
Parameters

_MVAR	0	Tool probe calibration
	10000	Calibrate tool probe incrementally
_MA	1...3	Number of measuring axis
	102...201	Number of the offset and measuring axis (possible for calibration in the plane; not with MVAR=10000) By means of additionally specifying the offset axis, first of all the exact center of the tool probe is detected in the offset axis before calibration takes place in the measuring axis.
_PRNUM	1...3	Number of tool probe
_FA	<0>	Measurement path. For incremental calibration the travel direction is also defined via _FA. _FA > 0 Travel direction + _FA > 0 Travel direction –

These following additional parameters are also valid:
_VMS, _TZL, _TSA and **_NMSP**.
See Sections 2.2 and 2.3.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

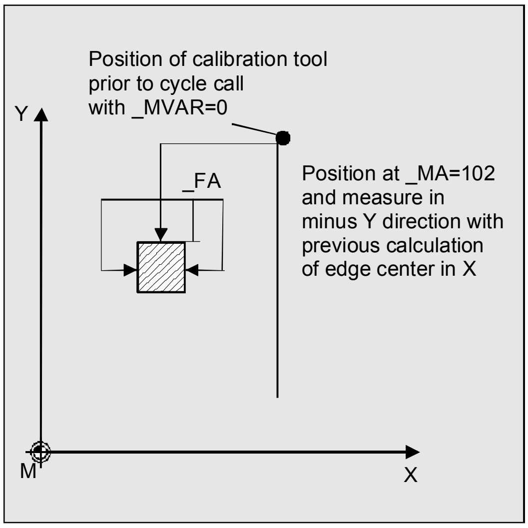
Procedure



Position before the cycle is called

The machining plane must be defined.
The calibration tool must be prepositioned as shown in the figure. The measuring cycle then calculates the approach position itself.

With incremental calibration, there is no generation of traversing movements before the actual measured block. The calibration tool must be positioned at the tool probe such that the calibration tool traverses to the tool probe when the measuring axis and an incremental measuring path (with sign) up to the expected edge are entered.



Position after the cycle has terminated

On completion of the calibration process, the calibration tool is positioned facing the measuring surface at a distance corresponding to _FA · 1 mm.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Programming example

Calibrating the tool probe

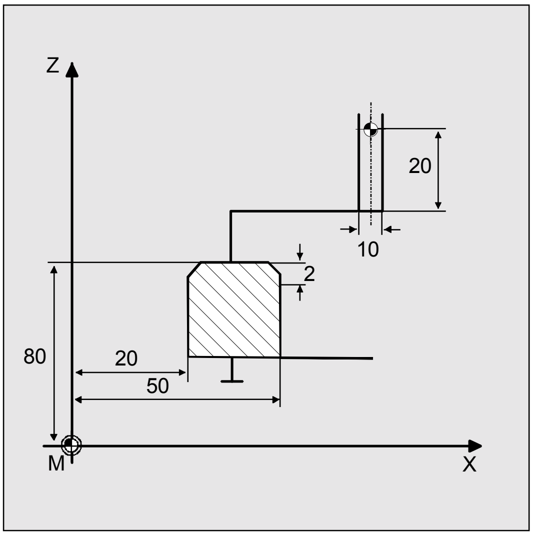
The tool probe is stationary but provides a switching signal. The calibration tool is in the spindle.

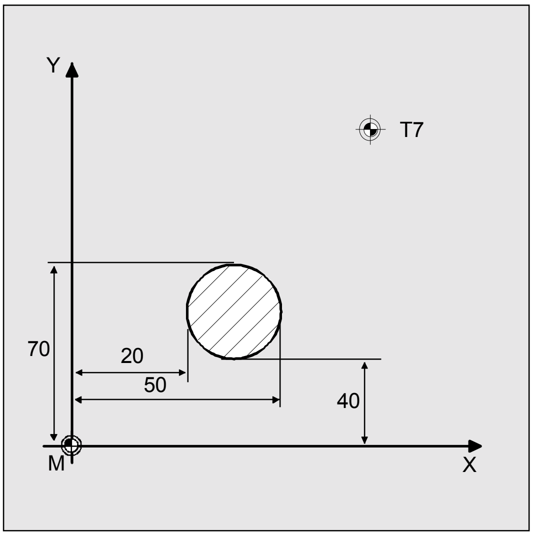
Values of the calibration tool in T7 D1 in this example:

Type 120
L1 20
R 5

Values of the tool probe 1 in module GUD6:

_TP[0,0] = 50
_TP[0,1] = 20
_TP[0,2] = 70
_TP[0,3] = 40
_TP[0,4] = 80
_TP[0,9] = 2





CALIBRATE_TOOL_PROBE	
N05 G0 G17 G94 G54	Define machining plane, zero offset and feed type
N10 T7 D1	Select calibration tool
N15 M6	Change calibration tool
N30 SUPA G0 Z100	Position in infeed axis above tool probe
N35 SUPA X70 Y90	Position in plane at tool probe

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

N40 _MVAR=0 _MA=102 _TZL=0.005 _TSA=5 _PRNUM=1 _VMS=0 _FA=5 _NMSP=1	Parameters for calibrating in the Y axis with detection of probe center in X. The data field of tool probe 1 is active.
N50 CYCLE971	Calibration in minus Y direction
N55 SUPA Z100	Run up in infeed axis in rapid traverse
N60 SUPA Y30 **N65 _MA=2**	In plane, traverse to position from which calibration is possible in plus Y direction
N70 CYCLE971	Calibrate in plus Y direction (probe centered in X)
N80 SUPA X70 Z100	Retract from probe in rapid traverse in X axis and Z axis
N85 _MA=1	Calibration in the X axis
N90 CYCLE971	Calibration in minus X direction
N100 SUPA Y10 Z100	Retract from probe in rapid traverse in Y axis and Z axis
N110 SUPA X10	In X axis, traverse to position from which calibration is possible in plus direction
N120 CYCLE971	Calibration in plus X direction
N130 SUPA Z100	Run up in infeed axis
N140 _MA=3	Calibration in the Z axis
N150 CYCLE971	Calibration in minus Z direction
...	
N160 M2	End of program

The new trigger values in -X, +X, -Y, +Y and -Z are stored in the global data of tool probe 1_TP[0,0...4] if they deviate by more than 0.005 mm from the old values. Deviations of up to 5 mm are permissible.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

5.2.3 CYCLE971 Measure tool

Function

The cycle calculates the new tool length or radius and checks whether the difference from the old tool length or radius, possibly corrected by an empirical value, is within a defined tolerance range (upper limits: safe area _TSA and dimension difference check _TDIF, lower limit: zero offset area _TZL).
If this range is not violated, the new tool length or radius is accepted, otherwise an alarm is output. Violation of the lower limit is not corrected.
Measuring is possible either with

- Motionless spindle
- Rotating spindle

The entry in the current tool offset memory can be optionally as absolute value in the tool offset data or as difference in the wear data.

Precondition

- The tool probe must be calibrated.
- The tool geometry data must be entered in a tool offset data record.
- The tool must be active.
- The desired machining plane must be activated.
- The tool must be prepositioned in such a manner that collision-free approach to the probe is possible in the measuring cycle.

Special features of measurement with rotating spindle

- As standard the cycle-internal calculation of feed and speed is executed from the limit values defined in the data field _CM[] for peripheral speed, rotation speed, minimum feed, maximum feed and measuring accuracy, as well as the intended direction of spindle rotation for measurement.
  Measuring is conducted by probing twice; the first probing action causes a higher feedrate. A maximum of three probing operations are possible for measuring.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

- The operator can deactivate the cycle-internal calculation via the measuring cycle bit _CBIT[12] and specify his or her own values for feed and speed.
- The data field _MFS is for entering the values.
- If the bit is set, the values from _MFS[0/1] are valid for the first probing and the values from _MFS[2/3] for the second. If _MFS[2] = 0 only one probing action is performed. If _MFS[4] > 0 and _MFS[2] > 0, probing is performed in three probing actions; the values from _MFS[4/5] are valid in the third action.
- The monitoring operations from data field _CM[] are not effective!
- If the spindle is motionless when the measuring cycle is called, the direction of rotation is determined from _CM[5].

Monitoring for measuring with rotating spindle and cycle-internal calculation

_CM[0]	Maximum permissible peripheral speed [m/min]/[feet/min] Default: 60 m/min
_CM[1]	Maximum permissible speed for measuring with rotating spindle [rpm] (if it is exceeded, the speed is automatically reduced) Default: 2000 rpm
_CM[2]	Minimum feedrate for probing [mm/min]/[inch/min] (prevents feeds from being too low with large tool radii) Default: 1mm/min
_CM[3]	Required measuring accuracy [mm]/[inch] is effective with the last probing action Default: 0.005 mm
_CM[4]	Maximum feedrate for probing [mm/min]/[inch/min] Default: 20 mm/min
_CM[5]	Direction of spindle rotation during measuring Default: 4 = M4
_CM[6]	Feed factor 1 0: One probing action only with calculated feed ≥1: 1st probing action with calculated feed · Feed factor 1 Default: 10
_CM[7]	Feed factor 2 0: 2nd probing operation with calculated feed (only valid with _CM[6]>0) ≥1: 2nd probing action with calculated feed · Feed factor 2 3rd probing with calculated feed Feed factor 2 should be smaller than feed factor 1. Default: 0

Notice

If the spindle is rotating when the measuring cycle is called, this direction of rotation remains independent of _CM[5]!

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Parameters

_MVAR	1	Measure tool with motionless spindle (Length or radius)
	2	Measure tool with rotating spindle (Length or radius)
_MA		Number of measuring axis
	1	Measuring the radius in direction of the abscissa
	2	Measuring the radius in direction of the ordinate
	3	Measuring the length at center point of the tool probe
	103	Measuring the length, shifted around radius in direction of the abscissa
	203	Measuring the length, shifted around radius in direction of the ordinate
_ID	REAL ≥ 0	Parameter is usually set to 0. With multiple cutters the offset of tool length and the highest point of the tool edge must be specified in _ID for radius measurement; the offset from the tool radius to the highest point of the tool edge must be specified for length measurement.
_MFS[0]	REAL	Speed 1st probing (only with _CBIT[12]=1)
_MFS[1]	REAL	Feed 1st probing
_MFS[2]	REAL	Speed 2nd probing 0: Measurement terminated after 1st probing
_MFS[3]	REAL	Feed 2nd probing
_MFS[4]	REAL	Speed 3rd probing 0: Measurement terminated after 2nd probing
_MFS[5]	REAL	Feed 3rd probing

These following additional parameters are also valid:
_VMS, _CORA, _TZL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM and **_NMSP**.
See Sections 2.2 and 2.3.

Bit 3 in the channel-oriented bits in the measuring cycles is for determining whether the measured value is to be written absolute in length/radius parameters with simultaneous deletion of the corresponding wear data (_CHBIT[3]=0) or the difference is to be written in the wear data (_CHBIT[3]=1).

840 D NCU 571

840 D NCU 572 NCU 573

810 D

Procedure

Position before the cycle is called

Before the cycle is called a start position must be adopted from which it is possible to conduct a collision-free approach to the probe. The measuring cycle then calculates the approach position itself.

Position after the cycle has terminated

On completion of the cycle, the tool nose is positioned facing the measuring surface at a distance corresponding to _FA.

5.2 CYCLE971 Tool measuring for milling tools

840 D NCU 571	840 D NCU 572 NCU 573	810 D

Programming example

Measuring the length and the radius of a T3 drilling tool

MEASURE_T3	
...	
N00 G17 G54 G94	
N05 T3 D1	Selection of the tool to be measured
N10 M6	Change tool
N15 G0 SUPA Z100	Position in infeed axis above the tool probe
N20 _CHBIT[3]=0 _CBIT[12]=0	Offset of tool geometry, cycle-internal calculation of feed and speed for measuring with rotating spindle
N30 _MVAR=1 _MA=3 _TZL=0.04 _TDIF=0.6 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2 _EVNUM=0	Parameters for the cycle
N40 CYCLE971	Measure length with motionless spindle
N50 SUPA X70	In X retracting from probe
N70 _MA=1 _MVAR=2	
N80 CYCLE971	Measure radius in minus X direction with rotating spindle
N90 M2	

The calculated length 1 and radius of the active tool are entered in the geometry memory of the active tool if they deviate by more than 0.04 mm or less than 0.6 mm from the old values.
Values are corrected without empirical values.
The wear memories of the active tool are cleared.

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

5.3 CYCLE976 Calibrate workpiece probe

Programming

CYCLE976

Function

With milling machines and machining centers, the probe is usually loaded into the spindle from a tool magazine.
This may result in errors when further measurements are taken on account of probe clamping tolerances in the spindle.

Moreover, the triggering points in the axis directions that not only depend on the probe tip diameter but also on the mechanical design of the probe and the velocity of contact between the probe and an obstacle must be calculated.

This is permitted by the calibration cycle which makes it possible to calibrate the probe either in a hole (plane) or on a surface (applicate).

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Result parameters

The measuring cycle CYCLE976 supplies the following values as results in the GUD5 module:

_OVR [4]	REAL	Actual value probe ball diameter
_OVR [5]	REAL	Difference probe ball diameter
_OVR [6][1)]	REAL	Center point of the hole in the abscissa
_OVR [7][1)]	REAL	Center point of the hole in the ordinate
_OVR [8]	REAL	Trigger point in minus direction, actual value, abscissa
_OVR [10]	REAL	Trigger point in plus direction, actual value, abscissa
_OVR [12]	REAL	Trigger point in minus direction, actual value, ordinate
_OVR [14]	REAL	Trigger point in plus direction, actual value, ordinate
_OVR [16]	REAL	Trigger point in minus direction, actual value, applicate
_OVR [18]	REAL	Trigger point in plus direction, actual value, applicate
_OVR [9]	REAL	Trigger point in minus direction, difference, abscissa
_OVR [11]	REAL	Trigger point in plus direction, difference, abscissa
_OVR [13]	REAL	Trigger point in minus direction, difference, ordinate
_OVR [15]	REAL	Trigger point in plus direction, difference, ordinate
_OVR [17]	REAL	Trigger point in minus direction, difference, applicate
_OVR [19]	REAL	Trigger point in plus direction, difference, applicate
_OVR [20]	REAL	Positional deviation abscissa
_OVR [21]	REAL	Positional deviation ordinate
_OVR [24]	REAL	Angle at which the trigger points were determined
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVI [2]	INTEGER	Measuring cycle number
_OVI [5]	INTEGER	Measuring probe number
_OVI [9]	INTEGER	Alarm number

1) for calibration variant with unknown drilling center point only

Applicable probe types

The measuring cycle operates with the following probe types which are coded via parameter _PRNUM:

- Multidirectional probe
- Monodirectional probe (bidirectional probe)

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Measurement variants

The measuring cycle CYCLE976 permits the following calibration variants which are specified via parameter _MVAR.

Possible parameter values lie between 0...112101 and are put together as follows:

- Calibrate in random hole (plane)

Digit						*Meaning*
6	**5**	**4**	**3**	**2**	**1**	
0						Paraxial calibration (in the plane)
1						Calibration at any angle (in the plane)
	0					No position calculation
	1					With calculation of position
		0				4 axis directions
		1				1 axis direction (specify measuring axis and axis direction)
		2				2 axis directions (indicate measuring axis)
			0			No calculation of probe ball
			1			Calculation of probe ball (for measurement in plane)
				0		With any data in the plane
					1	Hole (for measurement in the plane), center of the hole known
					8[1)]	Hole (for measurement in the plane), center of the hole not known

- Calibration on any surface (applicate)

Digit						*Meaning*
6	**5**	**4**	**3**	**2**	**1**	
					0	Calibration on random surface
	1	**0**	**0**	**0**	**0**	Calibration on any surface in applicate with calculation of probe length

1) Measuring cycles SW 4.4. and higher

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.3.1 CYCLE976 Calibrate workpiece probe in any hole (plane) with known hole center

Function

This measuring cycle makes it possible to calibrate the probe in the plane in a random hole, e.g. on the workpiece. The calculated trigger points are automatically loaded in module GUD6.DEF if the calculated difference from the stored trigger points lies within the tolerance band between _TZL and _TSA. If _TSA is exceeded an error message is output. Calibration is either paraxial or at a random angle.

Precondition

The probe must be called **with** tool length offset. Tool type 1x0 or 710 (3D probe) for SW 4 and higher is permitted. The center point of the hole and its diameter must be known!

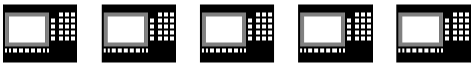
Parameters

_MVAR	See Section 5.3 "Measurement variants"	Definition of calibration variant
_SETVAL	REAL	Calibration setpoint = diameter of hole
_MA	1, 2	Measuring axis (depends on the measurement variant)
_MD	0 positive axis direction 1 negative axis direction	Measuring direction (depends on the measurement variant)
_PRNUM	INT	Measuring probe number
_STA1[1)]	REAL	Starting angle (calibration takes place at this angle)

1) Enter only for calibrating at an angle.

These following additional parameters are also valid:
_VMS, _CORA, _TZL, _TSA, _FA and **_NMSP**.
See Sections 2.2 and 2.3.

Notice!
When calibration is performed for the first time the default setting in the data field of the probe is still "0". For that reason, _TSA> radius probe ball must be programmed to avoid alarm "Safe area violated".



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Procedure

Position before the cycle is called

The probe must be positioned at the center of the hole in the abscissa and the ordinate of the selected measuring plane and at the calibration depth in the hole.

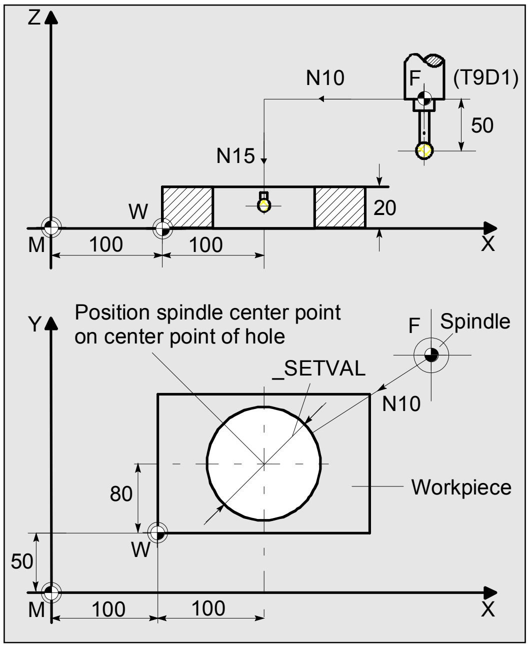
Position after the cycle has terminated

When the calibration procedure is completed the probe is positioned at the center of the hole.

Programming example

Calibration of workpiece probe 3 in the X-Y plane

The radius of the probe ball must be entered in the tool offset memory, e.g. under T9 D1, before the cycle is called.



CALIBRATE_IN_X_Y	
N10 G54 G17 G0 X100 Y80	Probe to center point and select ZO
N15 T9 D1 Z10	Select length compensation, position probe in hole
N20 _MVAR=10101 _SETVAL=100 _TSA=1 _PRNUM=3 _VMS=0 _NMSP=1 _FA=1 _TZL=0	Define parameters for calibrating cycle (calibration in 4 axis directions with position calculation and calculation of probe ball)
N25 CYCLE976	Measuring cycle call for calibration in X-Y plane
N50 M30	End of program

The new trigger values in -X, +X, -Y and +Y are stored in the global data of measuring probe 3 _WP[2,1...4]. The positional deviation calculated in the X and Y direction is stored in _WP[2,7...8], the active probe ball diameter in _WP[2,0].

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.3.2 CYCLE976 Calibrate workpiece probe in any hole (plane) with unknown hole center (measuring cycles SW 4.4 and higher)

Function

With this measuring cycle it is possible to calibrate the probe in any hole whose precise center point is not known.
With this measuring version, first the center and positional deviation (skew) is determined and then all the trigger points in all four axis directions of the plane. The measuring cycle also places the derived center point of the hole in OVR fields 6 and 7.

Precondition

- The probe must be called **with** tool length offset. Permissible tool types:
  - 1x0 or in measuring cycle SW 4 and higher also 710 (3D probe
  - in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.
- The exact diameter of the hole must be known.
- The spindle must be SPOS capable.
- Probe in spindle can be positioned 0...360 degrees (all-round coverage).

Parameters

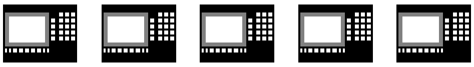
_MVAR	8...10108	Calibration in hole, center unknown
_SETVAL	REAL	Calibration setpoint = diameter of hole
_PRNUM	INT	Measuring probe number

These following additional parameters are also valid:
_VMS, _CORA, _TZL, _TSA, _FA and **_NMSP**.
See Sections 2.2 and 2.3.

Notice!
When calibration is performed for the first time the default setting in the data field of the probe is still "0".
For that reason, _TSA> radius probe ball must be programmed to avoid alarm "Safe area violated".

Procedure

Position before the cycle is called
The probe must be positioned near the hole center in the abscissa and the ordinate of the selected measuring plane and at the calibration depth in the hole.



Position after the cycle has terminated

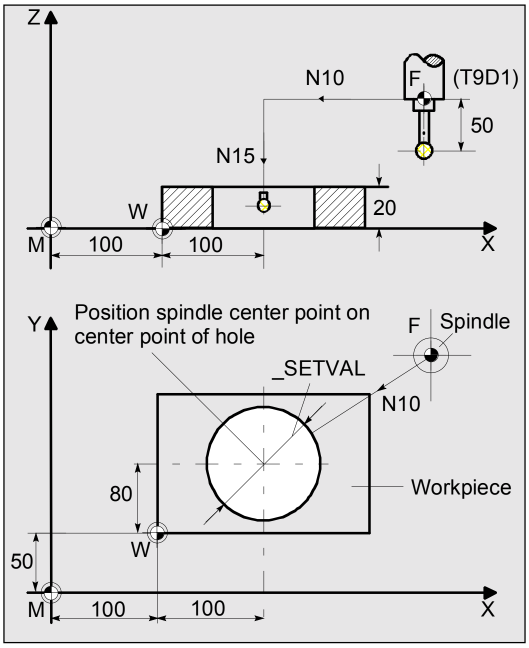
When the calibration procedure is completed the probe is positioned at the center of the hole.

If using probes whose triggering behavior differs greatly depending on the type of deflection or if a high degree of precision is required, the calibration procedure should be repeated.

Programming example

Calibration of workpiece probe 3 in the X-Y plane

The radius of the probe ball must be entered in the tool offset memory, e.g. under T9 D1, before the cycle is called.



CALIBRATE_IN_X_Y	
N10 G54 G17 G0 X100 Y80	Position probe in hole and select ZO
N15 T9 D1 Z10	Select length compensation, position probe in hole
N20 _MVAR=10108 _SETVAL=100 _TSA=1 _PRNUM=3 _VMS=0 _NMSP=1 _FA=_SETVAL/2 _TZL=0	Define parameters for calibration cycle (calibration in 4 axis directions with position calculation)
N25 CYCLE976	Measuring cycle call for calibration in X-Y plane
N50 M30	End of program

The hole center is determined twice, the spindle being rotated by 180° in-between times if a multiprobe is used, in order to record any positional deviation of the measuring probe (skew). Triggering is then determined accurately in all 4 axis directions.

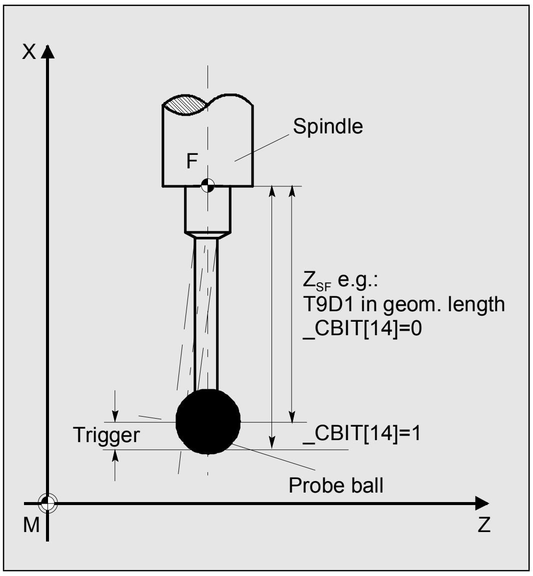
The new trigger values in -X, +X, -Y and +Y are stored in the global data of probe 3_WP[2,1...4], the positional deviation in the X and Y direction in _WP[2,7...8].

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

5.3.3 CYCLE976 Calibrate workpiece probe on a random surface

Function

With this measuring cycle you can calibrate the probe on a random surface, e.g. on the workpiece, in order to determine the trigger point in the axis and axis direction concerned.



Precondition

The probe must be called **with** tool length offset.
Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

When used with turning machines, the setting _CBIT[14]=0 must be made.

Parameters

_MVAR	0	Calibration variant: Calibration on any surface
_SETVAL	REAL	Calibration setpoint
_MA	1, 2 or 3	Measuring axis
_MD	0 positive axis direction 1 negative axis direction	Measurement direction
_PRNUM	INT	Measuring probe number

These following additional parameters are also valid:
_VMS, _CORA, _TZL, _TSA, _FA and **_NMSP**.

See Sections 2.2 and 2.3.

Notice!
When calibration is performed for the first time the default setting in the data field of the probe is still "0". For that reason, _TSA> radius probe ball must be programmed to avoid alarm "Safe area violated".

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Procedure

Position before the cycle is called

The probe must be positioned facing the calibration surface.

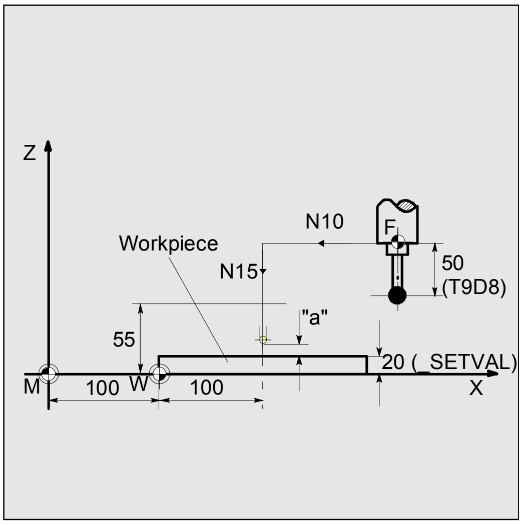
Position after the cycle has terminated

On completion of the calibration process, the probe is positioned above the calibration surface at a distance corresponding to "a".

Programming example

Calibration of workpiece probe 1 in the Z axis on the workpiece

The radius of the probe ball and the probe length (Z axis) must be entered in the tool offset memory, e.g. under T9 D1, before the measuring cycle is called.



CALIBRATE_IN_Z	
N10 G54 G17 G0 X100 Y80	Position probe above calibration point
N15 T9 D1 Z55	Select length compensation, position probe above surface
N20 _MVAR=0 _SETVAL=20 _MA=3 _MD=1	Set parameters for calibration cycle (calibration in Z direction)
N21 _TZL=0 _TSA=1 _PRNUM=1	
N22 _VMS=0 _NMSP=1 _FA=1	
N25 CYCLE976	Cycle call for calibration in Z axis
N50 M30	End of program

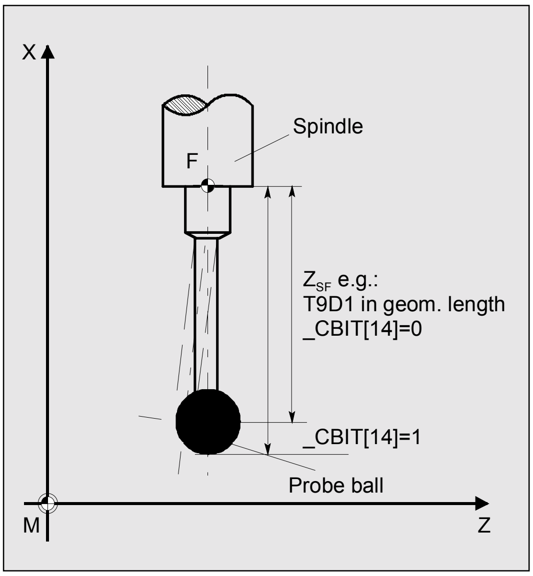
The new trigger value in Z is entered in the global data of probe 1 _WP[0,5].

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.3.4 Calibrate workpiece probe in applicate with calculation of probe length (measuring cycles SW 4.4. and higher)

Function

With this measuring cycle you can calibrate the probe on a random surface, e.g. on the workpiece, to determine the probe length in the applicate.



Precondition

The probe must be called **with** tool length offset.
Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

The probe length need not be known.

Parameters

_MVAR	10000	Calibration in applicate with length calculation
_SETVAL	REAL	Calibration setpoint
_MA	3	Measuring axis = applicate
_MD	0 positive axis direction 1 negative axis direction	Measurement direction
_PRNUM	INT	Measuring probe number

These following additional parameters are also valid:
_VMS, **_CORA**, **_FA** and **_NMSP**.

See Sections 2.2 and 2.3.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Procedure

Position before the cycle is called

The probe must be positioned opposite the calibration surface in such a way that the probe is deflected within the measurement path (2 · _FA · 1 mm) defined by _FA.

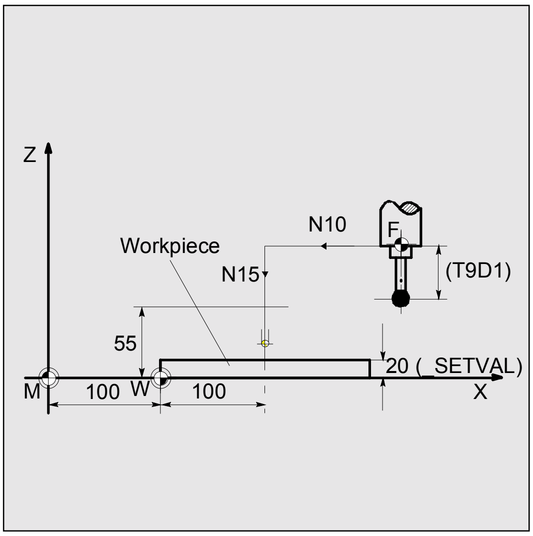
Position after the cycle has terminated

When the calibration procedure is completed the probe is positioned on the starting position.

Programming example

Calibration of workpiece probe 1 in the Z axis on the workpiece with length calculation

The radius of the probe ball must be entered in the tool offset memory, e.g. under T9 D1, before the cycle is called.



CALIBRATE_IN_Z	
N10 G54 G17 G0 X100 Y80	Position probe above calibration point
N15 T9 D1 Z55	Select length compensation, position probe above surface
N20 _MVAR=10000 _SETVAL=20 _MA=3 _MD=1	Parameter for calibration cycle (calibration in Z direction) with length calculation
N21 _PRNUM=3	
N22 _VMS=0 _NMSP=1 _FA=20	
N25 CYCLE976	Cycle call for calibration in Z axis
N50 M30	End of program

When the cycle is called the probe travels 40 mm in the Z direction at measurement feedrate 300 mm/min.
If the probe is triggered within this measuring path, length 1 is calculated in the tool offset memory of tool T9 and D offset D1. The radius of the probe ball is entered in the global data of probe _WP[2,5].

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.4 CYCLE977 Workpiece measurement: Hole/shaft/groove/web/rectangle (paraxial)

Programming

CYCLE977

Function

This cycle determines the dimensions of holes, shafts, grooves or webs. It can either perform automatic tool offset or zero offset to compensate for a difference from the derived center point of the hole, shaft, rectangle in the axes of the plane, or correct a groove or web in the measurement axis additively.

In SW 4.3 and higher the measuring cycle has been expanded to include the measurement variants

- Measurement of a ring inside and outside
- Measurement of a rectangle inside, outside with/without protection zone.

As of SW 4.5, it is also possible to measure the groove and web in the protection zone.

Result parameters

Depending on the measurement variant, the measuring cycle CYCLE977 supplies the following values as results in the GUD5 module (not for rectangle measurement):

_OVR [0]	REAL	Setpoint diameter/width hole, shaft, groove, web
_OVR [1]	REAL	Setpoint center point/center hole, shaft, groove, web in abscissa
_OVR [2]	REAL	Setpoint center point/center hole, shaft, groove, web in ordinate
_OVR [4]	REAL	Actual value diameter/width hole, shaft, groove, web
_OVR [5]	REAL	Actual value center point/center hole, shaft, groove, web in abscissa
_OVR [6]	REAL	Actual value center point/center hole, shaft, groove, web in ordinate
_OVR [8][1)]	REAL	Upper tolerance limit for diameter/width hole, shaft, groove, web
_OVR [12] [1)]	REAL	Lower tolerance limit for diameter/width hole, shaft, groove, web
_OVR [16]	REAL	Difference diameter/width hole, shaft, groove, web
_OVR [17]	REAL	Difference center point/center hole, shaft, groove, web in abscissa
_OVR [18]	REAL	Difference center point/center hole, shaft, groove, web in ordinate
_OVR [20] [1)]	REAL	Offset value
_OVR [27] [1)]	REAL	Zero offset area

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

_OVR [28]	REAL	Safe area
_OVR [29] [1)]	REAL	Dimensional difference
_OVR [30] [1)]	REAL	Empirical value
_OVR [31] [1)]	REAL	Mean value
_OVI [0]	INTEGER	D number or ZO number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [4] [1)]	INTEGER	Weighting factor
_OVI [5]	INTEGER	Measuring probe number
_OVI [6] [1)]	INTEGER	Mean value memory number
_OVI [7] [1)]	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number

1) For workpiece measurement with tool offset only

Result parameters

Depending on the measurement variant **rectangle measurement**, CYCLE977 supplies the following values as results in the GUD5 module:

_OVR [0]	REAL	Setpoint value rectangle length (in the abscissa)
_OVR [1]	REAL	Setpoint value rectangle length (in the ordinate)
_OVR [2]	REAL	Setpoint for rectangle center point, abscissa
_OVR [3]	REAL	Setpoint for rectangle center point, ordinate
_OVR [4]	REAL	Actual value for rectangle length (in the abscissa)
_OVR [5]	REAL	Actual value for rectangle length (in the ordinate)
_OVR [6]	REAL	Actual value for rectangle center point, abscissa
_OVR [7]	REAL	Actual value for rectangle center point, ordinate
_OVR [8] [1)]	REAL	Upper tolerance limit for rectangle length (in the abscissa)
_OVR [9] [1)]	REAL	Upper tolerance limit for rectangle length (in the ordinate)
_OVR [12] [1)]	REAL	Lower tolerance limit for rectangle length (in the abscissa)
_OVR [13] [1)]	REAL	Lower tolerance limit for rectangle length (in the ordinate)
_OVR [16]	REAL	Difference of rectangle length (in the abscissa)
_OVR [17]	REAL	Difference of rectangle length (in the ordinate)
_OVR [18]	REAL	Difference of rectangle center point, abscissa
_OVR [19]	REAL	Difference of rectangle center point, ordinate
_OVR [20] [1)]	REAL	Offset value
_OVR [27] [1)]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVR [29] [1)]	REAL	Dimensional difference
_OVR [30] [1)]	REAL	Empirical value
_OVR [31] [1)]	REAL	Mean value

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

_OVI [0]	INTEGER	D number or ZO number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [4] [1)]	INTEGER	Weighting factor
_OVI [5]	INTEGER	Measuring probe number
_OVI [6] [1)]	INTEGER	Mean value memory number
_OVI [7] [1)]	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number
_OVI [11] [2)]	INTEGER	Status offset request

1) For workpiece measurement with tool offset only
2) For measuring cycle SW 6.2 and higher; only for zero offset

Applicable probe types

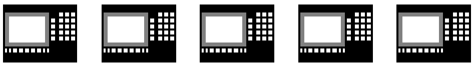
The measuring cycle operates with the following probe types which are coded via parameter _PRNUM:

- Multidirectional probe
- Monodirectional probe (bidirectional probe)

Measurement variants and prepositioning

CYCLE977 permits the following measurement variants which are specified via parameter _MVAR.

Value	*Measurement variant*	*Prepositioning in plane*	*Prepositioning in applicate*
1	Measure hole with tool offset	Hole center point	At measuring depth
2	Measure shaft with tool offset	Shaft center point	Above shaft
3	Measure groove with tool offset	Center point of groove	At measuring depth
4	Measure web with tool offset	Center point of web	Above web
5	Measure rectangle inside	Rectangle center point	At measuring depth
6	Measure rectangle outside	Rectangle center point	Above rectangle
101	ZO calculation in hole with ZO compensation	Hole center point	At measuring depth
102	ZO calculation on shaft with ZO compensation	Shaft center point	Above shaft
103	ZO calculation in groove with ZO compensation	Center point of groove	At measuring depth
104	ZO calculation on web with ZO compensation	Center point of web	Above web



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

105	ZO calculation in rectangle inside	Rectangle center point	At measuring depth
106	ZO calculation in rectangle outside	Rectangle center point	Above rectangle
1001	Measure hole with contouring of a protection zone	Hole center point	Above hole
1002	Measure shaft by including for a protection zone	Shaft center point	Above shaft
1003 [1]	Measure hole with contouring of a protection zone	Center point of groove	Above web
1004 [1]	Measure web by including for a protection zone	Web center point	Above web
1005	Measure rectangle inside with protection zone	Rectangle center point	Above rectangle
1006	Measure rectangle outside with protection zone	Rectangle center point	Above rectangle
1101	ZO calculation of hole with contouring of a protection zone	Hole center point	Above hole
1102	ZO calculation of shaft by including a protection zone	Shaft center point	Above shaft
1103 [1]	ZO calculation in groove with contouring of a protection zone	Groove center point	Above web
1104 [1]	ZO calculation at web by including a protection zone	Web center point	Above web
1105	ZO calculation in rectangle inside with protection zone	Rectangle center point	Above rectangle
1106	ZO calculation in rectangle outside with protection zone	Rectangle center point	Above rectangle

The measuring height in the applicate in which measuring is then performed in the plane is derived from the prepositioning in the applicate and the incremental parameter _ID.

1) Measuring cycles SW 4.5 and higher

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.4.1 CYCLE977 Measure hole, shaft, groove, web, rectangle

Function

Measure hole or shaft

This measuring cycle gauges

- within the hole or
- on a shaft

points P1, P2, P3 and P4 in the abscissa and ordinate.
These four measured values are used to calculate the actual value and the position of the hole center point in the abscissa and ordinate relative to the workpiece zero.
The center point of the abscissa is calculated from points P1 and P2. The probe is then positioned at the center point calculated and points P3 and P4 are measured.
These two points are used to calculate the hole/shaft center point in the ordinate and the hole/shaft diameter.

In SW 4.3 and higher, travel around (hole) and consideration (shaft) of a protection zone are supported. This provides for retraction for intermediate positioning in the applicate.

Measure groove or web

This measuring cycle gauges

- within the groove or
- on two parallel surfaces (web)

in the measuring axis.
The two measured values are used to calculate the actual value of the groove/the actual distance between the parallel surfaces, as well as the position of the groove center point/the center point in the measuring axis, relative to the workpiece zero.

Measure rectangle inside or outside

The measuring cycle automatically approaches 4 measuring points and determines the rectangle center point.
Optionally, a rectangle-shaped protection zone relating to the rectangle center point can be traveled around.

5.4.1 CYCLE977 Measure hole, shaft, groove, web, rectangle

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Options for hole and shaft diameter, groove or web width

- An empirical value stored in the GUD5 module is subsequently taken into account with the correct sign.
- A mean value derivation over several parts is possible as an option.
- Depending on the definition of _KNUM, no automatic offset is performed or, alternatively, length compensation or radius compensation (difference halved) of a tool to be specified is carried out.

Precondition

The probe must be called **with** tool length offset.

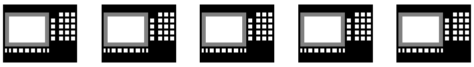
Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- on meas. cycle SW 5.4 → 500 or on meas. cycle SW 6.2 → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

Parameters

_MVAR	1	Measure hole with tool offset
	2	Measure shaft with tool offset
	3	Measure groove with tool offset
	4	Measure web with tool offset
	5	Measure rectangle inside with tool offset
	6	Measure rectangle outside with tool offset
	1001	Measure hole by contouring a protection zone with tool offset
	1002	Measure shaft by including a protection zone with tool offset
	1003[1)]	Measure groove by contouring a protection zone with tool offset
	1004[1)]	Measure web by including a protection zone with tool offset
	1005	Measure rectangle inside with protection zone with tool offset
	1006	Measure rectangle outside with protection zone with tool offset
_SETVAL	REAL	Setpoint (acc. to drawing) (only for hole/shaft/groove/web)
_SETV[0]	REAL	Setpoint value rectangle length (in the abscissa)
_SETV[1]		Setpoint value rectangle length (in the ordinate) (only when measuring a rectangle)

1) Measuring cycles SW 4.5 and higher



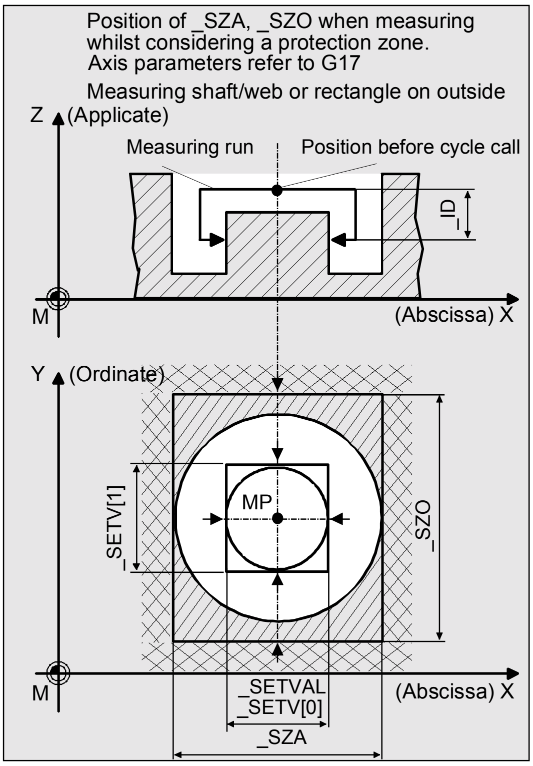
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

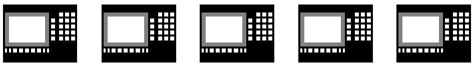
_ID	REAL	Incremental infeed of applicate with leading sign (only for measuring shaft, web or rectangle, and for measuring hole/groove/shaft/web with travel around or accounting for a protection zone)
_SZA	REAL	• Length of the protection zone in the abscissa (only for measuring rectangle) • Diameter/width of the protection zone (inside for hole/groove, outside for shaft/web)
_SZO	REAL	Length of the protection zone in the ordinate (only for measuring rectangle)
_MA	1...2	Number of measuring axis (only for measuring a groove or a web)
_KNUM	0 No automatic tool offset; >0 Automatic tool offset	With/without automatic tool offset
_TNUM	Integer, positive	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (as an alternative to _TNUM if tool management is active)

These following additional parameters are also valid:
_VMS, _CORA, _TZL, _TMV, _TUL, _TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP and **_K**.
See Sections 2.2 and 2.3.

The following applies to rectangle measuring:

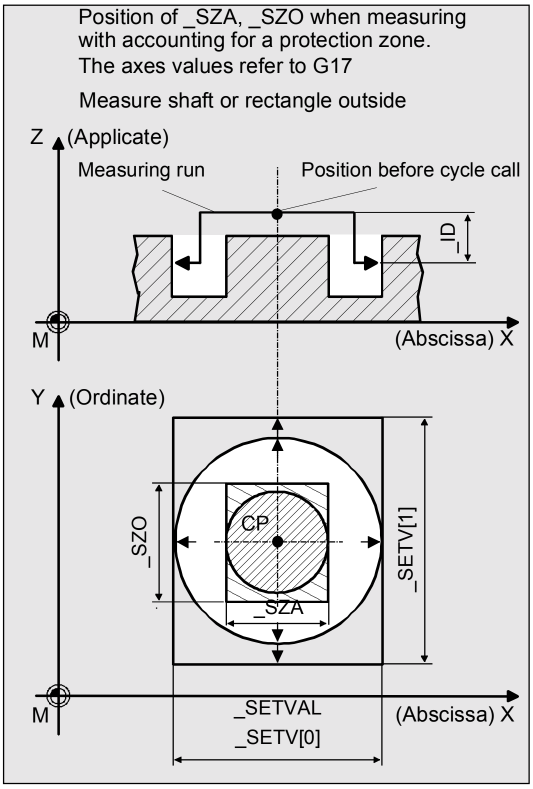
- All input parameters except for _MVAR and _SETVAL must be assigned in the same way as the corresponding measurement variants for groove/web.
- In addition to parameters _SETV, _SZA, _SZO, _ID, the parameters must be set for inside measurements on rectangles in the same way as for measuring grooves; and for outside measurements the remaining parameters must be set as for web measurements.





840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Procedure



Position before measuring cycle call with outside measurement (shaft, web, rectangle) or measuring with protection zone

The probe must be positioned at the center point in the plane, and the probe ball positioned above the upper edge, such that when infeed of value _ID is applied, the measurement level is reached.

Position before cycle call for inside measurement (hole, groove, rectangle)

The probe must be positioned at the center point in the plane. The probe ball must be positioned at measurement level inside the hole/groove/rectangle.

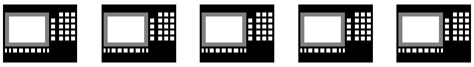
Position after the cycle has terminated

On completion of the measuring process, the probe is positioned above the calculated center point.

Notice!

Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane, orientation of the spindle in the plane and measuring velocity are the same for both measurement and calibration. Deviations can cause additional measuring errors.

5.4.1 CYCLE977 Measure hole, shaft, groove, web, rectangle



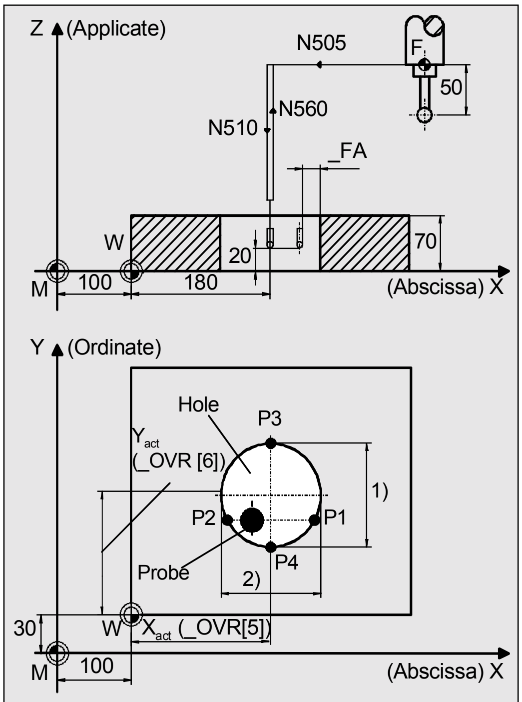
840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Programming example

Measuring a hole with CYCLE977

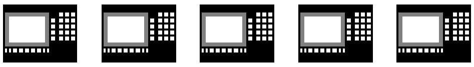
Probe length (Z axis) in TO memory T9 D1 (value 50). The difference calculated from the actual and setpoint diameter is compensated by the empirical value in the empirical value memory _EV[9] and compared with the tolerance parameter.

- If it is more than 1 mm (_TSA), alarm "Safe area violated" is output and the program is halted. Cancellation by resetting the control!
- If it is more than 0.06 mm (_TDIF), no compensation is performed and alarm "Permissible dimensional difference exceeded" is output and the program continues.
- If 0.03 mm is exceeded (_TUL/_TLL) the radius in T20 D1 is compensated 100% by this difference/2. Alarm "oversize" or "undersize" is displayed and the program continues.
- If 0.02 mm (_TMV) is exceeded the radius in T20 D1 is compensated 100% by this difference/2.
- If it is less than 0.02 mm (_TMV) the mean value is calculated (only if _CHBIT[4]=1! with mean value memory) with the mean value in mean value memory _MV[9] and by including weighting factor 3 (_K).
  - If the calculated mean value is >0.01 (_TZL) the radius from T20 D1 is compensated to a lesser degree by mean value/2 and the mean value in _MV[9] is deleted.
  - If the mean value is <0.01 (_TZL) the radius in T20 D1 is not compensated but is stored in mean value memory _MV[9].



1) Actual diameter (_OVR [4])
2) _SETVAL (set diameter 130)

MEASURE_HOLE	
N500 G54 T9	Select T No. probe
N505 G17 G0 X180 Y130	Position probe in X/Y plane at hole center
N510 Z20 D1	Position Z axis in hole
N515_MVAR=1 _SETVAL=130 _TUL=0.03 _TLL=-0.03 _KNUM=2001 _TNUM=20 _EVNUM=10 _K=3 _TZL=0.01 _TMV=0.02 _TDIF=0.06 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	Set parameters for measuring cycle call
N550 CYCLE977	Call measuring cycle
N560 G0 Z160	Retract Z axis from hole
N570 M30	End of program



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

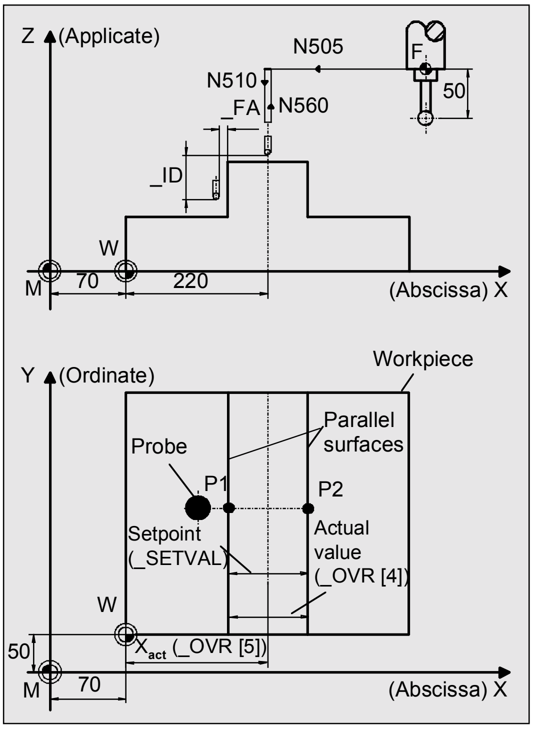
Programming example

Measuring a web with CYCLE977

Probe length (Z axis) in TO memory T9 D1 (value 50).

A web width 130 ± 0.03 is to be measured. The maximum permissible deviation from the web center is taken as 2 mm, the maximum permissible deviation of the web width is also 2 mm. To obtain a minimum measuring path of 1 mm, the measuring path is programmed as 2 + 1 + 1 = 4 mm (max. measuring path = 8 mm).
A measured deviation >1 mm is not permissible however.

The radius in T20 D1 is automatically compensated for according to the same criteria as described in programming example "Measuring a hole with CYCLE977".



MEASURE_WEB	
N500 G54 T9	Select T No. probe
N505 G17 G0 X220 Y130	Position probe in X/Y plane at web center
N510 Z101 D1	Position Z axis above web
N515_MVAR=4 _SETVAL=130 _TUL=0.03 _TLL=-0.03 _MA=1 _ID=-40, _KNUM=2001 _TNUM=20 _EVNUM=10 _K=3 _TZL=0.01 _TMV=0.02 _TDIF=0.06 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=4	Set parameters for measuring cycle call
N550 CYCLE977	Call measuring cycle
N560 G0 Z160	Run up Z axis
N570 M30	End of program

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.4.2 CYCLE977 ZO calculation in hole, shaft, groove, web, rectangle

Function

ZO calculation in a hole or on a shaft

The measuring cycle gauges points P1, P2, P3 and P4

- within the hole or
- on the shaft

points P1, P2, P3 and P4 in the abscissa and ordinate. These four measured values are used to calculate the position of the hole/shaft center point in the abscissa and ordinate relative to the workpiece zero.

The center point of the abscissa is calculated from points P1 and P2. The probe is then positioned at the center point calculated and points P3 and P4 are measured. These two points provide the hole/shaft center point of the ordinate.

In SW 4.3 and higher, travel around (hole) and consideration (shaft) of a protection zone are supported. This provides for retraction for intermediate positioning in the applicate.

ZO calculation in a groove or on a web

This measuring cycle gauges

- within the groove or
- on two parallel surfaces (web)

in the measuring axis. These two measured values are used to calculate the position of the groove center point - or the center point on a web - in the measuring axis in relation to the workpiece zero.

ZO calculation in rectangle inside or outside

The measuring cycle automatically approaches 4 measuring points and determines the rectangle center point.
Optionally, a rectangle-shaped protection zone relating to the rectangle center point can be traveled around.

5.4.2 CYCLE977 ZO calculation in hole, shaft, groove, web, rectangle

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

The following applies to all ZO calculations:

The difference is determined from the set center point (starting position) and the center point actual value determined by the cycle.

The multiplying factor for measurement path 1 mm makes it possible to take into account the scatter band of the blanks (set value).

Depending on the definition of _KNUM, either no automatic ZO entry is made, or the difference in the measuring axis when measuring a groove or web, or in the abscissa and ordinate, is added to the specified ZO memory. If a fine offset is active (MD 18600: MM_FRAME_FINE_TRANS), an additive ZO will be implemented in it, otherwise it is implemented in the coarse offset.

Precondition

The probe must be called **with** tool length offset.
Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- on meas. cycle SW 5.4 → 500 or on meas. cycle SW 6.2 → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

Parameters

_MVAR	101	ZO calculation in a hole with compensation of ZO
	102	ZO calculation on a shaft with compensation of ZO
	103	ZO calculation in a groove with compensation of ZO
	104	ZO calculation on a web with compensation of ZO
	105	ZO calculation in rectangle inside with comp. of the ZO
	106	ZO calculation in rectangle outside with comp. of the ZO
	1101	ZO calculation in hole by circumnavigating a protection zone with compensation of the ZO
	1102	ZO calculation of shaft by including a protection zone with compensation of the ZO
	1103 1)	ZO calculation in groove by circumnavigating a protection zone with compensation of the ZO
	1104 1)	ZO calculation of web by including a protection zone with compensation of the ZO
	1105	ZO calculation in rectangle inside with protection zone with compensation of the ZO
	1106	ZO calculation in rectangle outside with protection zone with compensation of the ZO

1) Measuring cycles SW 4.5. and higher

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

_ID	REAL	Incremental infeed of applicate with leading sign (only for ZO calculation on a shaft with/without consideration of a protection zone or on a web)
_MA	1...2	Number of measuring axis (only for ZO calculation in a groove or on a web)
_SETVAL	REAL	Setpoint value for diameter/width hole (only for hole, shaft, groove, web)
_SETV[0] **_SETV[1]**	REAL	Setpoint value rectangle length (in the abscissa) Setpoint value rectangle length (in the ordinate) (only for ZO calculation for a rectangle)
_SZA	REAL	• Length of the protection zone in the abscissa (only for ZO calculation on rectangle) • Diameter/width of the protection zone (inside for hole/groove, outside for shaft/web)
_SZO	REAL	Length of the protection zone in the ordinate (only for ZO calculation on rectangle)
_KNUM	0 no automatic ZO compensation; 1...99 automatic additive ZO compensation in G54...G57, G505...G599	With/without automatic ZO calculation
Meas. cycles SW 4.4 and higher	1000 automatic additive ZO compensation in channel-specific basic frame[1]	
Meas. cycles SW 6.2 and higher[2]	1011...1026 automatic ZO compensation in 1st to 16th basic frame (channel)	($P_CHBFR[0]...$P_CHBFR[15])
	1051...1066 automatic ZO compensation in 1st to 16th basic frame (global)	($P_NCBFR[0]...$P_NCBFR[15])
	2000 automatic ZO compensation in system frame	scratching system frame ($P_SETFR)
	9999 automatic ZO compensation in an active frame	settable frames G54..G57, G505...G599 or for G500 in last active basic frame according to $P_CHBFRMASK (most significant bit)

1) As of measuring cycles SW 5.3, compensation is carried out in the last basic frame (per MD 28081: MM_NUM_BASE_FRAMES) if more than one is available. If measuring cycles higher than SW 5.3 are used at a control with SW 4, parameter _SI[1] in the GUD 6 module must be set to 4!

2) **Measuring cycles version SW 6.2 and higher can only be used with NCK-SW 6.3 and higher.**

These following additional parameters are also valid:
_VMS, _CORA, _TSA, _FA, _PRNUM and **_NMSP**.
See Sections 2.2 and 2.3.

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

The following applies to rectangle measuring:

- All input parameters except for _MVAR and _SETVAL must be assigned in the same way as the corresponding measurement variants for groove/web.
- In addition to parameters _SETV, _SZA, _SZO, _ID, the parameters must be set for inside measurements on rectangles in the same way as for measuring grooves; and for outside measurements the remaining parameters must be set as for web measurements.

Procedure

Position before measuring cycle call with outside measurement (shaft, web, rectangle) or measuring with protection zone

The probe must be positioned at the center point in the plane, and the probe ball positioned above the upper edge, such that when infeed of value _ID is applied, the measurement level is reached.

Position before cycle call for inside measurement (hole, groove, rectangle)

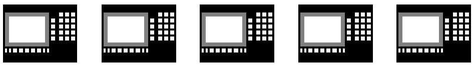
The probe must be positioned at the center point in the plane. The probe ball must be positioned at measurement level inside the hole/groove/rectangle.

Position after the cycle has terminated

When measuring is completed the probe is positioned on the calculated center point for inside and outside measurement.

Notice!

Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane, orientation of the spindle in the plane and measuring velocity are the same for both measurement and calibration. Deviations can cause additional measuring errors.



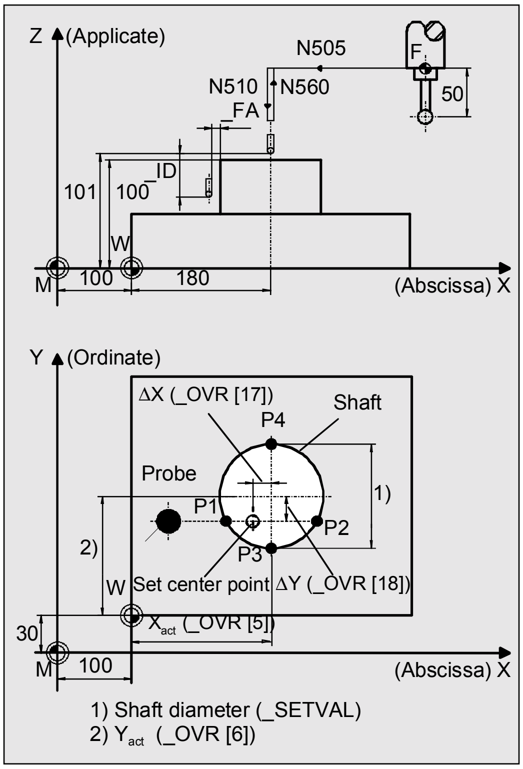
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

ZO calculation at a shaft with CYCLE977
Probe length (Z axis) in TO memory T9 D1 (value 50).

A shaft is measured, the aim being that the hole center measured is identical to the setpoint position approached. The permissible deviation from the shaft center is taken as 2 mm; the deviation of the shaft diameter from the setpoint diameter 130 may not exceed 6 mm. To obtain a minimum measuring path of 1 mm up to the edge, the measuring path is programmed as 2 + 3 + 1 = 6 mm (max. measuring path = 12 mm).

Automatic compensation is performed in G54, X and Y by the calculated difference between the actual value and set position of the shaft center, should it be less than 2 mm (_TSA) in both axes. Otherwise alarm "Safe area violated" is output and program execution cannot be continued.



ZO_SHAFT	
N500 G54 T9	Select T No. probe
N505 Z101 D1	Position Z axis above shaft
N510 G17 G0 X150 Y130	Position probe in X/Y plane at shaft center point (setpoint position)
N515_MVAR=102 _SETVAL=130 _ID=-30 _KNUM=1 _TSA=2 _PRNUM=1 _VMS=0 _NMSP=1 _FA=6	Set parameters for measuring cycle call
N550 CYCLE977	Call measuring cycle
N555 G54	Renewed call of the zero offset G45 so that the changes take effect through the measuring cycle!
N560 G0 Z160	Run up Z axis
N570 M30	End of program

5.4.2 CYCLE977 ZO calculation in hole, shaft, groove, web, rectangle

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

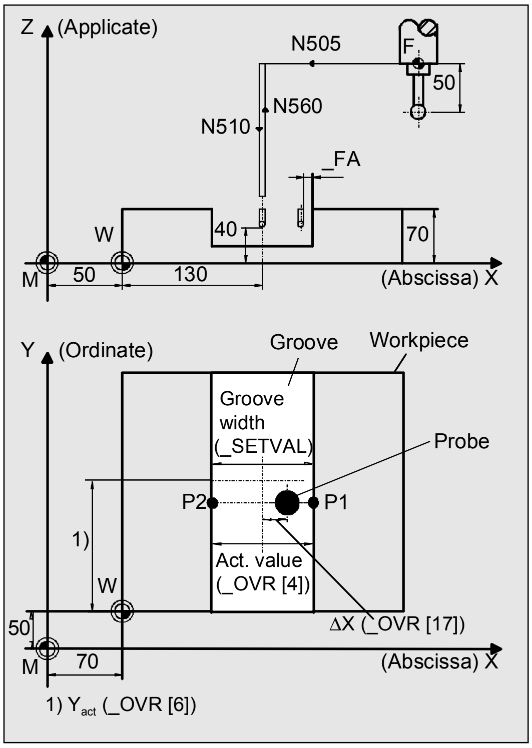
Programming example

ZO calculation in a groove with CYCLE977

Probe length (Z axis) in TO memory T9 D1 (value 50).

A groove in the X axis is measured, the aim being to determine a groove center in X that is equal to the setpoint position. The maximum permissible deviation of the groove center is taken as 2 mm. The groove width may deviate up to 6 mm from setpoint width 100. To obtain a minimum measuring path of 1 mm up to the edge, the measuring path is programmed as 2 + 3 + 1 = 6 mm (max. measurement path 12 mm).

Automatic compensation in X (abscissa) in G54 is performed in case the difference between the actual and setpoint position of the center of the groove in X is less than 2 mm (_TSA). Otherwise alarm "Safe area violated" is output and program execution cannot be continued.



ZO_SHAFT	
N500 G54 T9	Select T No. probe
N505 G17 G0 X150 Y130	Position probe in X/Y plane at groove center (setpoint position)
N510 Z40 D1	Position Z axis in groove
N515_MVAR=103 _SETVAL=100 _MA=1 _KNUM=1 _TSA=2 _PRNUM=1 _VMS=0 _NMSP=1 _FA=6	Set parameters for measuring cycle call, measuring axis is X (abscissa)
N550 CYCLE977	Call measuring cycle
N555 G54	Renewed call of the zero offset G45 so that the changes take effect through the measuring cycle!
N560 G0 Z160	Retract Z axis from groove
N570 M30	End of program

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.5 CYCLE978 Workpiece measurement: Surface

Programming

CYCLE978

Function

The measuring cycle determines the dimensions of surfaces paraxially with reference to the workpiece zero by single-point measurement and executes an automatic tool compensation or zero offset in the measuring axis.

Differential measurements are also possible with this cycle.

Result parameters

Depending on the measurement variant, CYCLE978 makes the following values available as results in the GUD5 module:

_OVR [0]	REAL	Setpoint for measuring axis
_OVR [1]	REAL	Setpoint for abscissa
_OVR [2]	REAL	Setpoint for ordinate
_OVR [3]	REAL	Setpoint for applicate
_OVR [4]	REAL	Actual value for measuring axis
_OVR [8] [1)]	REAL	Upper tolerance limit for measuring axis
_OVR [12] [1)]	REAL	Lower tolerance limit for measuring axis
_OVR [16]	REAL	Difference for measuring axis
_OVR [20] [1)]	REAL	Offset value
_OVR [27] [1)]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVR [29] [1)]	REAL	Dimensional difference
_OVR [30]	REAL	Empirical value
_OVR [31] [1)]	REAL	Mean value
_OVI [0]	INTEGER	D number or ZO number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [4] [1)]	INTEGER	Weighting factor
_OVI [5]	INTEGER	Measuring probe number
_OVI [6] [1)]	INTEGER	Mean value memory number
_OVI [7]	INTEGER	Empirical value memory number

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number
_OVI [11] 2)	INTEGER	Status offset request

1) for single-point measurement with automatic tool compensation only
2) for measuring cycle SW 6.2 and higher; only for zero offset

Differential measurement

Differential measurement means that the measuring point is measured twice, the first time at the probe position reached and the second time with a spindle reversal of 180° (rotation of probe through 180°). Determines the trigger point of the probe in the measuring axis. The trigger point is stored in the global user data for the appropriate axis direction. An uncalibrated probe can therefore be used for the measurement.

Preconditions for differential measurement

- Spindle orientation (with SPOS command) by means of NC
- Bidirectional/multidirectional probe

Random positioning of probe in spindle between 0° and 360° (at least every 90°) (all-round coverage).

Applicable probe types

The measuring cycle operates with the following probe types which are coded via parameter _PRNUM:

- Multidirectional probe
- Bidirectional probe
- Monodirectional probe

Monodirectional probes must be calibrated! These probes cannot be used to take differential measurements!

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Measurement variants

CYCLE978 permits the following measurement variants which are specified via parameter _MVAR.

Value	*Measurement variant*
0	Measure surface
100	ZO calculation on surface
1000	Measure surface with differential measurement
1100	ZO calculation on surface with differential measurement

Prepositioning
The probe is prepositioned facing the surface to be measured for all measurement variants.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.5.1 CYCLE978 ZO calculation on a surface (single point measuring cycle)

Function

This measuring cycle determines the actual value of a blank relative to the workpiece zero.

An empirical value stored in the GUD5 module is subsequently taken into account with the correct sign.

The multiplying factor for measurement path "2a" makes it possible to take into account the scatter band of the blanks (set value).

Depending on the definition of _KNUM, no automatic ZO entry is performed or, alternatively, the measuring axis difference is added in the specified ZO memory. If a fine offset is active (MD 18600: MM_FRAME_FINE_TRANS), an additive ZO will be implemented in it, otherwise it is implemented in the coarse offset.

Precondition

The probe must be called **with** tool length offset.
Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Parameters

_MVAR	100		ZO calculation on surface
	1100		ZO calculation on surface with differential measurement
_SETVAL	REAL		Setpoint with respect to workpiece zero
_MA	1...3		Number of measuring axis
_KNUM	0	no automatic ZO compensation;	With/without automatic ZO calculation
	1...99	automatic additive ZO compensation in G54...G57, G505...G599	
Meas. cycles ≥ SW 4.4	1000	automatic additive ZO compensation in channel-specific basic frame[1)]	
Meas. cycles ≥ SW 6.2[2)]	1011...1026	automatic ZO compensation in 1st to 16th basic frame (channel)	($P_CHBFR[0]...$P_CHBFR[15])
	1051...1066	automatic ZO compensation in 1st to 16th basic frame (global)	($P_NCBFR[0]...$P_NCBFR[15])
	2000	automatic ZO compensation in system frame	scratching system frame ($P_SETFR)
	9999	automatic ZO compensation in an active frame	settable frames G54..G57, G505...G599 or for G500 in last active basic frame according to $P_CHBFRMASK (most significant bit)

1) As of measuring cycles SW 5.3, compensation is carried out in the last basic frame (per MD 28081: MM_NUM_BASE_FRAMES) if more than one is available. If measuring cycles higher than SW 5.3 are used at a control with SW 4, parameter _SI[1] in the GUD 6 module must be set to 4!

2) **Measuring cycles version SW 6.2 and higher can only be used with NCK-SW 6.3 and higher.**

These following additional parameters are also valid:
_VMS, _CORA, _TSA, _FA, _PRNUM, _EVNUM and **_NMSP**.
See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called

The probe must be positioned facing the surface to be measured.

Position after the cycle has terminated

On completion of the measurement, the probe is positioned facing the measurement surface at a distance corresponding to _FA · 1 mm.

Notice!

Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane, orientation of the spindle in the plane and measuring velocity are the same for both measurement and calibration. Deviations can cause additional measuring errors.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

ZO calculation at a workpiece with CYCLE978

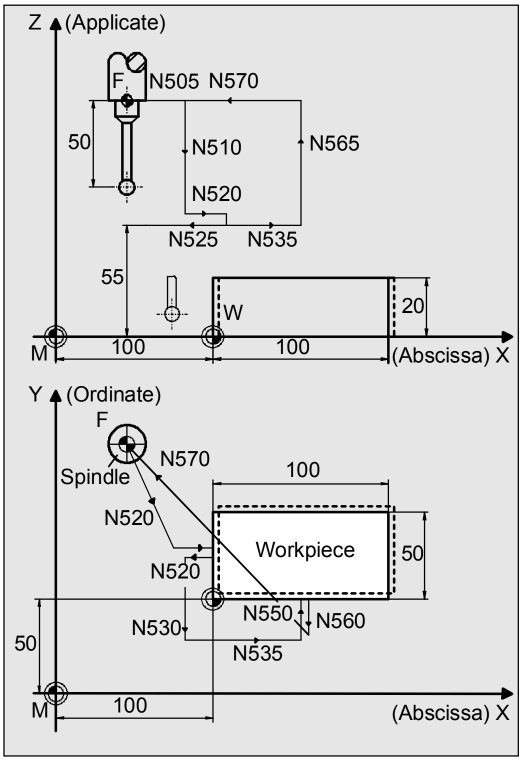
The ZO is to be checked on a workpiece. Any deviation from the selected ZO as a result of clamping tolerances must be compensated for automatically by means of additive ZO so that machining of the workpiece can be started.

The permissible deviation is 3 mm.

To obtain a minimum path of 1 mm up to the edge, the measuring path is programmed as 3 + 1 = 4 mm (max. measuring path = 8 mm).

Probe length (Z axis) stored in TO memory T9 D1.

Automatic compensation is performed in X (abscissa) G54 in case the difference between the actual and setpoint position compensated for by the empirical value in _EV[9] in measuring axis X is less than 3 mm (_TSA). Otherwise alarm "Safe area violated" is output and program execution cannot be continued.



ZO_CALCULATION_1	
N500 G54 T9	Select T No. probe
N505 G17 G0 G90 X-20 Y25	Position probe in X/Y plane opposite measuring surface
N510 Z10 D9	Position probe in Z and select tool offset
N515 _MVAR=100 _SETVAL=0 _MA=1 _KNUM=1 _EVNUM=10 _TSA=3 _PRNUM=1 _VMS=0 _NMSP=1 _FA=4	Set parameters for measuring cycle call
N520 CYCLE978	Measuring cycle for ZO calculation in
N525 G0 X-20	Retract in X axis
N530 Y-20	Position in Y axis
N535 X50	Position in X axis
N540 _EVNUM=11 _MA=2	Set parameters for measuring cycle call
N550 CYCLE978	ZO calculation in Y axis
N555 G54	Renewed call of the zero offset G45 so that the changes take effect through the measuring cycle!
N560 G0 Y-20	Retract in Y axis
N565 Z100	Retract in Z axis
N570 X-40 Y80	Retract in X/Y
N580 M30	End of program

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.5.2 CYCLE978 Single-point measurement

Function

This measuring cycle determines the actual value of the workpiece in the measuring axis selected relative to the workpiece zero as well as the difference between set and actual values.

An empirical value stored in the GUD5 module is subsequently taken into account with the correct sign.

Optionally, averaging is performed over a number of parts and the tolerance bands are checked.

Depending on the definition of _KNUM, no automatic offset, length compensation or radius compensation is carried out.

Precondition

The probe must be called **with** tool length offset.

Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- In measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Parameters

_MVAR	0 1000	Measure surface Measure surface with differential measurement
_SETVAL	REAL	Setpoint (acc. to drawing)
_MA	1...3	Number of measuring axis
_KNUM	0 No automatic tool offset; >0 Automatic tool offset	With/without automatic tool offset
_TNUM	Integer, positive	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)

These following additional parameters are also valid:
_VMS, _CORA, _TZL, _TMV, _TUL, _TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP and **_K**.

See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called
The probe must be positioned facing the surface to be measured.

Position after the cycle has terminated
On completion of the measurement, the probe is positioned facing the measurement surface at a distance corresponding to _FA · 1 mm.

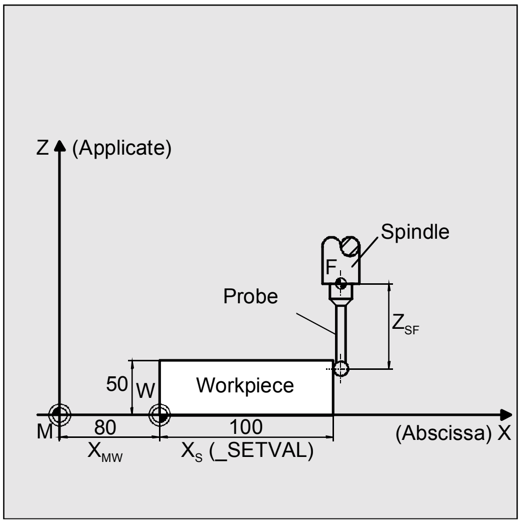
Notice!
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane, orientation of the spindle in the plane and measuring velocity are the same for both measurement and calibration. Deviations can cause additional measuring errors.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Single-point measurement in X axis with CYCLE978

Probe length (Z axis) in TO memory T9 D1 (value 50). The dimensional accuracy is to be checked for the edge of a workpiece machined tool T20D1. For a deviation > 0.01, the tool radius is to be compensated automatically for this tool. The maximum permissible deviation is taken as 1 mm. To obtain a minimum measuring path of 1 mm, the measuring path is programmed as 1 + 1 = 2 mm (max. measuring path = 4 mm). The difference calculated from the actual and setpoint diameter is compensated for by the empirical value in the empirical value memory _EV[19] and compared with the tolerance parameter.



- If it is more than 1 mm (_TSA), alarm "Safe area violated" is output and program execution is not continued.
- If it is more than 0.06 mm (_TDIF), no compensation is performed and alarm "Permissible dimensional difference exceeded" is output and the program continues.
- If 0.03 mm (_TUL/_TLL) is exceeded, the radius in T20 D1 is compensated 100% by this difference. Alarm "Oversize" or "Undersize" is displayed and the program is continued.
- If 0.02 mm (_TMV) is exceeded, the radius in T20 D1 is compensated 100% by this difference.
- If it is less than 0.02 mm (_TMV), the mean value is calculated (only if _CHBIT[4]=1! with mean value memory) with the mean value in mean value memory _MV[19] and by including weighting factor 3 (_K).
  - If the calculated mean value is > 0.01 (_TZL), the radius from T20 D1 is compensated to a lesser degree by mean value/2 and the mean value in _MV[19] is deleted.
  - If the mean values is < 0.01 (_TZL), there is no radius offset for T20 D1, but it is stored in the mean value memory _MV[19] if the mean value storage (_CHBIT[4]=1) is active.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

SINGLE_POINT_MEASUREMENT	
N500 G54 T9	Select T No. probe
N505 G17 G0 G90 X120 Y150	Position probe in X/Y plane opposite measuring point
N510 Z40 D1	Position Z axis on level with measuring point and select tool offset
N515 _MVAR=0 _SETVAL=100 _TUL=0.03 _TLL=-0.03 _MA=1 _KNUM=2001 _TNUM=20 _EVNUM=20 _K=3 _TZL=0.01 _TMV=0.02 _TDIF=0.06 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2	Set parameters for measuring cycle call
N520 CYCLE978	Measuring cycle for single-point measurement in X axis
N525 G0 Z160	Run up Z axis
N580 M30	End of program

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.6 CYCLE979 Workpiece measurement: Hole/shaft/groove/web (at a random angle)

Programming

CYCLE979

Function

A hole or shaft is determined by this cycle by means of three-point or four-point measurement. It is thus possible to measure **circle segments**, the center point of which is located well outside the machine.

Measurement at points P1, P2, P3 and P4 is performed at random angles (2D = two-dimensional; measure in 2 axes simultaneously, depending on the angle of measurement).

The probe is positioned from P1 to P2, from P2 to P3 and from P3 to P4 with circular interpolation (with measurements of holes and shafts). The _FA distance between probe and the contour is maintained.

On completion of the cycle, the probe is facing P3 (or P4; in the case of groove and web measurements, it is facing P2) at a distance corresponding to _FA.

Precondition
The probe must be positioned in the vicinity of P1 at the desired depth, so that point P1 can be approached without collision with linear interpolation from that position.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Result parameters

Depending on the measurement variant, CYCLE979 supplies the following values as results in the GUD5 module:

_OVR [0]	REAL	Setpoint diameter/width hole, shaft, groove, web
_OVR [1]	REAL	Setpoint center point/center in abscissa
_OVR [2]	REAL	Setpoint center point/center in ordinate
_OVR [4]	REAL	Actual value diameter/width hole, shaft, groove, web
_OVR [5]	REAL	Actual value center point/center in abscissa
_OVR [6]	REAL	Actual value center point/center in ordinate
_OVR [8] 1)	REAL	Upper tolerance limit for diameter/width hole, shaft, groove, web
_OVR [12] 1)	REAL	Lower tolerance limit for diameter/width hole, shaft, groove, web
_OVR [16]	REAL	Difference diameter/width hole, shaft, groove, web
_OVR [17]	REAL	Difference center point/center in abscissa
_OVR [18]	REAL	Difference center point/center in ordinate
_OVR [20] 1)	REAL	Offset value
_OVR [27] 1)	REAL	Zero offset area
_OVR [28] 1)	REAL	Safe area
_OVR [29] 1)	REAL	Permissible dimension difference
_OVR [30] 1)	REAL	Empirical value
_OVR [31] 1)	REAL	Mean value
_OVI [0]	INTEGER	D number or ZO number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [4] 1)	INTEGER	Weighting factor
_OVI [5]	INTEGER	Measuring probe number
_OVI [6] 1)	INTEGER	Mean value memory number
_OVI [7] 1)	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number
_OVI [11] 2)	INTEGER	Status offset request

1) For workpiece measurement with tool offset only
2) For measuring cycle SW 6.2 and higher; only for zero offset

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Applicable probe types

The measuring cycle operates with the following probe types which are coded via parameter _PRNUM:

- Multidirectional probe
- Bidirectional probe
- Monodirectional probe

This parameter also contains the specification for three-point and four-point measurements and has the following values:

Digit				*Meaning*
4	**3**	**2**	**1**	
0				Three-point measurement
1				Four-point measurement
	0			Multidirectional probe
	1			Monodirectional probe
		-	-	Probe number (two digits)

Measurement variants and prepositioning

CYCLE979 permits the following measurement variants which are specified via parameter _MVAR.

Value	*Measurement variant*	*Prepositioning in plane*
1	Measure hole with tool offset	In hole at measuring height
2	Measure shaft with tool offset	Near P1 at measuring height
3	Measure groove with tool offset	In groove at measuring height
4	Measure web with tool offset	Near P1 at measuring height
101	ZO calculation in hole with ZO compensation	In hole at measuring height
102	ZO calculation on shaft with ZO compensation	Near P1 at measuring height
103	ZO calculation in groove with ZO compensation	In groove at measuring height
104	ZO calculation on web with ZO compensation	Near P1 at measuring height

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.6.1 CYCLE979 Measure hole, shaft, groove, web

Function

Measure hole or shaft

This measuring cycle gauges

- within the hole or
- when contouring the shaft.

The position of these points is determined by initial angle _STA1 and indexing angle _INCA. These four measured values are used to calculate the actual value of the diameter and position of the center point in the abscissa and ordinate relative to the workpiece zero.

Measure groove or web

This measuring cycle gauges points P1 and P2 inside the groove or web. These measured values are used to calculate the actual value of the groove/web width as well as the position of the groove/web center point in the measuring axis relative to the workpiece zero.

Options for hole and shaft diameter, groove or web width

- An empirical value stored in the GUD5 module is subsequently taken into account with the correct sign.
- A mean value derivation over several parts is possible as an option.
- Depending on the definition of _KNUM, no automatic offset is performed or, alternatively, length compensation or radius compensation (difference halved) is carried out.

Precondition

The probe must be called **with** tool length offset.

Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Parameters

_MVAR	1 2 3 4	Measure hole Measure shaft Measure groove Measure web
_SETVAL	REAL	Setpoint = diameter/width (acc. to drawing)
_CPA	REAL	Center point abscissa (referred to workpiece zero)
_CPO	REAL	Center point ordinate (referred to workpiece zero)
_STA1	0...360 degrees	Starting angle
_ID	REAL	Incremental infeed of the applicates with sign (measure only in web)
_INCA	0...360 degrees	Indexing angle (only for measuring hole/shaft)
_KNUM	0 No automatic tool offset; >0 Automatic tool offset	With/without automatic tool offset
_TNUM	Integer, positive	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)

These following additional parameters are also valid:
_VMS, _RF, _CORA, _TZL, _TMV, _TUL, _TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP and **_K**.
See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called
The probe must be positioned facing P1 and the probe ball at the measurement level.

Position after the cycle has terminated for measuring the hole/shaft
On completion of the measuring process, the probe is positioned facing P3 (or P4 for four-point measurement) at a distance corresponding to _FA · 1 mm.

Position after the cycle has terminated for measuring groove/web
On completion of the measuring process, the probe is positioned opposite P2 at a distance corresponding to _FA.

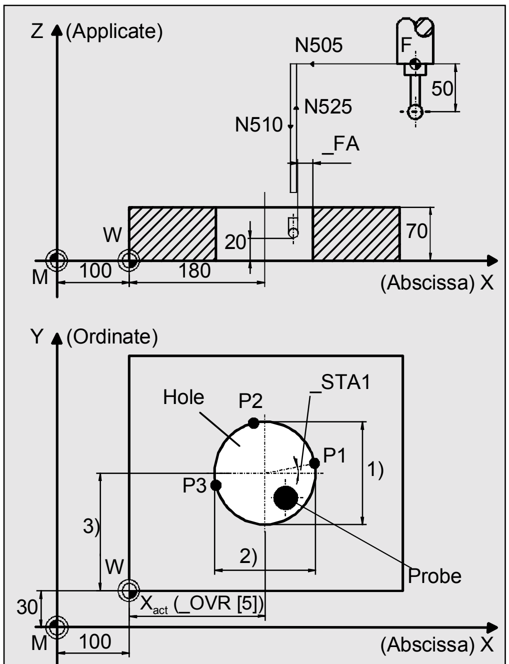
Notice!
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane, orientation of the spindle in the plane and measuring velocity are the same for both measurement and calibration. Deviations can cause additional measuring errors.

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Programming example

Measuring a hole with CYCLE979

The dimensional accuracy of a workpiece with a hole drilled using tool T20D1 is to be checked. For a deviation from the setpoint diameter 130 >0.01, the tool radius is to be corrected automatically. The maximum permissible deviation is taken as max. 1 mm. To obtain a minimum measuring path of 1 mm up to the edge of the hole, the measuring path is programmed as 1 + 1 = 2 mm (max. measuring path = 4 mm).
The center point of the hole lies at X180 Y130. The points P1, P2 and P3, whose position is defined by the start angle 10° and the following angle 90°. Traversing between points is carried out with a circular feed of 1000 mm/min.



1) Actual diameter (_OVR [4])
2) _SETVAL (Set diameter 130)
3) Y_{act} (_OVR [6])

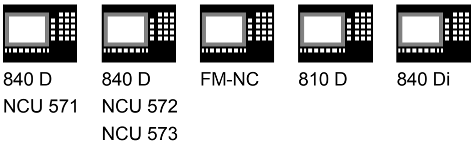
The difference calculated from the actual and setpoint diameter is compensated for by the empirical value in the empirical value memory _EV[19] and compared with the tolerance parameter.

- If it is more than 1 mm (_TSA), alarm "Safe area violated" is output and program execution is not continued.
- If it is more than 0.06 mm (_TDIF), no compensation is performed and alarm "Permissible dimensional difference exceeded" is output and the program continues.
- If 0.03 mm is exceeded (_TUL/_TLL), the radius in T20 D1 is compensated 100% by this difference/2. Alarm "oversize" or "undersize" is displayed and the program continues.
- If 0.02 mm (_TMV) is exceeded, the radius in T20 D1 is compensated 100% by this difference/2.
- If it is less than 0.02 mm (_TMV), the mean value is calculated (only if _CHBIT[4]=1! with mean value memory) with the mean value in mean value memory _MV[19] and by including weighting factor 3 (_K).

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

- If the mean value obtained is >0.01 (_TZL), the reduced compensation of the radius for T20 D1 is the mean value/2 and the mean value is deleted in _MV[19].
- If the mean values is < 0.01 (_TZL), there is no radius offset for T20 D1, but it is stored in the mean value memory _MV[19] if the mean value storage (_CHBIT[4]=1) is active.

MEASURE_HOLE	
N500 G54 T9	Select T No. probe
N505 G17 G0 G90 X120 Y150	Position probe in X/Y plane in vicinity of P1
N510 Z20 D1	Position Z axis at P1 level and select tool offset
N515_MVAR=1 _SETVAL=130 _TUL=0.03 _TLL=-0.03 _CPA=180 _CPO=130 _STA1=10 _INCA=90 _RF=1000 _KNUM=2001 _TNUM=20 _EVNUM=20 _K=3 _TZL=0.01 _TMV=0.02 _TDIF=0.06 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2	Set parameters for measuring cycle call
N520 CYCLE979	Call measuring cycle for hole measurement in X/Y
N525 G0 Z160	Run up Z axis
N570 M30	End of program

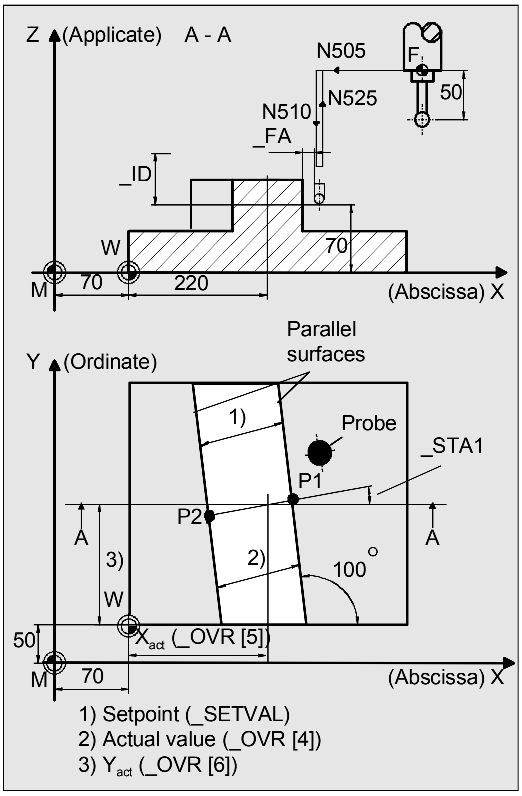


Programming example

Measuring a web with CYCLE979

The dimensional accuracy of a workpiece web produced using tool T20D1 is to be checked. For a deviation >0.01, web width 100 the radius of this tool is to be compensated automatically. The maximum permissible deviation is taken as max. 1 mm. To ensure a minimum measuring path of 1 mm up to the path edge, the measuring path is programmed with 1 + 1 = 2 mm (max. measuring path = 4 mm).
The center of the web lies at X220 Y130. The length of P1 is defined by the start angle 10°.

The radius in T20 D1 is automatically compensated according to the same criteria as described in programming example "Measuring a hole with CYCLE979".



MEASURE_WEB	
N500 G54 T9	Select T No. probe
N505 G17 G0 G90 X260 Y130	Position probe in X/Y plane in vicinity of P1
N510 Z70 D1	Position Z axis at P1 level and select tool offset
N515_MVAR=4 _SETVAL=100 _TUL=0.03 _TLL=-0.03 _CPA=220 _CPO=130 _STA1=10 _ID=35 _KNUM=2001 _TNUM=20 _EVNUM=10 _K=3 _TZL=0.01 _TMV=0.02 _TDIF=0.06 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2	Set parameters for measuring cycle call
N520 CYCLE979	Call measuring cycle for web measurement in X/Y plane
N525 G0 Z160	Run up Z axis
N570 M30	End of program

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.6.2 CYCLE979 ZO calculation in hole, shaft, groove, web

Function

ZO calculation in a hole or on a shaft

This measuring cycle gauges

- within the hole or
- when contouring the shaft.

These four measured values are used to calculate the actual hole/shaft diameter and the position of the hole/shaft center point in the abscissa and ordinate relative to the workpiece zero.

ZO calculation in a groove or on a web

This measuring cycle gauges

- within the groove or
- on two parallel surfaces (web)

The two measured values are used to calculate the actual groove/web width as well as the position of the groove/web center point in the measuring axis relative to the workpiece zero.

The following applies to all ZO calculations:

The difference is determined from the set center point (_CPA and _CPO) and the calculated center point.

Depending on the definition of _KNUM, no automatic ZO entry is performed or, alternatively, the measuring axis difference is added in the specified ZO memory. If a fine offset is active (MD 18600: MM_FRAME_FINE_TRANS), an additive ZO will be implemented in it, otherwise it is implemented in the coarse offset.

Precondition

The probe must be called **with** tool length offset.

Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Parameters

_MVAR	101	ZO calculation in a hole with ZO compensation
	102	ZO calculation on a shaft with ZO compensation
	103	ZO calculation in a groove with ZO compensation
	104	ZO calculation on a web with ZO compensation
_SETVAL	REAL	Setpoint for diameter/width
_CPA	REAL	Center point abscissa (referred to workpiece zero)
_CPO	REAL	Center point ordinate (referred to workpiece zero)
_STA1	0...360 degrees	Starting angle
_ID	REAL	Incremental infeed of applicate with leading sign (only for ZO calculation on a web)
_INCA	0...360 degrees	Indexing angle (only for ZO calculation in hole or on shaft)
_KNUM	0 no automatic ZO compensation;	With/without automatic ZO calculation
	1...99 automatic additive ZO compensation in G54...G57, G505...G599	
Meas. cycles ≥ SW 4.4	1000 automatic additive ZO compensation in channel-specific basic frame[1)]	
Meas. cycles ≥ SW 6.2[2)]	1011...1026 automatic ZO compensation in 1st to 16th basic frame (channel)	($P_CHBFR[0]...$P_CHBFR[15])
	1051...1066 automatic ZO compensation in 1st to 16th basic frame (global)	($P_NCBFR[0]...$P_NCBFR[15])
	2000 automatic ZO compensation in system frame	scratching system frame ($P_SETFR)
	9999 automatic ZO compensation in an active frame	settable frames G54..G57, G505...G599 or for G500 in last active basic frame according to $P_CHBFRMASK (most significant bit)

1) As of measuring cycles SW 5.3, compensation is carried out in the last basic frame (per MD 28081: MM_NUM_BASE_FRAMES) if more than one is available. If measuring cycles higher than SW 5.3 are used at a control with SW 4, parameter _SI[1] in the GUD 6 module must be set to 4!

2) **Measuring cycles version SW 6.2 and higher can only be used with NCK-SW 6.3 and higher!!**

These following additional parameters are also valid:
_VMS, _RF, _CORA, _TSA, _FA, _PRNUM and **_NMSP**.
See Sections 2.2 and 2.3.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

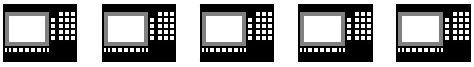
Procedure

Position before the cycle is called
The probe must be positioned facing P1 and the probe ball at the measurement level.

Position after the measuring cycle has terminated with ZO calculation in hole or on shaft
On completion of the measuring process, the probe is positioned facing P3 (or P4 for four-point measurement) at a distance corresponding to _FA · 1 mm.

Position after the measuring cycle has terminated with ZO calculation in groove or on shaft
On completion of the measuring process, the probe is positioned facing P2 at a distance corresponding to _FA · 1 mm.

Notice!
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane, orientation of the spindle in the plane and measuring velocity are the same for both measurement and calibration. Deviations can cause additional measuring errors.



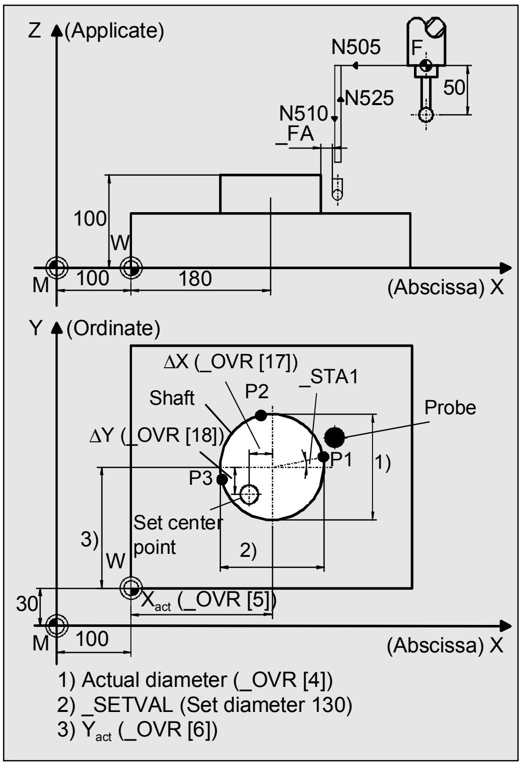
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

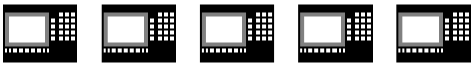
ZO calculation of a shaft with CYCLE979

The ZO is to be checked on a workpiece. Any deviation from the selected ZO must be compensated for automatically by means of additive ZO.
The maximum conceivable deviation from the center point of the shaft is taken as 1 mm in both axes. The measuring path is programmed with 2 mm (max. measuring path = 4 mm) to ensure a minimum measuring path of 1 mm up to the edge of the shaft.
The center point of the shaft lies at X180 Y130. The start angle is 10°, the following angle 90°. Points P1, P2 and P3 are measured. Traversing between the points is carried out with a circular feedrate of 1000 mm/min.

Automatic compensation is performed in G54, X and Y by the calculated difference between the actual value and set position of the shaft center, should it be less than 1 mm (_TSA) in both axes. Otherwise alarm "Safe area violated" is output and program execution cannot be continued.



OFFSET_SHAFT	
N500 G54 T9	Select T No. probe
N505 G17 G0 G90 X260 Y170	Position probe in X/Y plane in vicinity of P1
N510 Z40 D1	Position Z axis at P1 level and select tool offset
N515_MVAR=102 _SETVAL=130 _CPA=180 _CPO=130 _STA1=10 _INCA=90 _RF=1000 _KNUM=1 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2	Set parameters for measuring cycle call
N520 CYCLE979	Call measuring cycle for ZO calculation in X/Y
N525 G0 Z160	Run up Z axis
N530 G54	Renewed call of the zero offset G45 so that the changes take effect through the measuring cycle!
N570 M30	End of program



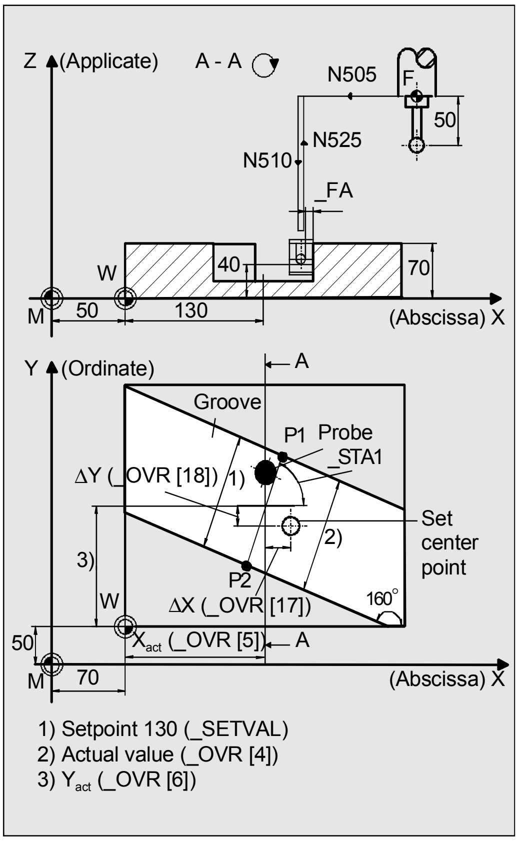
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

ZO calculation of a groove with CYCLE979

The ZO is to be checked on a workpiece. Any deviation from the selected ZO must be compensated for automatically by means of additive ZO.
The maximum conceivable deviation of the groove center is taken as 1 mm. The measuring path is therefore specified as 2 mm (max. measuring path = 4 mm) and this ensures that there is still a minimum measuring path of 1 mm up to the edge of the groove.
The groove center lies at X150 Y130. The start angle is 70°.

Automatic compensation is performed in G55, X and Y by the calculated difference between the actual value and set position of the groove center, should it be less than 1 mm [_TSA] in both axes. Otherwise alarm "Safe area violated" is output and program execution cannot be continued.



1) Setpoint 130 (_SETVAL)
2) Actual value (_OVR [4])
3) Y_{act} (_OVR [6])

OFFSET_OF_GROOVE	
N500 G55 T9	Select T No. probe
N505 G17 G0 G90 X150 Y180	Position probe in X/Y plane in vicinity of P1
N510 Z40 D1	Position Z axis at P1 level and select tool offset
N515_MVAR=103 _SETVAL=130 _CPA=150 _CPO=130 _STA1=70 _KNUM=2 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2	Set parameters for measuring cycle call
N520 CYCLE979	Call measuring cycle for ZO calculation in X/Y
N525 G0 Z160	Run up Z axis
N530 G55	Renewed call of the zero offset G55 so that the changes take effect through the measuring cycle!
N570 M30	End of program

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

5.7 CYCLE998 Angular measurement (ZO calculation)

Programming

CYCLE998

Function

This cycle makes it possible to determine the angular position of a workpiece relative to the set angle value _STA 1 to the offset axis.

An empirical value stored in the GUD5 module is subsequently taken into account with the correct sign.

The multiplying factor for measurement path 1 mm makes it possible to take into account the scatter band of the blanks (set value).

Depending on the definition of _KNUM either no automatic ZO compensation is performed, or the difference between the actual and setpoint value of the angle is added to the specified ZO memory **of the rotary axis**. If a fine offset is active (MD 18600: MM_FRAME_FINE_TRANS), an additive ZO will be implemented in it, otherwise it is implemented in the coarse offset.

This cycle can also be used to perform differential measurements.

In Measuring cycles SW 4.4 and higher, the angular difference can be added to the rotary component of the specified ZO memory (coordinate rotation).

2 angle measurement for measuring cycle SW 6.2 and higher

With measuring variants _MVAR=106 and _MVAR=100106 it is possible calculated and correct the angular position of an oblique plane on a workpiece by measuring 3 points. The angles refer to rotation about the axes or the active plane G17 to G19.
Otherwise, the same conditions apply as for simple angle measurement.
Additional data are required for the setpoint input of the 2nd angle. A ZO is implemented in the rotary part of the set ZO memory (coordinate rotation).

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Result parameters

CYCLE998 makes the following values available as results in the GUD5 module:

_OVR [0]	REAL	Setpoint angle/setpoint angle between workpiece area and 1st axis of the plane (abscissa) of the active WCS[1)]
_OVR [1][1)]	REAL	Setpoint angle between workpiece area and 2nd axis of the plane (ordinate) of the active WCS
_OVR [4]	REAL	Actual value angle/actual value angle between workpiece area and 1st axis of the plane (abscissa) of the active WCS[1)]
_OVR [5][1)]	REAL	Actual value angle between workpiece area and 2nd axis of the plane (ordinate) of the active WCS
_OVR [16]	REAL	Difference angle/difference angle about 1st axis of the plane[1)]
_OVR [17][1)]	REAL	Difference angle about 2nd axis of the plane
_OVR [20]	REAL	Offset value angle
_OVR [21][1)]	REAL	Offset value angle about 1st axis of the plane
_OVR [22][1)]	REAL	Offset value angle about 2nd axis of the plane
_OVR [23][1)]	REAL	Offset value angle about 3rd axis of the plane
_OVR [28]	REAL	Safe area
_OVR [30]	REAL	Empirical value
_OVI [0]	INTEGER	ZO number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [5]	INTEGER	Measuring probe number
_OVI [7]	INTEGER	Empirical value memory number
_OVI [9]	INTEGER	Alarm number
_OVI [11][2)]	INTEGER	Status offset request
_OVI [12][2)]	INTEGER	Internal error number of the measure function

1) As of measuring cycles SW 6.2 and higher; measuring variant _MVAR=x00106 only
2) For measuring cycle SW 6.2 and higher; only for zero offset

Differential measurement

Differential measurement means that measuring point 1 is measured twice with a spindle reversal of 180°, i.e. rotation of probe through 180 degrees. This determines the trigger point for the measuring direction and the positional deviation when measuring in the plane in the measuring axis and stores it in the GUD6 module.

An uncalibrated probe can therefore be used for the measurement.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Precondition for differential measurement

- Spindle orientation (with previously programmed SPOS command) by NC
- Bidirectional/multidirectional probe
- Random positioning of probe in spindle between 0° and 360° (min. every 90°) (all-round coverage).

Preconditions for angular measurement

- The probe must be positioned **with** tool length offset and opposite the 1st measuring point.
- Tool type 1x0 or 710 (3D probe) for measuring cycles SW 4 and higher is permitted.
  As of measuring cycles SW 5.4, tool type 500 and as of measuring cycles SW 6.2 also 580 with tool edge positions 5 to 8 is also possible under the conditions stated in Section 5.1.
- Parameter _ID is used to specify the distance in the offset axis between MP1 and MP2 (positive values only).
- The cycle is capable of measuring a maximum angle of -45°...45°. However, the measurement can be taken from all sides.
- The angle between the offset axis and the workpiece edge is defined as the setpoint angle. The setpoint has a negative sign in the clockwise direction and a positive sign in the counterclockwise direction.

Applicable probe types

The measuring cycle operates with the following probe types which are coded via parameter _PRNUM:

- Multidirectional probe
- Bidirectional probe
- Monodirectional probe

Monodirectional probes must be calibrated! These probes cannot be used to take differential measurements!

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Measurement variants

CYCLE998 permits the following measurement variants which are specified via parameter _MVAR.

Value	*Measurement variant*
105	Angle measurement, ZO calculation
1105	Angle measurement with differential measurement, ZO calculation
1001051)	Angle measurement, ZO calculation with paraxial positioning of measuring point 1 to measuring point 2 in the offset axis
1011051)	Angle measurement with difference measurement, ZO calculation and paraxial positioning of measuring point 1 to measuring point 2 in the offset axis
1061)	2 angle measurement, ZO calculation
1001061)	2 angle measurement, ZO calculation with paraxial positioning between measuring points 1, 2, 3

When differential measurement (_MVAR=1105) is selected **only** MP1 is measured twice.

1) Measuring cycles SW 6.2. and higher

Parameters

_MVAR	105	Angular measurement ZO calculation with ZO compensation
	1105	Angular measurement with differential measurement, ZO calculation with ZO compensation
Meas. cycles ≥ SW 6.2[2]	106	2 angle measurement (oblique plane), ZO calculation with ZO
	100105 or 100106	6th digit=1: Measuring point to measuring point positioning is paraxial
	105 or 106	6th digit=0: Positioning is effected taking account of the set angle and distances and the deviation permissible in _TSA
_SETVAL	REAL	Setpoint (axis position) in measuring point 1 in the measuring axis For measurement variant 106: expected position on the workpiece surface in measuring point P1 on the applicate (no meaning if _MVAR=1xx10x)
_STA1	REAL	Setpoint angle or angle about 1st axis of the plane
_INCA Meas. cycles ≥ SW 6.2[2]	REAL	Only if _MVAR=x00106: Setpoint angle about 2nd axis of the plane
_MA	102 or 201 102...302 (Measuring cycles SW 4.4 and higher)	Number of offset axis and measuring axis If _MVAR=x00106: No entry required, applicate is always measuring axis
_MD Meas. cycles ≥ SW 6.2[2]	0 positive measuring direction 1 negative measuring direction	Measuring direction in the measuring axis (only relevant if _MVAR=10x10x)

840 D	840 D	FM-NC	810 D	840 Di
NCU 571	NCU 572 NCU 573			

_ID	REAL (without sign)	Distance between measuring points P1 and P2 in the offset axis If _MVAR=x00106: Distance between measuring points P1 and P2 in the 1st axis of the plane (abscissa)
_SETV[0] Meas. cycles ≥ SW 6.2[2)]	REAL (without sign)	Only if _MVAR=x00106: Distance between measuring points P1 and P3 in the 2nd axis of the plane (ordinate)
_RA Meas. cycles ≥ SW 4.4[2)]	0	Compensation is performed in the rotary component of the ZO compensation defined in _KNUM
	>0	Number of the rotary axis, Compensation implemented in the translation part defined in _KNUM of the rotary axis determined by _RA (not if _MVAR=x00106)
_KNUM	0 no automatic ZO compensation; 1...99 automatic ZO compensation in G54...G57, G505...G599	With/without automatic ZO calculation
Meas. cycles ≥ SW 4.4[2)]	1000 automatic additive ZO compensation in channel-specific basic frame[1)]	
Meas. cycles ≥ SW 6.2[2]	1011...1026 automatic ZO compensation in 1st to 16th basic frame (channel)	($P_CHBFR[0]...$P_CHBFR[15])
	1051...1066 automatic ZO compensation in 1st to 16th basic frame (global)	($P_NCBFR[0]...$P_NCBFR[15])
	2000 automatic ZO compensation in system frame	scratching system frame ($P_SETFR)
	9999 automatic ZO compensation in an active frame	settable frames G54..G57, G505...G599 or for G500 in last active basic frame according to $P_CHBFRMASK (most significant bit)

1) As of measuring cycles SW 5.3, compensation is carried out in the last channel-specific basic frame (per MD 28081: MM_NUM_BASE_FRAMES) if more than one is available. If measuring cycles higher than SW 5.3 are used at a control with SW 4, parameter _SI[1] in the GUD 6 module must be set to 4!

2) **Measuring cycles version SW 6.2 and higher can only be used with NCK-SW 6.3 and higher.**

3) Only for angle measurement with offset in the ZO memory of a rotary axis.

These following additional parameters are also valid:

_VMS, _CORA, _TSA, _FA, _PRNUM, _EVNUM and **_NMSP**.

See Sections 2.2 and 2.3.

Notice!

Precise angle definition requires a minimum surface finish at least at the measuring points. The distances between the measuring points must be selected as large as possible.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Procedure

Position before the cycle is called

Before measuring cycle is called, the probe must be positioned with respect to the 1st measuring point.

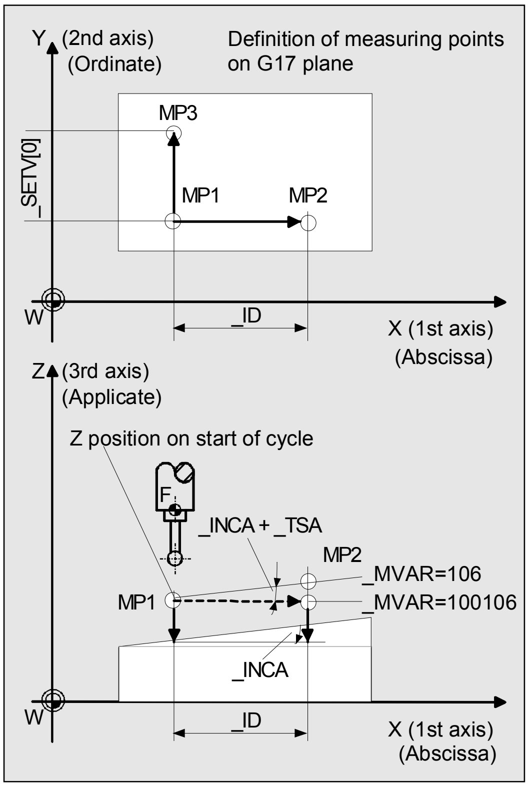
Position after the cycle has terminated

On completion of the measurement, the probe is positioned facing the measurement surface at a distance corresponding to _FA · 1 mm.

Procedure for 2 angle measurement

Position before the cycle is called

Before the cycle is called, the probe must be positioned over the 1st measuring point (MP1) in the plane and at the appropriate depth in the applicate. The meas. axis is always the applicate. MP1 must be selected in the plane such that _ID and _SETV[0] result in positive values.



Further procedure

- If _MVAR=106:
  After the measurement as been performed in MP1, positioning for MP2 is performed in the 1st and 3rd axis of the plane (for G17 in X and Z), taking the angle between the workpiece surface and the 2nd axis of the _INCA plane and the maximum deviation in _TSA into account. After the measurement has been performed in MP2, repositioning to MP1 is performed by the same path. Then positioning is performed from MP1 to MP3 in the 2nd and 3rd axis of the plane (for G17 in Y and Z), taking the angle between the workpiece surface and the 1st axis of plane _STA1 and maximum deviation in _TSA into account, and measuring is performed.
- If _MVAR=100106:
  Positioning of MP1 to MP2 is only performed in the 1st axis of the plane, from MP1 to MP3 in the 2nd axis of the plane. MP2 or MP3 must therefore be accessible collision-free with the initial position in the 3rd axis of the plane (for G17 in Z) from MP1.

Position after the cycle has terminated

After completion of the measuring operation, the probe will always be amount _FA (MVAR=106) above the 3rd measuring point in the applicate or, if _ MVAR= 100106, at the initial height (positioning height).

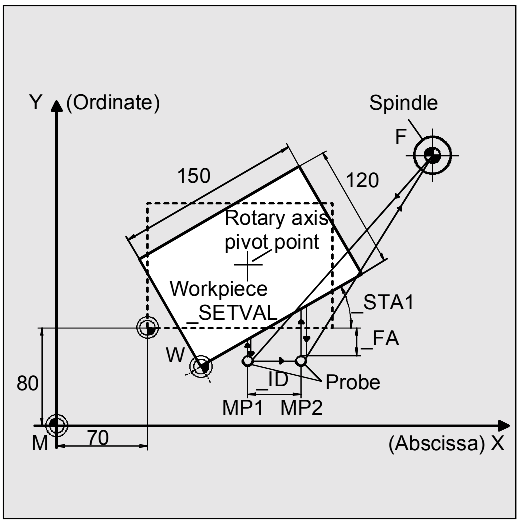
840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Programming example 1

Angular measurement with CYCLE998

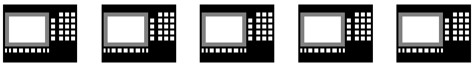
Probe length (Z axis) in TO memory T9 D1 (value 50).

The workpiece clamped on a rotary table should be positioned so that its edges lie parallel to the X and Y axes. An angular deviation detected is to be compensated automatically through additive ZO compensation of the rotary axes. The maximum possible angular deviation is taken as 5°. The measuring path is programmed with 5 mm (max. measuring path = 10 mm). The rotary table is the 4th axis in the channel.



Measurement is performed in the Y direction, offset in the X direction.
The cycle determines the measuring direction from the actual position in the Y direction and _SETVAL.
Automatic compensation is performed in the G54 ZO memory of the rotary axis.

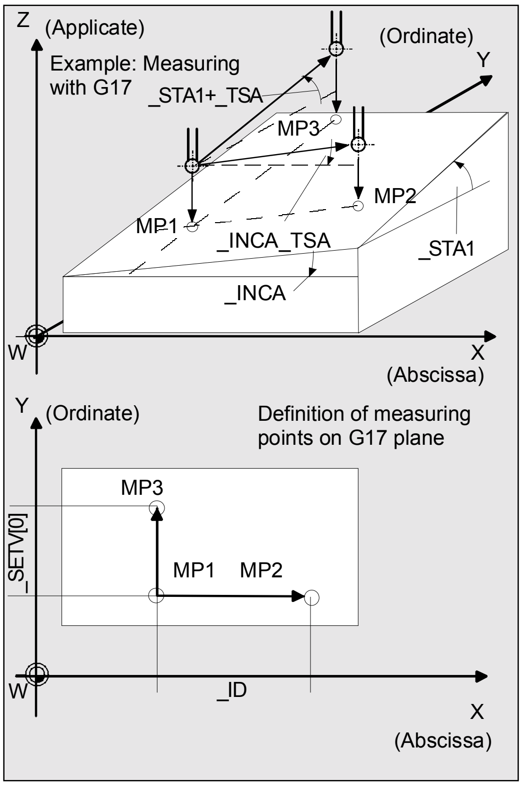
ANGLE_MEASUREMENT	
N500 G54 T9	Select T No. probe
N502 G0 C0	Position rotary table at 0°
N505 G17 G90 X70 Y-10	Position probe in X/Y plane opposite measuring point
N510 Z40 D1	Position Z axis at measuring point level and select tool offset
N515_MVAR=105 _SETVAL=0 _MA=102 _ID=40 _RA=4 _KNUM=1 _STA1=0 _TSA=5 _PRNUM=1 _VMS=0 _NMSP=1 _FA=5 _EVNUM=0	Set parameters for measuring cycle call
N520 CYCLE998	Measuring cycle for angle measurement
N525 G0 Z160	Run up Z axis
N530 G54 C0	Renewed called of the zero offset G45 so that the changes take effect through the measuring cycle! Position rotary table at 0° (edge is now setup).
N570 M30	End of program



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example 2

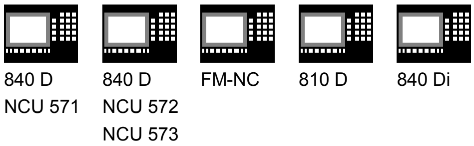
2 angle measurement with CYCLE988
(determining an oblique plane in space)



Probe length (Z axis) in TO memory T9 D1 (value 50).

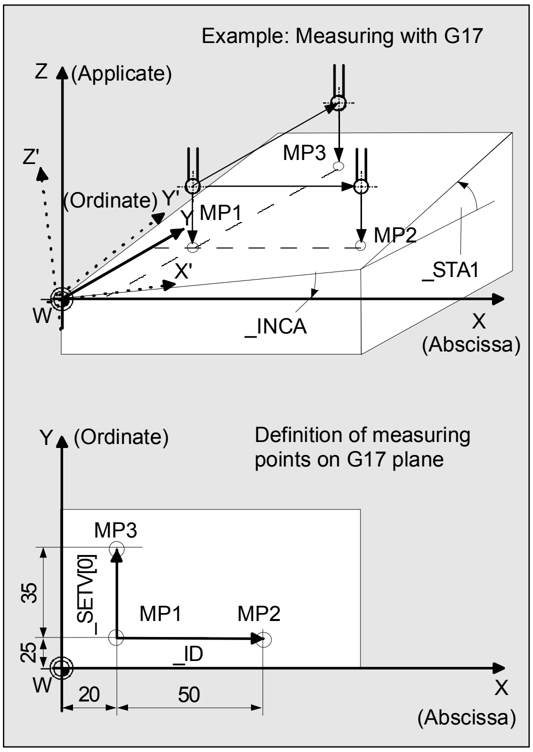
The task is to check the angular position of a machined oblique surface on a workpiece.
The result is taken from the result parameters for evaluation.
Measuring point (MP) 1 must be set such that MP2 in the ordinate (for G17: Y axis) has the same value as MP1 and the abscissa value (_ID) is positive. Moreover, MP3 in the abscissa (for G17: X axis) must have the same value as MP1. The ordinate value (_SETV[0]) must be positive.
Positioning in the applicate must be performed parallel with the oblique plane (set angle).
The machined oblique plane has set angle about Y: 12 degrees (_INCA) and about X: 8 degrees (_STA1).

OBLIQUE_MEASUREMENT	
N500 G54 T9	Select T No. probe
N505 G17 G90 X70 Y-10	Position probe in X/Y plane above measuring point
N510 Z40 D1	Position Z axis at measuring point level and select tool offset
N515 _MVAR=106 _SETV[0]=30 _ID=40 _KNUM=0 _RA=0 _STA1=8 _INCA=12 _TSA=5 _PRNUM=1 _VMS=0 _NMSP=1 _FA=5 _EVNUM=0	Set parameters for measuring cycle call
N520 CYCLE998	Measuring cycle for measuring the oblique plane
N530 G0 Z160	Run up Z axis
N540 M30	End of program



Programming example 3

Orientation of an oblique workpiece surface for remachining using the swivel cycle CYCLE800



Initial state

- The workpiece is clamped on the swivel table (assuming a swiveling workpiece holder) and aligned roughly paraxially to the machine axes.
- The swivel table is in its home position.
- The probe is in place and positioned in JOG mode approximately 20 mm above the front left corner of the workpiece to be set up.
- The scratch function is used to define the zero point of the required ZO G56 at which the 2 angle measurement is to be performed and the G17 machining plane is defined as X0 Y0 Z20.

Remachining will be performed with G57 active.

Procedure

- CYCLE998 (2 angle measurement) measures the oblique workpiece surface and an offset is entered in the rotation part of the ZO memory G57.
- After CYCLE800 has been called, axes X, Y, and Z and the rotary axes involved are positioned such that the probe is perpendicular above the oblique workpiece surface.
- Subsequent measurement with ZO in the Z' direction with CYCLE978 zeroes the workpiece surface in the Z' direction.
- Determining the angular position of the front workpiece edge with respect to the X' direction and offset in the ZO memory G57 with CYCLE998 aligns the front edge paraxially with the X' direction.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

- Then the workpiece zero is precisely defined in the plane by measuring with the ZO in the +X' direction and +Y' direction with CYCLE978.
- After that, remachining can begin on the setup surface.

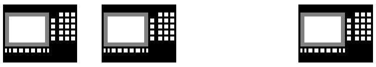
Set up plane	
N500 G56 G17	Select ZO and machining plane
N505 T9 D1	Select probe and activate tool offset
N510 CYCLE800(1,"",0,57,0,0,0,0,0,0,0,0,0,-1)	Align swivel table
N520 $P_UIFR[4] = $P_UIFR[3]	Copy the data of the ZO memory G56 to G57
N530 G1 F500 X20 Y25	Approach 1st MP for 2 angle measurement in the plane
N540 Z40	Positioning height in Z, in which all 3 MPs can be approached
N550 _VMS=0 _PRNUM=1 _TSA=20 _EVNUM=0 _NMSP=1 _FA=40 _STA1=0 _INCA=0 _MVAR=100106 _MD=1 _ID=50 _SETV[0]=35 _KNUM=4	Measuring velocity 300 mm/min, data field 1 for probe, safe area 20°, without empirical value, number of measurements at same position =1, measurement path 40 mm, angles 1 and 2 =0, 2 angle measurement with paraxial positioning, measurement in the minus direction, distance in X between MP1 and MP2 50 mm, distance in Y between MP1 and MP3 35 mm, ZO in G57
N560 CYCLE998	Call measuring cycle
N570 G57	Activate ZO G57
N580 CYCLE800(1,"",0,57,0,0,0,0,0,0,0,0,0,-1)	Align swivel table, probe is perpendicular above oblique surface
N590 X20 Y25	Approach 1st MP in the plane
N600 Z20	Lower in Z' about 20 mm above surface
N610 _MVAR=100 _SETVAL=0 _MA=3 _TSA=10 _FA=20 _KNUM=4	ZO calculation on surface, setpoint 0, measuring axis Z', safe area 10 mm, measurement path 20 mm before and after expected switching position, ZO in G57
N620 CYCLE978	ZO calculation on surface in Z' axis for placing the zero in Z'
N625 G57	Activate the changed zero offset
N630 X20 Y-20	Place in plane before the front edge

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

N640 Z-5	Place lower in Z' direction to align the front edge in the X' direction
N650 _MVAR=105 _MA=102 _SETVAL=0 _RA=0 _STA1=0	Angle measurement measuring axis Y', displacement in X' axis, distance between measuring points 50 mm; offset in the rotation part of the ZO memory G57, set angle between edge and X' direction 0
N660 CYCLE998	Angle measurement by measuring in Y' and displacement between the 2 measuring points in X' with offset in G57
N665 G57	Activate the changed ZO G57
N680 X20 Y-20	
N690 Z-5	Place at measuring height before the front edge
N700 _MVAR=100 _MA=2 _SETVAL=0 _FA=10	ZO calculation on surface, measurement in Y' direction, measurement path 10 mm in front of to 10 mm behind expected edge
N710 CYCLE978	ZO calculation on surface with measurement in +Y' direction and ZO in G57 for setting the zero in Y'
N720 G57	Activate the changed ZO G57
N730 X-20 Y-20	
N740 Y25	Place in front of the left edge
N750 _MA=1	Measure in +X'
N760 CYCLE978	ZO calculation on surface, measurement in +X' direction, and ZO offset in G57 memory. Measurement path 10 mm in front of to 10 mm behind expected edge for setting zero in X'
N770 G57	Activate the changed ZO G57
N780 Z20	Raise in Z
. . .	The oblique surface is now completely set up
N1000 M2	End of program

Comment about CYCLE800

The swivel cycle CYCLE800 is used to measure and operate on any surface by converting the active workpiece zero and the active tool offset to the oblique surface in the cycle by calling the relevant NC functions, taking account of the kinematic chain of the machine, and positioning the rotary axes.



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

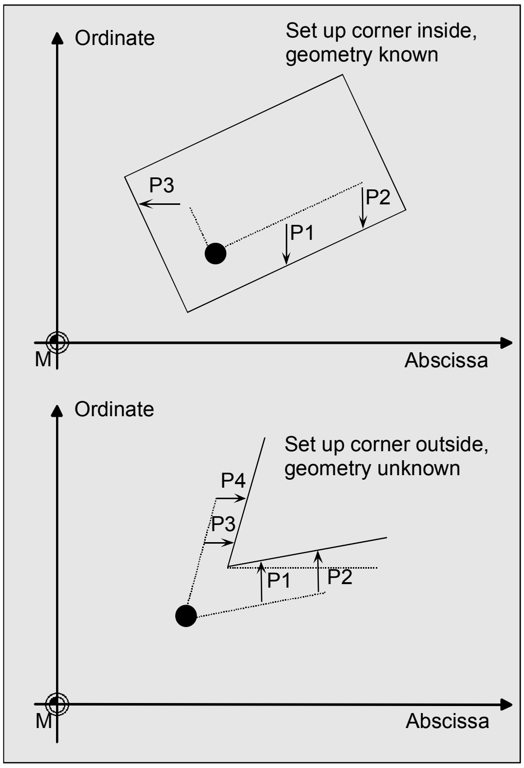
5.8 CYCLE961 Automatic setup of inside and outside corner

5.8.1 Automatic setup of corner with distances and angles specified

Programming

CYCLE961

Function



The cycle approaches either 3 (a rectangle if the workpiece geometry is known) or 4 measuring points (if the workpiece geometry is not known) and calculates the point of intersection of the resulting straight lines and the angle of rotation to the positive abscissa axis of the current plane. If the workpiece geometry is known (precondition is a rectangle) the corner to be calculated can be selected. The result is stored as an absolute value in the corresponding zero offsets of the axes (see result parameters).

The measuring points are approached paraxially. With **Set up corner inside,** the cycle only travels in one plane; on intermediate positioning from one measuring point to the other, no probe retraction movement is generated. With **Set up corner outside,** the corner is either traversed by the shortest path or bypassed in the plane.

Precondition

The probe must be called with tool length offset.
Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in Section 5.1.

Before the cycle is called the probe is positioned at measuring depth opposite the corner to be measured. It must be possible to approach the measuring points without danger of collision (no obstacle at measuring depth).

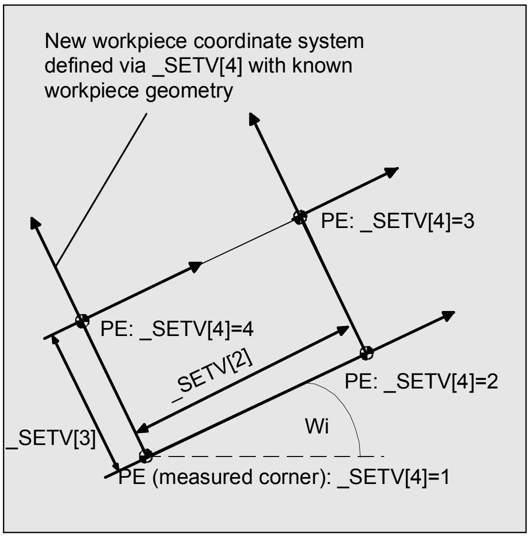
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Result parameters

Results: Set up corner automatically

1. Corner point PE
2. Angle Wi

Measuring cycle CYCLE961 supplies the following values as results in the GUD5 module:



_OVR [4]	REAL	Wi (angle to abscissa axis) in the workpiece coordinate system (WCS)
_OVR [5]	REAL	Abscissa PE (actual value corner point in the abscissa) in WCS
_OVR [6]	REAL	Ordinate PE (actual value corner point in the ordinate) in WCS
_OVR [20]	REAL	Wi (angle to abscissa axis) in the machine coordinate system (MCS)
_OVR [21]	REAL	Abscissa PE (actual value corner point in the abscissa) in MCS
_OVR [22]	REAL	Ordinate PE (actual value corner point in the ordinate) in MCS
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Measuring probe number
_OVI [9]	INTEGER	Alarm number

Explanation

Compensation of the zero offset

When _KNUM=0, no settable zero offset is corrected. When _KNUM <> 0, the corresponding zero offset for the abscissa and ordinate is calculated in such a way that the calculated corner point becomes the workpiece zero. The rotary component for the applicate (in Z for G17) is offset in such a way that the workpiece coordinate system lies in the plane parallel to edge 1.

The offset is implemented in the coarse offset, if a fine offset is active (MD18600: MM_FRAME_FINE_TRANS) it will be reset.

5.8 CYCLE961 Automatic setup of inside and outside corner

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Parameters

_MVAR	105		Set up corner inside at rectangle (geometry known, 3 measuring points)
	106		Set up corner outside at rectangle (geometry known, 3 measuring points)
	107		Set up corner inside (geometry unknown, 4 measuring points)
	108		Set up corner outside (geometry unknown, 4 measuring points)
_FA	REAL		Measuring path, only included if calculated as larger than internal value
_KNUM	0	No automatic ZO compensation;	with/without ZO compensation No. of the zero offset in which the calculated offset and the angle of rotation are stored
	1...99	Autom. ZO compensation in G54...G57, G505...G599	
Meas. cycles ≥ SW 4.4[2]	1000	Automatic additive ZO compensation in channel-specific basic frame[1)]	
Meas. cycles ≥ SW 6.2[2)]	1011...1026	automatic ZO compensation in 1st to 16th basic frame (channel)	($P_CHBFR[0]...$P_CHBFR[15])
	2000	automatic ZO compensation in system frame	scratching system frame ($P_SETFR)
	9999	automatic ZO compensation in an active frame	settable frames G54..G57, G505...G599 or for G500 in last active basic frame according to $P_CHBFRMASK (most significant bit)
_STA1	REAL		Approximate angle from positive direction of the abscissa to the 1st edge (reference edge) in MCS: Negative value in clockwise direction; Positive direction in counterclockwise direction
_INCA	REAL		Angle from 1st edge to 2nd edge of the workpiece Negative value in clockwise direction; Positive value in counterclockwise direction
_ID	REAL		Retraction in applicate when measuring outside corner, used to overtravel the corner (when _ ID=0 the corner is bypassed) (incremental)
_SETV[0]	REAL		Distance between starting point and measuring point 2 (positive only)
_SETV[1]	REAL		Distance between starting point and measuring point 4 (positive only)
For measurement variants 105 and 106 only:			
_SETV[2]	REAL		Offset of origin in the abscissa (measured corner – next corner of rectangle with an edge length 1=_SETV[2]) in counterclockwise direction

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

_SETV[3]	REAL	Offset of origin in the ordinate (measured corner – next corner of rectangle with an edge length 2=_SETV[3]) in clockwise direction
_SETV[4]	REAL	Specification of corner point, values 1 ... 4 (counted in counterclockwise direction) 1 Measured corner 2 Next corner in counterclockwise direction 3 Opposite corner 4 Next corner in clockwise direction

1) As of measuring cycles SW 5.3, compensation is carried out in the last basic frame (per MD 28081: MM_NUM_BASE_FRAMES) if more than one is available. If measuring cycles higher than SW 5.3 are used at a control with SW 4, parameter _SI[1] in the GUD 6 module must be set to 4!

2) **Measuring cycles version SW 6.2 and higher can only be used with NCK-SW 6.3 and higher.**

These following additional parameters are also valid:
_VMS, _PRNUM and **_NMSP**.
See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called

The probe is positioned at measuring depth opposite the corner to be measured.

The measuring points are derived from the programmed distance between starting point and measuring point 2 or measuring point 4 (measuring points 1 and 3 half distance). It must be possible to approach them without collision (no obstacle at measuring depth). The measuring cycle generates the required traversing blocks and performs the measurements at the measuring points. First measuring point MP 2 is approached, then MP 1, MP 3, and then, depending on the parameterization, MP 4. The probe travels between MP 1 and MP 3 as a function of parameter _ID. If _ID=0 the corner is bypassed. If _ID>0 the probe is retracted from MP 1 in the applicate by the value parameterized in _ID and then traversed via corner MP 3.

Position after the cycle has terminated

The probe is again positioned at the starting point (at measuring depth opposite the corner to be measured).

5.8 CYCLE961 Automatic setup of inside and outside corner

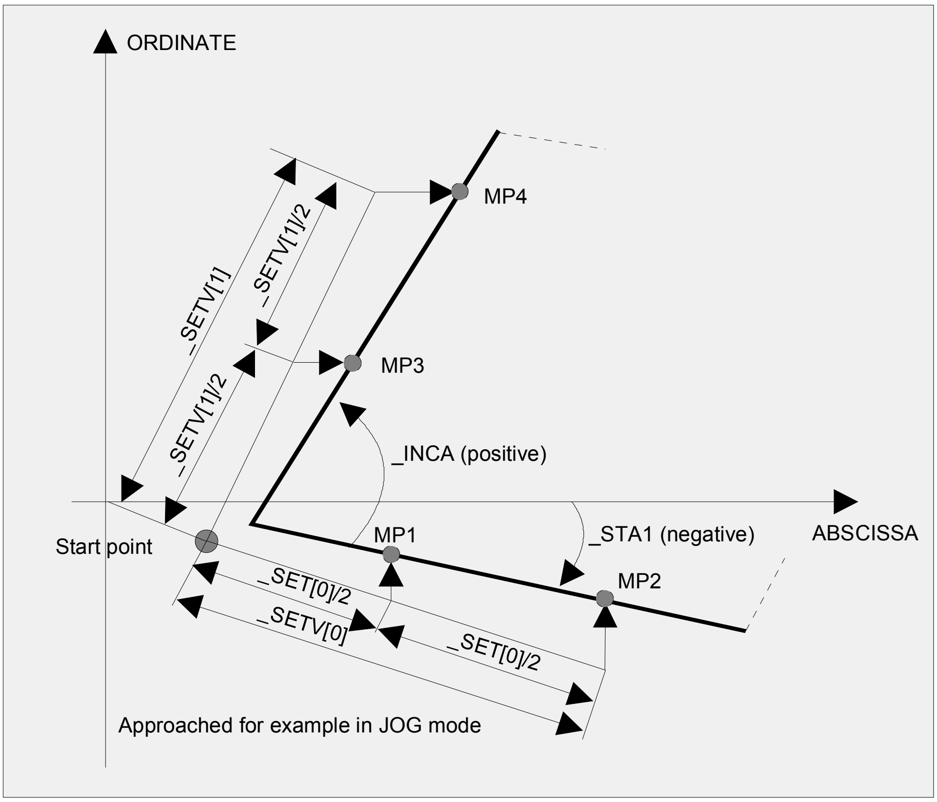
840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Programming example

The coordinates of the corner of a workpiece with unknown geometry are to be determined with an outside measurement and the zero offset G55 compensated so that the corner is the workpiece zero for active G55. The input parameters _STA1 and _INCA are estimated values. The distance to measuring points 2 and 4 is 100 mm. The corner is to be bypassed. The starting point opposite the corner that is to be set up is reached before the measuring cycle is called. It can be approached in operating modes Automatic or JOG.



SETUP_CORNER	
N100 G17 T10 D1	Select probe
N110 _MVAR=108 _FA=20 _KNUM=2 _STA1=-35 _INCA=80 _ID=30 _SETV[0]=100 _SETV[1]=100 _PRNUM=1 _VMS=0 _NMSP=1	Parameterize cycle 961
N115 CYCLE961	
N120 M30	

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

5.8.2 Automatic setup of corner by defining 4 points (measuring cycles SW 4.5 and higher)

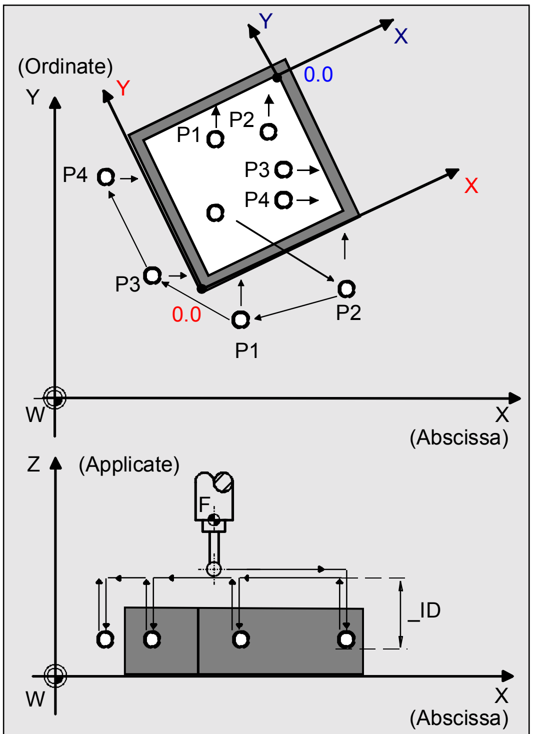
Programming

CYCLE961

Function

Points P2, P1, P3 and P4 are approached in succession in the cycle at positioning depth, from which traversing is carried out paraxially at the measured feedrate to the measuring depth against the workpiece edge.

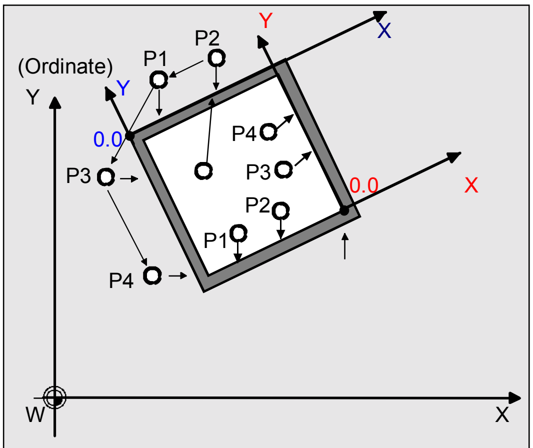
The cycle uses the relative positions of points P1 to P4 to determine the approach directions and the measuring axis. The cycle uses the measured results to compute the corner points and the angle of edge 1 (determined by measuring P2 and P1) relative to the positive abscissa axis of the current plane and enters the coordinates of the corner point and the angle of the relevant points of the _OVR[ ] field.



Position of points P1...P4 for measurement of an outside corner, bottom left, or an inside corner, top right

The position of points P1 and P2 relative to each other determines the direction of the abscissa axis (for G17 X axis) of the new coordinate system; a negative offset of P1 and P2 in the abscissa (for G17 X axis) produces an additional 108° rotation.

Precondition



Position of points P1...P4 for measurement of an outside corner, top left, or an inside corner, bottom right

The probe must be called with tool length offset.
Permissible tool types:

- 1x0 or, for measuring cycles SW 4 and higher, 710 (3D probe)
- in measuring cycle SW 5.4 and higher → 500 or in measuring cycle SW 6.2 and higher → also 580 with tool cutting edge positions 5 to 8 under the conditions stated in 5.1.

The probe lies at the positioning on which all 4 points can be approached without collision (no obstacle at positioning depth).

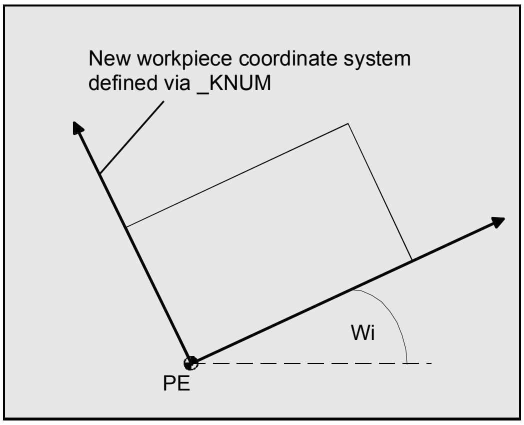
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Result parameters

Results: Set up corner automatically

1. Corner point PE
2. Angle Wi

Measuring cycle CYCLE961 supplies the following values as results in the GUD5 module:



_OVR [4]	REAL	Wi (angle to abscissa axis) in the workpiece coordinate system (WCS)
_OVR [5]	REAL	Abscissa PE (actual value corner point in the abscissa) in WCS
_OVR [6]	REAL	Ordinate PE (actual value corner point in the ordinate) in WCS
_OVR [20]	REAL	Wi (angle to abscissa axis) in the machine coordinate system (MCS)
_OVR [21]	REAL	Abscissa PE (actual value corner point in the abscissa) in MCS
_OVR [22]	REAL	Ordinate PE (actual value corner point in the ordinate) in MCS
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Measuring probe number
_OVI [9]	INTEGER	Alarm number

Explanation

Compensation of the zero offset

When _KNUM=0, no settable zero offset is corrected. When _KNUM <> 0, the corresponding zero offset for the abscissa and ordinate is calculated in such a way that the calculated corner point becomes the workpiece zero. The rotary component for the applicate (in Z for G17) is offset in such a way that the workpiece coordinate system lies in the plane parallel to edge 1.

The offset is implemented in the coarse offset, if a fine offset is active (MD18600: MM_FRAME_FINE_TRANS) it will be reset.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Parameters

_MVAR	117	Set up corner inside (4 measuring points)
	118	Set up corner outside (4 measuring points)
_FA	REAL	Measurement path
_KNUM	REAL	No. of the zero offset in which the calculated offset and the angle of rotation are stored; (or 0)
_ID	REAL	Infeed of positioning depth to measuring depth (incremental)
_SETV[0]	REAL	Abscissa P1 in active WCS
_SETV[1]	REAL	Ordinate P1 in active WCS
_SETV[2]	REAL	Abscissa P2 in active WCS
_SETV[3]	REAL	Ordinate P2 in active WCS
_SETV[4]	REAL	Abscissa P3 in active WCS
_SETV[5]	REAL	Ordinate P3 in active WCS
_SETV[6]	REAL	Abscissa P4 in active WCS
_SETV[7]	REAL	Ordinate P4 in active WCS

These following additional parameters are also valid:
_VMS, _PRNUM and **_NMSP**.

See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called
The measuring probe lies at the positioning depth.
It must be possible to approach points P1 to P4 without danger of collision. The measuring cycle generates the traversing blocks and performs the measurements at the measuring depth for points P1 to P4. Point P2 is approached first, followed by P1, P3 and P4.

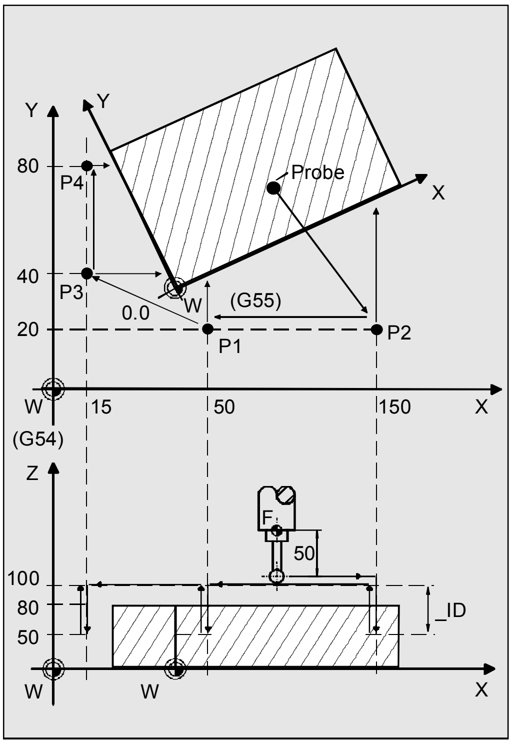
Position after the cycle has terminated
The probe is at the positioning depth at point P4.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Programming example

The coordinates of the corner of a workpiece are to be determined by outside measurement. The ZO memory G55 must be compensated so that the corner point has the coordinates 0.0 on selection of G55. Probe length (Z axis) in TO memory T9D1 (value 50).
The visual representation corresponds to _CBIT[14]=0, i. e. length of the probe relative to the center of the probe ball!
The measurement is carried out in the G17 plane with active G54. The coordinates of points P1...P4, from which the workpiece can be traversed parallel to the axis, are as follows

P1.x=50 P1.y=20
P2.x=150 P2.y=20
P3.x=15 P3.y=40
P4.x=15 P4.y=80



CORNER_SETUP_1	
N500 G54 T9	Select T No. probe
N505 G17 G0 Z100 D1	Position probe at positioning depth, activate probe length
N510 X100 Y70	Position probe above workpiece in the X/Y plane
N515 _MVAR=118 _SETV[0]=50 _SETV[1]=20 _SETV[2]=150 _SETV[3]=20 _SETV[4]=15 _SETV[5]=40 _SETV[6]=15 _SETV[7]=80 _ID=-50	Measurement variant measure corner outside Coordinates of P1...P4 Infeed to measurement depth
N520 _VMS=0 _NMSP=1 _PRNUM=2 _FA=100 _KNUM=2	Measurement path 100 mm to expected edge (max. measurement path=200 mm)
N525 CYCLE961	Cycle call
N530 G55	Call ZO G55
N535 G0 X0 Y0	Position probe in X/Y plane above corner
. . .	
N600 M30	End of program

■

Measuring Cycles for Turning Machines

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.1 General preconditions

Function

Measuring cycles are subroutines that have been kept general for solving a certain measuring problem and which are adapted to the specific problem by the input data.
The measuring cycles are created as a program package comprising the actual measuring cycles and utilities.
To be able to run the measuring cycles described in this Chapter, the following programs must be stored in the part program memory of the control.

Programming

Overview of the measuring cycles

CYCLE972	Calibrate tool probe, measure turning tools
CYCLE973	Calibrate workpiece probe in the reference groove or on any surface
CYCLE974	Single-point measurement or ZO calculation on surface, single-point measurement with reversal
CYCLE982	Calibrate a tool probe, gauge turning and milling tools (measuring cycles SW 5.3 and higher)
CYCLE994	Two-point measurement on the diameter

Overview of the auxiliary programs required

CYCLE100	Log ON
CYCLE101	Log OFF
CYCLE102	Measurement result display selection
CYCLE103	Preassignment of input data
CYCLE104	Internal subroutine
CYCLE105	Generate log contents logging
CYCLE106	Logging the sequential controller logging
CYCLE107	Output of message texts
CYCLE108	Output of alarms
CYCLE110	Internal subroutine
CYCLE111	Internal subroutine
CYCLE113	Read system date and time logging
CYCLE114	Internal subroutine (tool offset)
CYCLE115	Intern subroutine (zero offset, measuring cycle SW 6.2 and higher)
CYCLE117	Internal subroutine: Measuring functions
CYCLE118	Format real values logging

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

The two data blocks

- **GUD5.DEF**
- **GUD6.DEF**

are also required where all the data required by the measuring cycles are defined.

Procedure

Call and return conditions

The following general call and return conditions must be observed:

- D compensation containing the data of the calibration tool or the workpiece probe or the tool to be measured must be activated before a measuring cycle is called. As the tool type for the workpiece probe, type 500 is permissible and as of measuring cycles SW 6.2 also type 580 with cutting edge positions 5 to 8. With the variants for ZO calculation, a settable zero offset must be active.
- No mirroring, scale factors <> 1 or coordinate rotation must be active. As of measuring cycles SW 5.4, mirroring of workpiece measuring cycles is permissible, except for calibration (condition: MD 10610=0).
- The G functions active before the measuring cycle is called remain active after the measuring cycle call, even if they have been changed inside the measuring cycle.
- **Measuring cycles version SW 6.2 can only be used with NCK-SW 6.3 and higher.**

Plane definition

The measuring cycles work internally with the 1st axis (abscissa), 2nd axis (ordinate) and 3rd axis (applicate) of the current plane. Which plane is the current plane is set with G17, G18 or G19 before the measuring cycle is called.

Spindle handling

The measuring cycles have been written so that the spindle commands they contain always refer to the active master spindle of the control. If the measuring cycles are used on machines with several spindles, the spindle with which the cycle must work must be defined as the master spindle before the cycle is called.

References: /PG/ "Programming Guide, Fundamentals"

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.2 CYCLE972 Tool measurement

Programming

CYCLE972

Function

CYCLE972 performs calibration of a tool probe and measures tool lengths L1 and L2 for turning tools with tool edge positions 1 to 8.

Result parameters

The measuring cycle CYCLE972 returns the following values in the GUD5 module for the measurement variant **calibration:**

_OVR [8]	REAL	Trigger point in minus direction, actual value, abscissa
_OVR [10]	REAL	Trigger point in plus direction, actual value, abscissa
_OVR [12]	REAL	Trigger point in minus direction, actual value, ordinate
_OVR [14]	REAL	Trigger point in plus direction, actual value, ordinate
_OVR [9]	REAL	Trigger point in minus direction, difference, abscissa
_OVR [11]	REAL	Trigger point in plus direction, difference, abscissa
_OVR [13]	REAL	Trigger point in minus direction, difference, ordinate
_OVR [15]	REAL	Trigger point in plus direction, difference, ordinate
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Result parameters

The measuring cycle CYCLE972 returns the following result values in the GUD5 module after **tool measurement:**

_OVR [8]	REAL	Actual value length L1
_OVR [9]	REAL	Difference length L1
_OVR [10]	REAL	Actual value length L2
_OVR [11]	REAL	Difference length L2
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVR [29]	REAL	Permissible dimension difference
_OVR [30]	REAL	Empirical value
_OVI [0]	INTEGER	D number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Probe number
_OVI [7]	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	T number
_OVI [9]	INTEGER	Alarm number

Measurement variants

Measuring cycle CYCLE972 permits the following measurement variants which are specified via parameter _MVAR.

Value	*Meaning*
0	Tool probe calibration
1	Tool measurement

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.2.1 CYCLE972 Calibrating the tool probe

Function

The cycle uses the calibration tool to ascertain the current distance dimensions between the machine zero and the probe trigger point and automatically loads them into the appropriate data area in the GUD6 module. They are always calculated without empirical or mean values.

Precondition
The lateral surfaces of the probe cube must be aligned parallel to the machining axes abscissa and ordinate.

The approximate coordinates of the tool probe regarding the machine zero have to be entered in the data field _TP[_PRNUM-1,0] to _TP[_PRNUM-1,3] before starting calibration. Length 1 and 2 and the radius of the calibration tool must be known exactly and stored in a tool offset data block.

This tool offset must be active when the probe is called. A turning tool must be entered as the tool type, together with tool edge position 3.

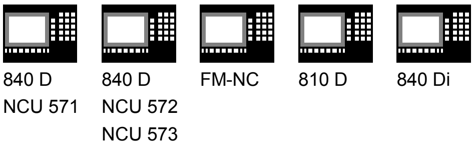
Parameters

_MVAR	0	Calibration variant: Tool probe calibration
_MA	1, 2	Measuring axis
_PRNUM	INT	Probe number

These following additional parameters are also valid:
_VMS, _TZL, _TSA, _FA and **_NMSP**.
See Sections 2.2 and 2.3.

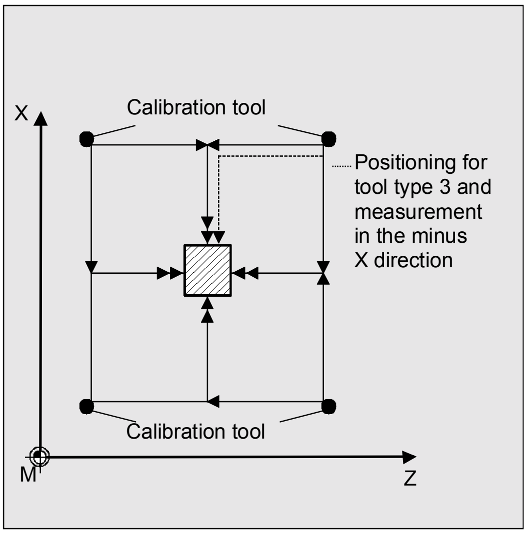
Procedure

Position before the cycle is called
The calibration tool must be prepositioned as shown in the figure. The measuring cycle then calculates the approach position itself.



Position after the cycle has terminated

On completion of the calibration process, the calibration tool is positioned facing the measuring surface at a distance corresponding to _FA · 1 mm.

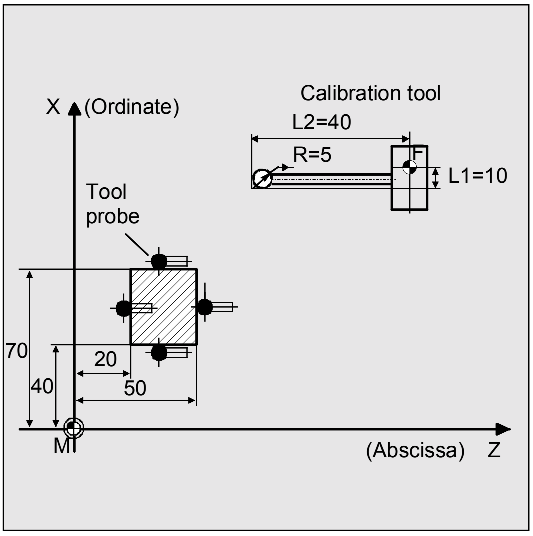


840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Calibrating the tool probe

The tool probe is stationary but provides a switching signal. The calibration tool is positioned with the turret.



Values of the calibration tool in T7 D1 in this example:

Type	500
Tool edge position	3
L1	10
L2	40
R	5

Values of tool probe 1 in GUD6 block, which were determined manually to 5 mm accuracy beforehand (relative to the machine zero):

_TP[0,0] = 50
_TP[0,1] = 20
_TP[0,2] = 70
_TP[0,3] = 40

To obtain a minimum path of 1 mm, the measuring path is programmed as 1 + 5 = 6 (max. measuring path = 12 mm).

CALIBRATE_TOOL_PROBE	
N05 G0 SUPA G94 Z300 DIAMOF	Approach any change position
N10 SUPA X240 T7 D1	Calibration tool
N20 M71	Swing in tool probe (M function is machine-specific)
N25 _MVAR=0 _MA=2 _TZL=0.004 _TSA=2 _PRNUM=1 _VMS=0 _NMSP=1 _FA=6	Parameters for calibration cycle
N30 CYCLE972	Calibration in minus X direction
N35 G0 SUPA Z60	Approach new start position
N38 _MA=1	Select different measuring axis
N40 CYCLE972	Calibration in minus Z direction
N45 G0 SUPA X30	Approach new start position
N48 _MA=2	
N50 CYCLE972	Calibration in plus X direction
N55 G0 SUPA Z0	Approach new start position
N58 _MA=1	
N60 CYCLE972	Calibration in plus Z direction
N65 G0 SUPA X240	Approach any change position axis by axis
N70 SUPA Z300	
N99 M2	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.2.2 CYCLE972 Determine dimensions of calibration tools

Function

With the following procedure it is possible to determine the dimensions of the calibration tools:

1. Enter probe data in the GUD6 module (e.g. in parameters _TP[0,0] ... _TP[0,3]) and specify the calibration tool data in tool offset (e.g. T7 D1).
2. Measure the turning tool at the presetting location.
3. Enter the tool data in the tool offset (e.g. X60) and insert the tool into the turret.
4. Machine a test part (turn to X dimension)
   Set diameter: 200.000 mm
   Actual diameter: 200.100 mm.
5. Adapt the tool offset (X59.95).
6. Turn the same test part again
   Set diameter: 195.000 mm
   Actual diameter: 195.000 mm.
7. Calibrate tool probe (see sample program in Section 6.2.1).
8. Measure the tool with CYCLE972. The value 59.95 (see step 5.) should be returned.
9. Change calibration tool X axis in D1
   Change L1 = 40 ===> to 40.95.
10. Calibrate tool probe (as for step 7.).
11. Measure tool with CYCLE972. The correct value X59.95 is then in D1. The value of the calibration tool in X is therefore O.K.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.2.3 CYCLE972 Measure tool

Function

The cycle calculates the new tool length and checks whether the corrected difference from the old tool length is within a defined tolerance range (upper limits: safe area _TSA and dimension difference check _TDIF, lower limit: zero offset area _TZL). If this range is not violated, the new tool length is accepted, otherwise an alarm is output. Violation of the lower limit is not corrected.
Empirical values can be included as an option, mean value calculation is not performed.

Precondition
The tool probe must be calibrated.
The tool to be measured must be called with tool length offset.

The tool geometry data have been entered in tool offset (tool type, tool edge position, tool nose radius, length 1, length 2).

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

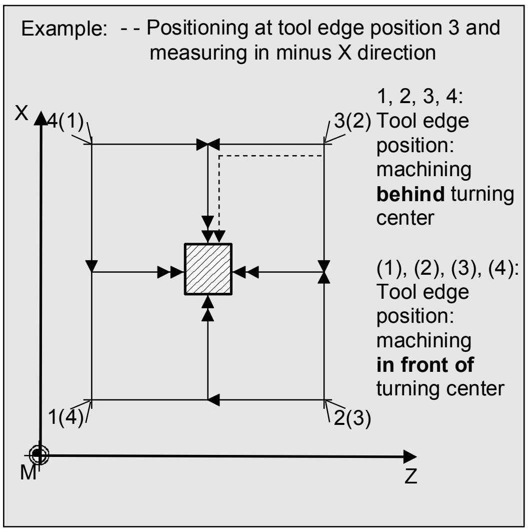
Parameters

_MVAR	1	Measurement variant: Tool measurement
_MA	1, 2	Measuring axis

These following additional parameters are also valid:
_VMS, _TZL, _TDIF, _TSA, _FA, _PRNUM,
_EVNUM, and **_NMSP**.

See Sections 2.2 and 2.3.

Procedure



Position before the cycle is called

Before the cycle is called a start position must be adopted as shown in the figure. The measuring cycle then calculates the approach position itself.

Position after the cycle has terminated

On completion of the cycle, the tool nose is positioned facing the measuring surface at a distance corresponding to _FA · 1 mm.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

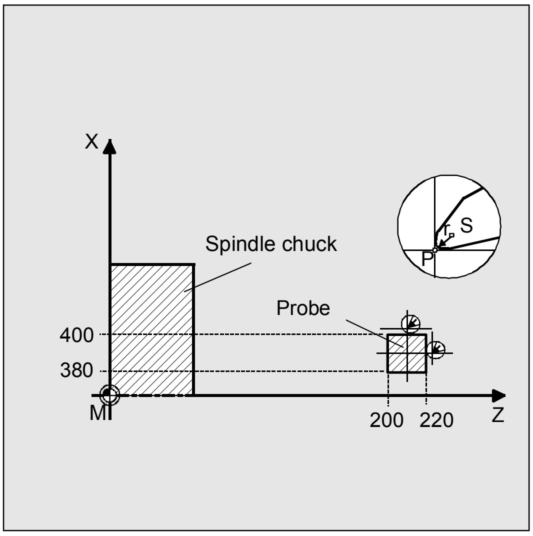
Programming example

Calibrating the tool probe with subsequent measurement of turning tool T3

The values of tool probe 1 must be preset in module GUD6 with a tolerance of approx. 1 mm, e.g.:

_TP[0,0] = 220
_TP[0,1] = 200
_TP[0,2] = 400
_TP[0,3] = 380

After calibration, the measured value (calibration value) is set.
All 4 points must be calibrated.



The dimensions of the calibration tool T7 are in lengths L1, L2 and the radius (R=5 mm) are known precisely and entered in offset field D1.
The tool edge position is 3.

The lengths and radius of tool T3 to be measured are known and entered in offset field D1. The cutting edge position is 3. The task is to measure the precise wear in both axes (adding measured value difference in the wear).

MEASURE_T3	
; Calibration:	
N1 G0 G18 DIAMOF	
N2 T7 D1	Call calibration tool
N3 SUPA Z250 X575	Start position for calibration
N5 _MVAR=0 _MA=2 _TZL=0.004 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	Parameter definition
N6 CYCLE982	Calibration in minus X direction
N7 G0 SUPA Z240	New start position
N8 _MA=1	Set other measuring axis (Z)
N9 CYLCE982	Calibration in minus Z direction
N10 G0 SUPA X360	New start position
N11_MA=2	Set other measuring axis (X)
N12 CYCLE982	Calibration in plus X direction
N13 G0 SUPA Z180	New start position
N14 _MA=1	Set other measuring axis (Z)
N15 CYLCE982	Calibration in plus Z direction
N16 G0 SUPA X575	Traverse each axis to the tool change
N17 SUPA Z520	position

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

; Measurement:	
N100 T3 D1	Selection of the tool to be measured
N110 G0 SUPA Z250 X575	Start position for measurement
N120 _MA=2 _TDIF=0.8 _MAVR=1	Change of parameter definition for measurement, otherwise calibration
N130 _CHBIT[3]=1	Offset in wear
N140 CYCLE982	Tool measurement in minus X direction (L1)
N150 G0 SUPA Z240	New start position
N160 _MA=1	Set other measuring axis (Z)
N170 CYCLE982	Tool measurement in minus Z direction (L2)
N180 G0 SUPA X575	Retraction axis by axis
N190 SUPA Z520	
N200 M2	

Explanation

Calibrate N1 to N17:

The "tip" of the calibration tool T7 is positioned in measuring axis X from the starting position at distance _FA=1 mm (dimension → with reference to the radius) before the probe. In axis Z, the probe tip center is centered with respect to the probe.

The measuring process is initiated in the negative X direction (_MA=2, starting position) with measuring velocity 150 mm/min (_VMS=0, _FA=1). The switching signal is expected by the probe 1 (_PRNUM=1) within a distance of 2 · _FA=2 mm. Otherwise, an alarm will be triggered.

Measurement is performed once (_NMSP=1).

After successful measurement, the "tip" of T7 is _FA=1 mm in front of the probe in the X direction.

The calculated probe value is entered in _TP[0,2].

Calibration with the measuring process has been completed in minus X.

Calibration is then are performed in the other measuring directions/axes.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Explanation

Measure N100 to N200:

The probe has been calibrated completely.
The "nose" of the turning tool T3 is positioned in measuring axis X from the starting position at distance _FA=1 mm (dimension → with reference to the radius) in front of the probe. In axis Z the center of the cutting edge is centered with respect to the probe. If the cutting edge radius =0, it is the tool nose.
The measuring process is initiated in the negative X direction (_MA=2, starting position) with measuring velocity 150 mm/min (_VMS=0, _FA=1). The switching signal is expected by the probe 1 (_PRNUM=1) within a distance of 2 · _FA=2 mm. Otherwise, an alarm will be triggered. Measurement is performed once (_NMSP=1).
After successful measurement, the "nose" of T3 is _FA=1 mm in front of the probe in the X direction.
The calculated length difference of L1 (tool type 5xy, _MA=2, _MVAR=xx**0**xx**1**) is summated and entered in D1 from T3 in the wear (_CHBIT[3]=1).
Measurement and wear offset are then performed in L2 in the minus Z direction.

Recommended parameters

The following parameters are suggested so that this programming example runs reliably:

- Calibration:

_TZL=0.001	Zero offset area
_TSA=1	Safe area
_FA=1	Multiplication factor for measuring path

- Measurement:

_TZL=0.001	Zero offset area
_TSA=1	Safe area for continuous operation
_FA=1	Multiplication factor for measuring path for continuous operation
_TSA=3	Safe area for setup
_FA=3	Multiplication factor for measuring path during setup
_TDIF=0.3	Dimensional difference for continuous operation
_TDIF=3	Dimensional difference during setup

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.3 CYCLE982 Tool measurement (SW 5.3 and higher)

Programming

CYCLE982

Function

Cycle CYCLE982 performs

- calibration of a tool probe,
- measures tool lengths L1 and L2 for turning tools with tool edge positions 1 to 8 (function same as CYCLE972)
- for milling tools and drills on turning machines, tool lengths;
- for mills, also the radius

For measurement of mill/drill, NCK SW 5 or higher is required.

Supports the following measuring tasks:

- **Calibration** as preparation for measurement/automatic measurement
  The switching positions of the probe are known roughly. Positioning of the calibration tool with respect to the probe is performed in the cycle.
  It is only possible to determine the switching position that is in the measuring axis (_MA) and measuring direction according to starting position. Before beginning measurement, all four switching positions of the probe must be known.
- **Measurement**
  The geometry of the tool to be measured is known roughly. Positioning of the tool with respect to the probe is performed in the cycle. The geometry is to be determined exactly, or the wear. Only measured values that are in the measurement axis (_MA=) can be calculated.
- **Automatic measurement**
  The geometry of the tool to be measured is known roughly. Positioning of the tool with respect to the probe is performed in the cycle. The geometry is to be determined exactly, or the wear. All values that can be determined are determined automatically according to tool type.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

- **Incremental calibration** as preparation for incremental measurement
  The switching positions of the probe are not known. The calibration tool must have been positioned in front of the probe manually before the cycle is called.
  It is only possible to determine the switching position that is in the measuring axis (_MA) and the stated measuring direction (_MD).
  Only the probe switching position in which the axis and direction will subsequently be measured incrementally have to be calibrated.
- **Incremental measurement**
  The geometry of the tool to be measured is not known. The tool must have been positioned in front of the probe manually before the cycle is called.
  The geometry is to be determined exactly. Only one measured value that is in the measurement axis (_MA=) can be calculated. Travel up to the probe is performed in the cycle in the measuring axis and the stated measuring direction (_MD).

Special aspects with milling tools
The tool length correction is specific to the turning machine (SD:TOOL_LENGTH_TYPE=2). The length assignment (L1, L2) is performed like for a turning tool. Measurement is possible with a rotating (M3, M4) or with a stationary milling spindle (M5). If the milling spindle is stationary, it is positioned at the specified starting angle (_STA1) at the beginning. For simple measuring tasks, this positioning with _STA1 can be suppressed. If suppression is active, measurement not requiring an SPOS-capable milling spindle is possible.
To measure a second cutting edge, you can select "measurement with reversal". This involves mean value calculation of both measured values.

Not all functions are available in SW 5.3 and higher. Certain functions require a certain SW software version of the measuring cycles and NCK. This information is given with each function.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Result parameters

The measuring cycle CYCLE982 returns the following values in the GUD5 module for the measurement variant **calibration:**

_OVR [8]	REAL	Trigger point in minus direction, actual value, abscissa
_OVR [10]	REAL	Trigger point in plus direction, actual value, abscissa
_OVR [12]	REAL	Trigger point in minus direction, actual value, ordinate
_OVR [14]	REAL	Trigger point in plus direction, actual value, ordinate
_OVR [9]	REAL	Trigger point in minus direction, difference, abscissa
_OVR [11]	REAL	Trigger point in plus direction, difference, abscissa
_OVR [13]	REAL	Trigger point in minus direction, difference, ordinate
_OVR [15]	REAL	Trigger point in plus direction, difference, ordinate
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

Result parameters

The measuring cycle CYCLE982 returns the following result values in the GUD5 module after **tool measurement:**

_OVR [8]	REAL	Actual value length L1
_OVR [9]	REAL	Difference length L1
_OVR [10]	REAL	Actual value length L2
_OVR [11]	REAL	Difference length L2
_OVR [12]	REAL	Actual value for radius
_OVR [13]	REAL	Difference for radius
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVR [29]	REAL	Permissible dimension difference
_OVR [30]	REAL	Empirical value
_OVI [0]	INTEGER	D number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measurement variant
_OVI [5]	INTEGER	Probe number
_OVI [7]	INTEGER	Empirical value memory
_OVI [8]	INTEGER	T number
_OVI [9]	INTEGER	Alarm number

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Measurement variants

Measuring cycle CYCLE982 permits the following measurement variants which are specified via parameter _MVAR.

_MVAR= (max. 6 decimal places)

Digit						*Meaning*	
6	**5**	**4**	**3**	**2**	**1**		- : No decimal point or value =0
-	-	-	-	-	**0**	**Calibrate tool probe** with calibration tool	
					1	**Measure turning or milling tool/drill**, measuring axis in _MA (specified for Turning tools: tool edge positions 1...8, Milling tools: points 3 to 5 in _MVAR)	
					2	**Automatic measurement** in abscissa and/or ordinate (specified for Turning tools: of edge positions 1...8, Milling tools: points 3 to 5 in _MVAR)	
				0		Fixed value (reserved for other functions)	
				1		Decimal place reserved – Do not use value 1	
						Significance for measuring milling tools only, also automatically:	
			0			Measuring without reversal	
			1			Measuring with reversal	
						Significance for measuring milling tools only, also automatically:	
		0			1	Correct length only	(for measuring only)
		1			1	Correct radius only	(for measuring only)
		2			1	Correct length and radius	(only for measurement, not for incremental measurement)
		3			2	Correct length and radius, travel round measuring cube opposite starting position side (for automatic measurement only)	
		4			2	Correct length and radius, measuring direction for determining length opposite to traversing direction, measuring sequence as for _MVAR=x3x02 but with different traversing motion (for automatic measurement only)	
						Significance for measuring milling tools only, also automatically:	
	0					Axial position of milling tool/drill (Radius in ordinate, for G18: X axis, SD 42950: value =2)	
	1					Radial position of milling tool/drill (Radius in abscissa, for G18: Z axis, SD 42950: value =2)	
0						Measurement and calibration	
1						Incremental measurement or calibration (measuring cycles SW 6.2 and higher) (restricted variants, no automatic measurement)	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

- The following measurement variants are not possible for incremental measurement:
  1xxxx2; 102xx1; 112xx1
- The following measurement variants are permitted if _CHBIT[20]=1 (suppression of the starting angle position with _STA1) on a milling tool:
  xxx001 (with x: 0 or 1, no other values)
- A measurement variant can also be impermissible if it cannot be performed with the stated measuring axis _MA, e.g.: The miller radius must be determined. However, with this position of the miller it is not in the measuring axis.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.3.1 CYCLE982 Calibrate tool measuring probe

Function (as described in CYCLE972)

The cycle uses the calibration tool to ascertain the current distance dimensions between the machine zero and the probe trigger point and automatically loads them into the appropriate data area in the GUD6 module. They are always calculated without empirical or mean values.

Precondition

The lateral surfaces of the probe cube must be aligned parallel to the machining axes abscissa and ordinate.

The approximate coordinates of the tool probe with respect to the machine zero have to be entered in the data field _TP[_PRNUM-1,0] to _TP[_PRNUM-1,3] in block GUD6 before starting calibration. Length 1 and 2 and the radius of the calibration tool must be stored in a tool offset data block.

This tool offset must be active when the probe is called. A **turning tool** must be specified as the tool type. The tool edge positions must be 3.

Parameters

_MVAR	0	Calibration variant: Tool probe calibration
_MA	1, 2	Measuring axis
_PRNUM	INT	Probe number

These following additional parameters are also valid:
_VMS, _TZL, _TSA, _FA and **_NMSP**.
See Sections 2.2 and 2.3.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Calibrating the tool probe

The tool probe is stationary but provides a switching signal. The calibration tool is positioned with the turret.

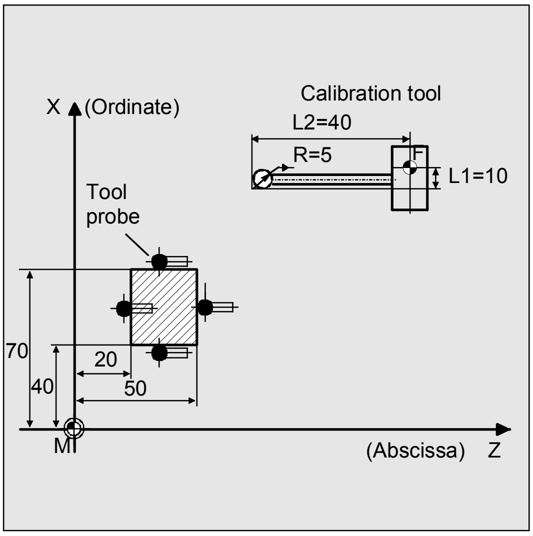
Values of the calibration tool in T7 D1 in this example:

Type	500
Tool edge position	3
L1	10
L2	40
R	5

Values of tool probe 1 in GUD6 block, which were determined manually to 5 mm accuracy beforehand (relative to the machine zero):

_TP[0,0] = 50
_TP[0,1] = 20
_TP[0,2] = 70
_TP[0,3] = 40

To obtain a minimum path of 1 mm, the measuring path is programmed as 1 + 5 = 6 (max. measuring path = 12 mm).



CALIBRATE_TOOL_PROBE	
N05 G0 SUPA G94 Z300 DIAMOF	Approach any change position
N10 SUPA X240 T7 D1	Calibration tool
N20 M71	Swing in tool probe (M function is machine-specific)
N25 _MVAR=0 _MA=2 _TZL=0.004 _TSA=2 _PRNUM=1 _VMS=0 _NMSP=1 _FA=6	Parameters for calibration cycle
N30 CYCLE982	Calibration in minus X direction
N35 G0 SUPA Z60	Approach new start position
N38 _MA=1	Select different measuring axis
N40 CYCLE982	Calibration in minus Z direction
N45 G0 SUPA X30	Approach new start position
N48 _MA=2	
N50 CYCLE982	Calibration in plus X direction
N55 G0 SUPA Z0	Approach new start position
N58 _MA=1	
N60 CYCLE982	Calibration in plus Z direction
N65 G0 SUPA X240	Approach any change position axis by axis
N70 SUPA Z300	
N99 M2	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.3.2 CYCLE982 Measure tool

Function

The cycle calculates the new tool length and checks whether the corrected difference from the old tool length is within a defined tolerance range (upper limits: safe area _TSA and dimension difference check _TDIF, lower limit: zero offset area _TZL). If this range is not violated, the new tool length is accepted, otherwise an alarm is output. Violation of the lower limit is not corrected.

Empirical values are included if selected (with the value of _EVNUM).

The lengths of turning tools (type 5xy) or milling tools/drills (type 1xy / 2xy) can be measured on lathes. In the case of milling tools, the tool radius offset can also be measured.

With milling tools, the measurement is further specified with the 3rd to 5th decimal places of parameter _MVAR.

The calculated offsets are entered in the active D number.

Whether the offset is entered in the geometry data thus deleting the wear data (first measurement) or whether the entry is made in the wear data (remeasurement), depends on the position of measurement bit _CHBIT[3].

The offset values in the measurement axis (_MA=) can be calculated.

If _CHBIT[20]=1, positioning of the milling spindle at the value of _STA1 can be suppressed.

That is possible with the following miller measurement variants:

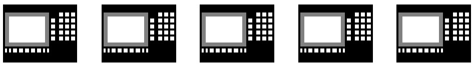
_MVAR=xxx001 (with x: 0 or 1, no other values)

General preconditions

The tool probe must be calibrated completely.

The tool to be measured must be called with tool length offset (D number).

The tool geometry data in tool offset have been entered (tool type, tool edge position, tool nose radius/cutter radius, length 1, length 2).



840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

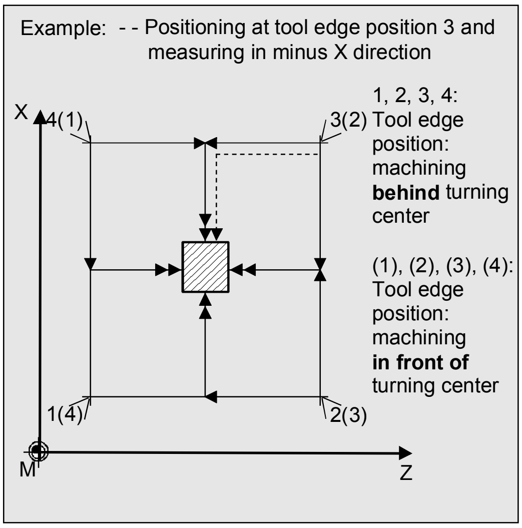
For mills/drills, setting data SD 42950: TOOL_LENGTH_TYPE =2 must be set (length calculation as for turning tool). For milling tools, the tool spindle must be declared the master spindle.

Parameters

_MVAR	1 or...01	Measurement variant: Tool measurement More precise specification for milling tools via the 3rd to 5th decimal places
_MA	1, 2	Measuring axis
_STA1		For milling tools: Initial angle
_CORA		For milling tools: Offset angle position after reversal

These following additional parameters are also valid: **_VMS, _TZL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM**, and **_NMSP**.

Procedure



Position before the cycle is called

Before the cycle is called, the tool must be moved to the starting position, as is shown in the diagram for turning tools. The measuring cycle then calculates the approach position itself. This position determines the direction of measurement in the measuring axis. For milling tools, the measuring point on the tool is determined by entered lengths 1 and 2 (please note: SD 42950). If the radius value is not equal to zero, this is also a determining factor. The measuring point is then located on the side which the measuring probe faces (+R or –R). The axial or radial position of the tool must be specified (_MVAR). The starting position must ensure collision-free approach.

Note for turning machines with a Y axis:
Before CYCLE982 is called, the Y axis must be put in a position corresponding to the center of the measurement cube in this axis. The Y axis is not positioned in the cycle itself.

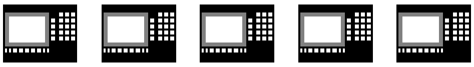
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

In the case of milling tools, length and radius can be selected as an alternative to length only to determine the cutter radius.

For length and radius, two measuring points are required. These are approached from different sides of the measuring probe. First the measuring point facing the measuring probe at the starting point is approached. Then, after travel round the probe (in the direction of the starting point), the 2nd measuring point is measured in the opposite direction. If the spindle is stationary (M5) and measurement without reversal is selected, the 2nd measurement is performed with a spindle rotation of 180 degrees. The same cutting edge used for the 1st measurement is now used.
The L1 or L2 offset values and the cutter radius are calculated from these two measurements.

Measurement with reversal can be selected separately with _MVAR:
First the measuring point is measured in the selected axis and in a milling spindle position according to starting angle _STA1. Then the tool (spindle) is turned 180 degrees and measured again. The average value is the measured value.
Measurement with reversal causes a second measurement at each measuring point P with a spindle rotation through 180 degrees from the starting angle. The offset angle entered in _CORA is summated to these 180 degrees. That enables selection of a certain 2nd milling cutting edge that is offset from the 1st cutting edge by precisely 180 degrees. Measurement with reversal permits measurement of two cutting edges of one tool. The mean value is the offset value.

If _CHBIT[20]=1, selected measurement variants are possible for a milling cutter without taking the starting angle _STA1 into account.

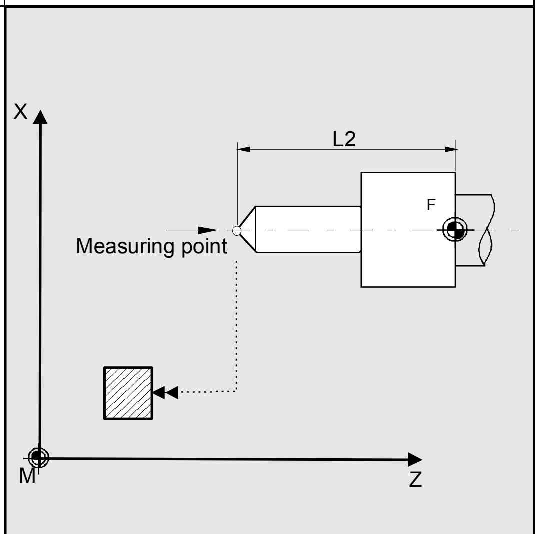


840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Information about measuring with a rotating spindle:
If selection of a certain miller cutting edge is not possible, it is possible to measure with a rotating spindle. The user must then program the direction of rotation, speed, and feedrate very carefully before calling up CYCLE982 to prevent damage to the probe. A low speed and feedrate must be selected. The direction of rotation must not be "cutting".

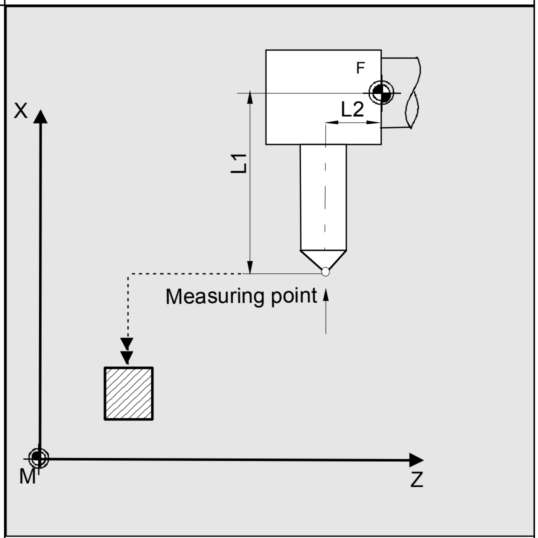
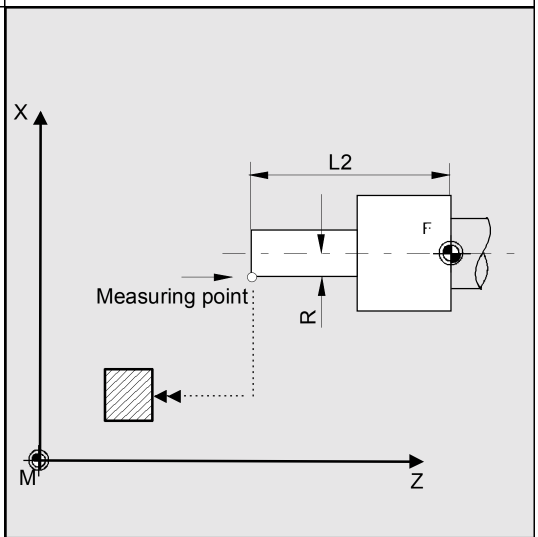
Position after the cycle has terminated

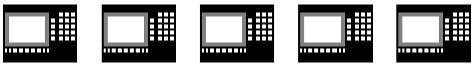
When the cycle is completed, the tool nose is positioned facing the measuring surface at a distance corresponding to _FA · 1 mm.

Measurement variant	Given geometry	Offset stored in	Milling tools, drills
Example: **Axial position,** R=0, Measuring without reversal, calculate length only **_MVAR=1** **_MA=1** Values L1 ≠ 0 are also possible	L1=0 L2=... R=0	L2	

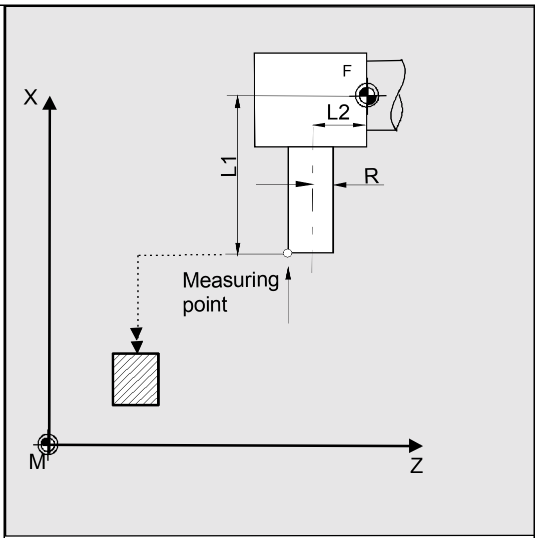
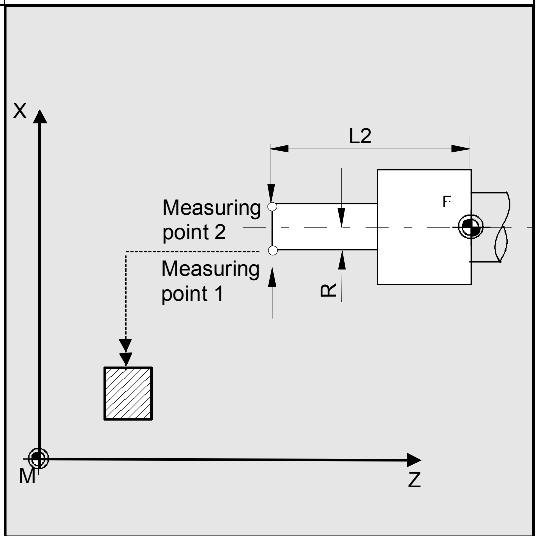
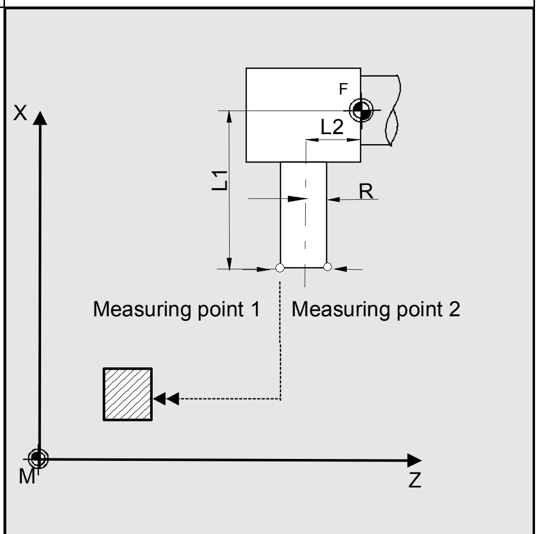
6.3 CYCLE982 Tool measurement (SW 5.3 and higher)

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Measurement variant	Given geometry	Offset stored in	Milling tools, drills
Example: **Radial position,** R=0, Measuring without reversal, calculate length only **_MVAR=10001** **_MA=2**	L1=... L2=... R=0	L1	
Example: **Axial position,** R ≠ 0, Measuring without reversal, calculate length only **_MVAR=1** **_MA=1**	L1=0 L2=... R=...	L2	

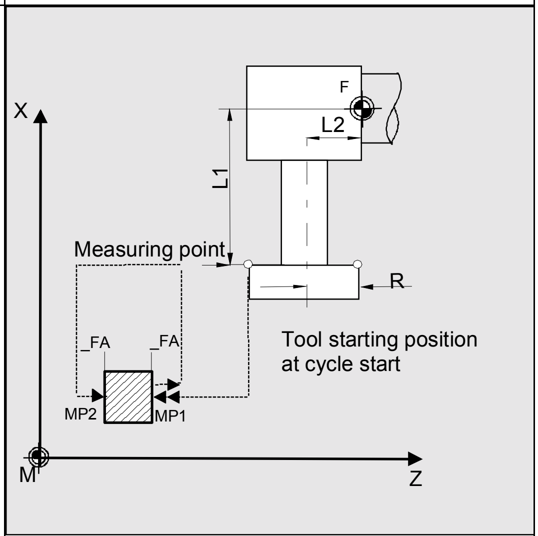


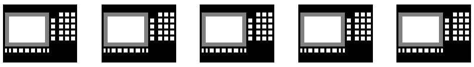
840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Example: **Radial position,** R ≠ 0, Measuring without reversal, calculate length only **_MVAR=10001** **_MA=2**	L1=... L2=... R=...	L1	
Example: **Axial position,** R ≠ 0, Measuring with reversal, calculate radius only **_MVAR=1101** **_MA=2** L1 must be known Values L1 ≠ 0 are also possible	L1=0 L2=... R=...	R R= ABS(MP – L1)	
Example: **Radial position,** R ≠ 0, Measuring with reversal, calculate length only **_MVAR=10101** **_MA=1** R must be known	L1=... L2=... R=...	L2 L2= (MP - R) or other measuring direction: L2= (MP + R)	

6.3 CYCLE982 Tool measurement (SW 5.3 and higher)

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Measurement variant	Given geometry	Offset stored in	Milling tools, drills
Example: **Radial position,** R ≠ 0, Measuring without reversal, calculate length and radius, 2 measuring points required **_MVAR=12001** **_MA=1** Notes: Before starting, the measuring point in both coordinates must be outside the measuring cube coordinates. On the opposite side of the measuring cube (MP2) measurement is performed with a rotated spindle (by 180 degrees). The same cutting edge is then measured. This only happens if the spindle is stationary and without reversal. In this example, L1 refers to the upper cutting edge. If L1 is to be calculated in another measurement, the starting position must be below the measuring cube.	L1=... L2=... R=...	L2 R L2= (MP1 + MP2)/2 R= ABS(MP1-MP2)/2	



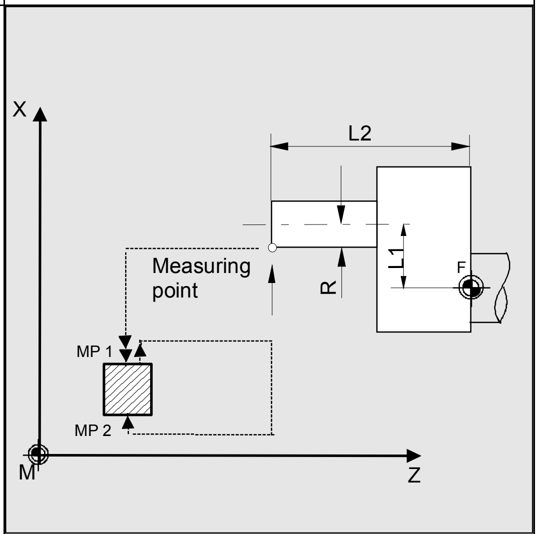
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

<table>
<tr><th>Measurement variant</th><th>Given geometry</th><th>Offset stored in</th><th>Milling tools, drills</th></tr>
<tr><td>Example:
Axial position,
R ≠ 0,
Measuring without reversal,
calculate length and radius,
2 measuring points required

_MVAR=2001
_MA=2

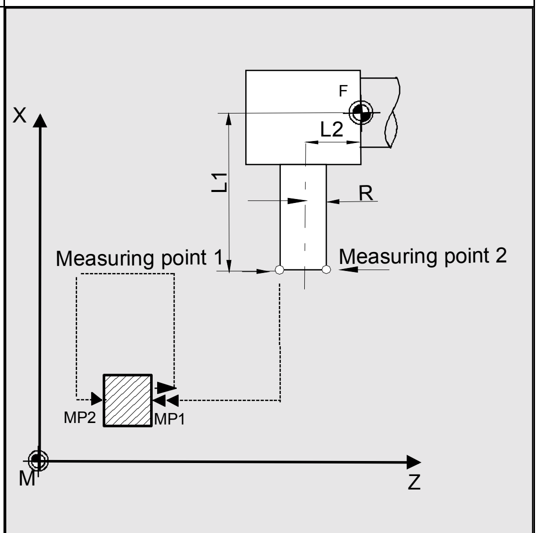
MP2 is measured with a rotated spindle (by 180 degrees) - if measurement is performed with a stationary spindle.</td><td>L1=..
L2=...
R=...</td><td>L1
R

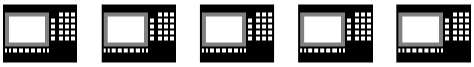
L1=
(MP1 + MP2)/2

R=
ABS(MP1-MP2)/2</td><td>
</td></tr>
<tr><td>Example:
Radial position,
R ≠ 0,
Measurement with reversal at each measuring point,
calculate length and radius,
2 measuring points necessary (4 measurements)

_MVAR=12101
_MA=1</td><td>L1=...
L2=...
R=...</td><td>L2
R

L2=
(MP1 + MP2)/2

R=
ABS(MP1-MP2)/2</td><td>
</td></tr>
</table>



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

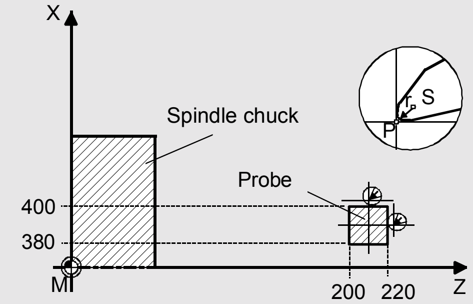
Programming example

Calibrating the tool probe with subsequent measurement of turning tool T3

The values of tool probe 1 must be preset in module GUD6 with a tolerance of approx. 1 mm, e.g.:

_TP[0,0] = 220
_TP[0,1] = 200
_TP[0,2] = 400
_TP[0,3] = 380

After calibration, the measured value (calibration value) is set.
All 4 points must be calibrated.



P - Tool nose
S - Center of cutting edge
r - Cutting edge radius

The dimensions of the calibration tool T7 are in lengths L1, L2 and the radius (R=5 mm) are known precisely and entered in offset field D1.
The tool edge position is 3.

The lengths and radius of tool T3 to be measured are known and entered in offset field D1. The cutting edge position is 3. The task is to measure the precise wear in both axes (adding measured value difference in the wear).

MEASURE_T3	
; Calibration:	
N1 G0 G18 DIAMOF	
N2 T7 D1	Call calibration tool
N3 SUPA Z250 X575	Start position for calibration
N5 _MVAR=0 _MA=2 _TZL=0.004 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	Parameter definition
N6 CYCLE982	Calibration in minus X direction
N7 G0 SUPA Z240	New start position
N8 _MA=1	Set other measuring axis (Z)
N9 CYLCE982	Calibration in minus Z direction
N10 G0 SUPA X360	New start position
N11_MA=2	Set other measuring axis (X)
N12 CYCLE982	Calibration in plus X direction
N13 G0 SUPA Z180	New start position
N14 _MA=1	Set other measuring axis (Z)
N15 CYLCE982	Calibration in plus Z direction
N16 G0 SUPA X575	Traverse each axis to the tool change
N17 SUPA Z520	position

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

; Measurement:	
N100 T3 D1	Selection of the tool to be measured
N110 G0 SUPA Z250 X575	Start position for measurement
N120 _MA=2 _TDIF=0.8 _MAVR=1	Change of parameter definition for measurement, otherwise calibration
N130 _CHBIT[3]=1	Offset in wear
N140 CYCLE982	Tool measurement in minus X direction (L1)
N150 G0 SUPA Z240	New start position
N160 _MA=1	Set other measuring axis (Z)
N170 CYCLE982	Tool measurement in minus Z direction (L2)
N180 G0 SUPA X575	Retraction axis by axis
N190 SUPA Z520	
N200 M2	

Explanation

Calibrate N1 to N17:
The "tip" of the calibration tool T7 is positioned in measuring axis X from the starting position at distance _FA=1 mm (dimension → with reference to the radius) before the probe. In axis Z, the probe tip center is centered with respect to the probe.
The measuring process is initiated in the negative X direction (_MA=2, starting position) with measuring velocity 150 mm/min (_VMS=0, _FA=1). The switching signal is expected by the probe 1 (_PRNUM=1) within a distance of 2 · _FA=2 mm. Otherwise, an alarm will be triggered.
Measurement is performed once (_NMSP=1).
After successful measurement, the "tip" of T7 is _FA=1 mm in front of the probe in the X direction.
The calculated probe value is entered in _TP[0,2].
Calibration with the measuring process has been completed in minus X.
Calibration is then performed in the other measuring directions/axes.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Explanation

Measure N100 to N200:
The probe has been calibrated completely.
The "nose" of the turning tool T3 is positioned in measuring axis X from the starting position at distance _FA=1 mm (dimension → with reference to the radius) in front of the probe. In axis Z the center of the cutting edge is centered with respect to the probe. If the cutting edge radius =0, it is the tool nose.
The measuring process is initiated in the negative X direction (_MA=2, starting position) with measuring velocity 150 mm/min (_VMS=0, _FA=1). The switching signal is expected by the probe 1 (_PRNUM=1) within a distance of 2 · _FA=2 mm. Otherwise, an alarm will be triggered. Measurement is performed once (_NMSP=1).
After successful measurement, the "nose" of T3 is _FA=1 mm in front of the probe in the X direction.
The calculated length difference of L1 (tool type 5xy, _MA=2, _MVAR=xx**0**xx**1**) is summated and entered in D1 from T3 in the wear (_CHBIT[3]=1).
Measurement and wear offset are then performed in L2 in the minus Z direction.

Recommended parameters

The following parameters are suggested so that this programming example runs reliably:

- Calibration:

_TZL = 0.001	zero offset range
_TSA = 1	safe area

- Measurement:

_TZL = 0.001	zero offset range
_TSA = 1	safe area during continuous operation
_TSA = 3	safe area during setup
_TDIF = 0.3	dimensional difference check during continuous operation
_TDIF = 3	dimensional difference check during setup

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

6.3.3 CYCLE982 Automatic tool measurement

Function

Function as for – non-automatic measurement

Relevant information:
In the case of turning tools, both lengths are calculated. (for tool edge positions 5, 6, 7 and 8, only one length. With milling tools/drills, the measurement is further specified with the 3rd to 5th decimal places of parameter _MVAR.
The measuring cycle generates the approach blocks to the measuring probe and the traversing movement for measurement from length 1, length 2, and the radius itself. Prerequisite is a correctly selected starting position.

In automatic measurement, the offsets to be calculated are defined by the tool type.

- Turning tool: Both lengths (2 measurements)
- Drill: Length according to axial or radial position (1 measurement)
- Mill: Both lengths and radius (4 measurements), if the radius is R=0, both lengths only are calculated (2 measurements).

The calculated offsets are entered in the active D number. Whether the offset is entered in the geometry data thus deleting the wear data (first measurement) or whether the entry is made in the wear data (remeasurement), depends on the position of measurement bit _CHBIT[3].

Precondition

Non-automatic – as for tool measurement

6.3 CYCLE982 Tool measurement (SW 5.3 and higher)

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

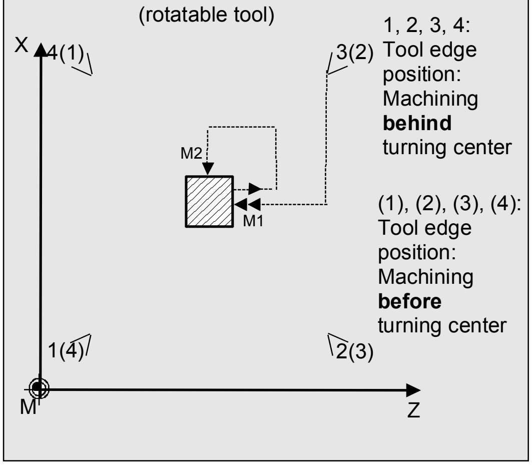
Parameters

_MVAR	2 or ...02	Measurement variant: Automatic tool measurement More precise specification for milling tools/drills via the 3rd to 5th decimal places
_STA1		For milling tools: Initial angle
_CORA		For milling tools: Offset angle position after reversal

These following additional parameters are also valid:
_VMS, _TZL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, and **_NMSP**.

Procedure

Example: Approach with tool edge position 3 and automatic measurement (rotatable tool)



Position before the cycle is called

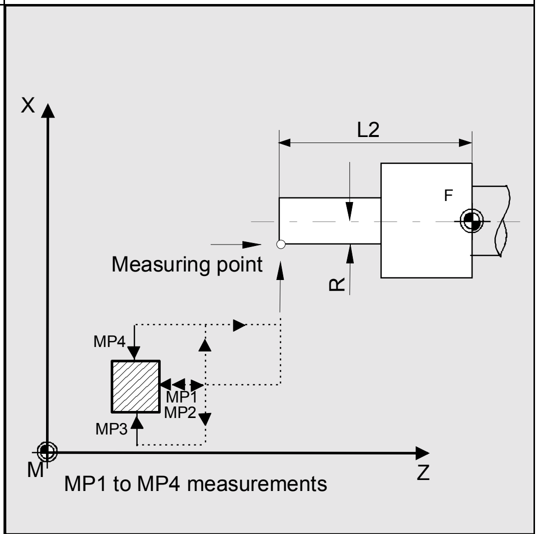
Before the cycle is called, the tool must be moved to the starting position, as is shown in the diagram for turning tools. The measuring cycle then calculates the approach position itself.
First the length in the abscissa (Z axis for G18) and then in the ordinate (X axis for G18) is measured. For turning tools, the measuring probe travels round the measuring cube at distance _FA.

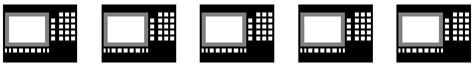
For milling tools, the measuring points on the tool are determined by entered lengths 1 and 2 (please note: SD 42950). If the radius value is not equal to zero, this is also a determining factor. The 1st measuring point is located on the side which the measuring probe faces (+R or –R). The axial or radial position of the tool must be specified in _MVAR, and the starting position approached accordingly. First, the values in the abscissa (Z axis for G18) are measured.
Measurement with reversal can be selected separately with _MVAR.
The probe travels round the measuring cube at distance _FA · 1 mm or starting point coordinate/ measuring cube (see figs.).

Position after the cycle has terminated

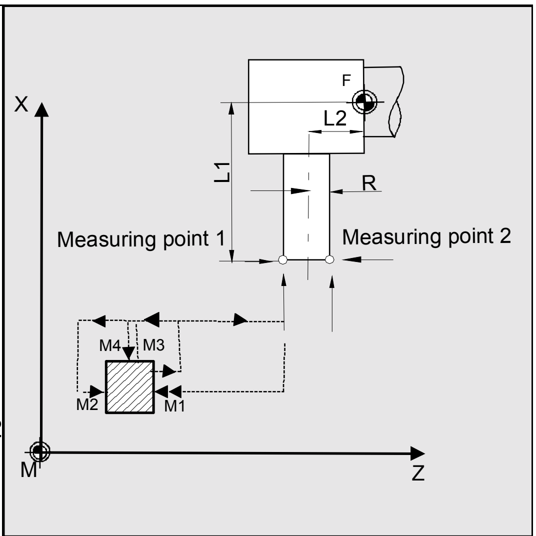
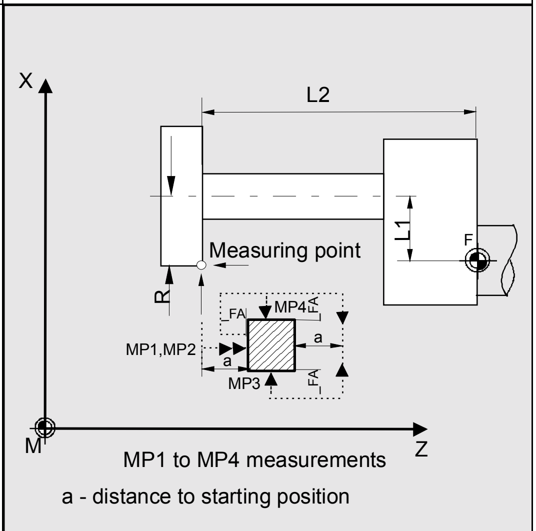
When the cycle is complete, the tool nose is again located at the starting point. A movement to this point is automatically generated in the cycle.

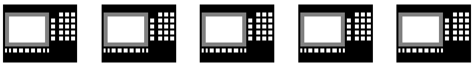
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Measurement variant	Given geometry	Offset in	Milling tools
Example: **Axial position,** R ≠ 0, Measuring without reversal, spindle stationary, 4 measurements necessary **_MVAR=2** Values for L1 ≠ 0 are also possible Procedure MP1 is approached with the staring angle position _STA1 of the milling spindle and measured. As the spindle is stationary (M5) and reversal measurement is not selected, the spindle is rotated by 180 degrees and the same cutting edge is measured again after it has been positioned in the center of the measuring cube. The mean value of both measurements is L2. Then MP3 is approached and measured, after that the spindle is again rotated by 180 degrees and MP4 is measured. L1 and R are calculated from these two measurements. The probe is then retracted to the starting point in axis sequence abscissa/ ordinate.	L1=0 L2=... R=...	L1 L2 R L1= (MP3x + MP4x)/2 L2= (MP1z + MP2z)/2 R= ABS(MP3x-MP4x)/2	

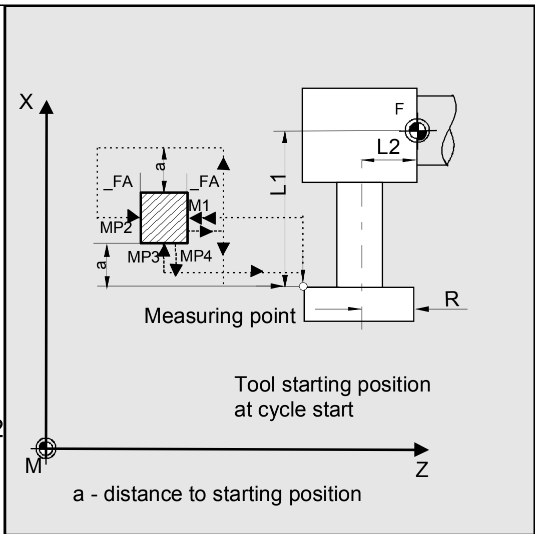


840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

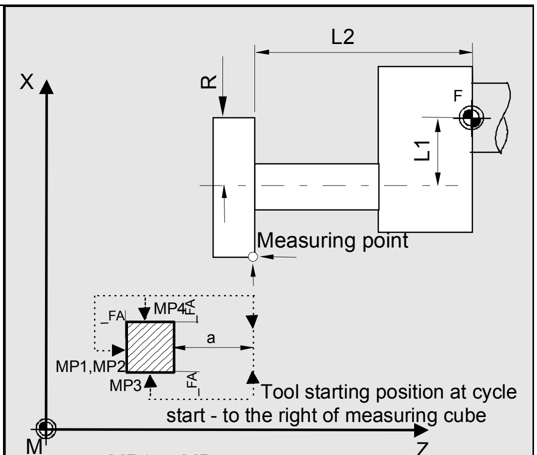
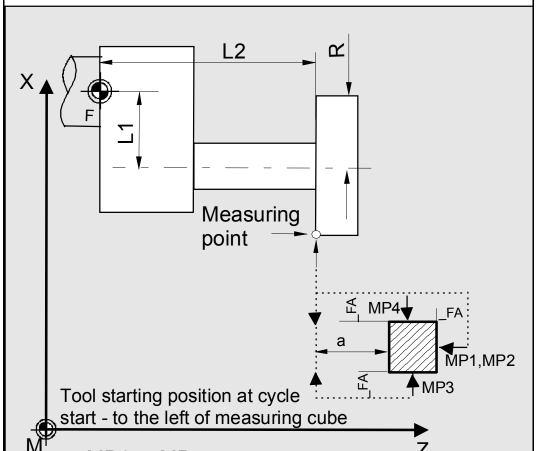
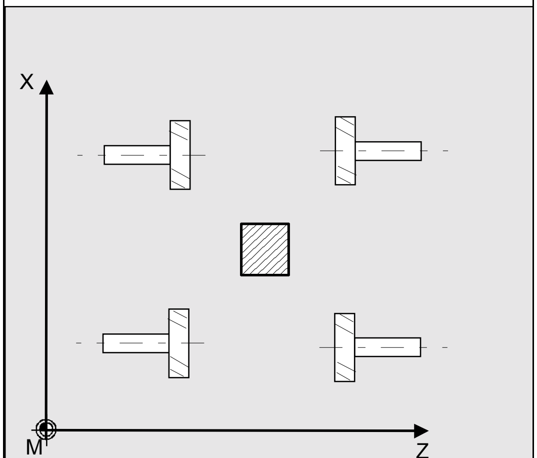
Example: **Radial position,** R ≠ 0 Measuring with reversal, 8 measurements necessary (MP1 to MP4 each with reversal) **_MVAR=10102**	L1=... L2=... R=...	L1 L2 R L1= (MP3x + MP4x)/2 L2= (MP1z + MP2z)/2 R= ABS(MP1z-MP2z)/2	
Example: **Axial position,** R ≠ 0, Measuring without reversal, 4 measurements necessary **_MVAR=3002** The probe travels round the measuring cube opposite the starting position side. Note: Length measurements for L2 (MP1,MP2) are performed here at the same measuring point 1 – without rotating the spindle by 180 degrees. The same cutting edge is always measured (starting angle _STA1).	L1=... L2=... R=...	L1 L2 R L1= (MP3x + MP4x)/2 L2= (MP1z + MP2z)/2 R= ABS(MP3x-MP4x)/2	

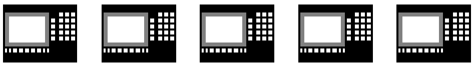


840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

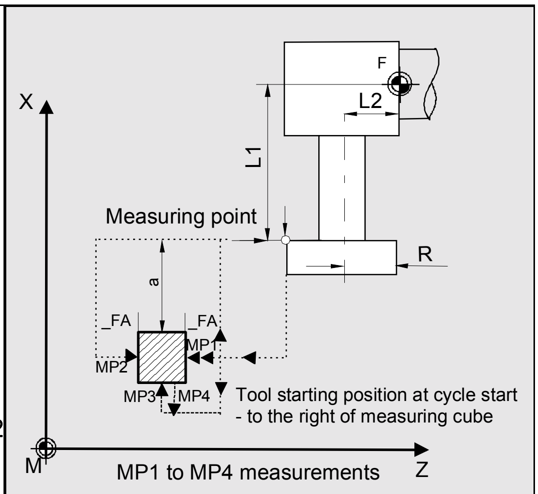
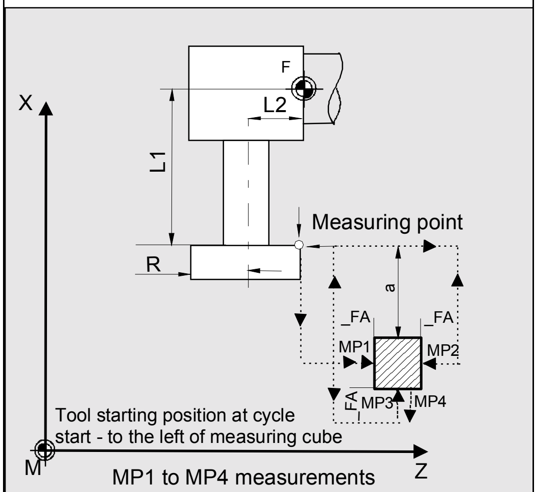
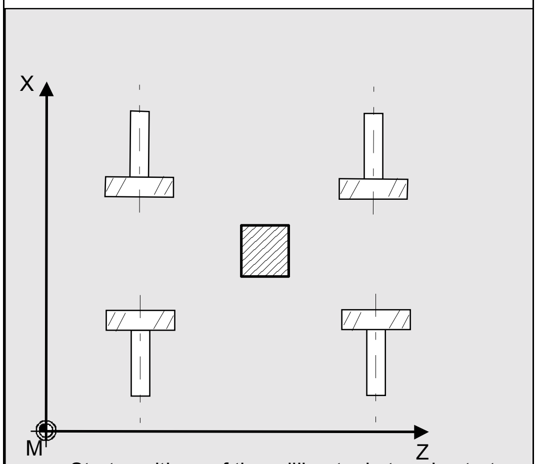
Example: **Radial position,** R ≠ 0 Measuring without reversal, 4 measurements necessary **_MVAR=13002** The probe travels round the measuring cube opposite the starting position side. Note: Length measurements for L2 (MP3 MP4) are performed here at the same measuring point 1 – without rotating the spindle by 180 degrees. The same cutting edge is always measured (starting angle _STA1).	L1=... L2=... R=...	L1 L2 R L1= (MP3x + MP4x)/2 L2= (MP1z + MP2z)/2 R= ABS(MP1z-MP2z)/2	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Example: (as of SW 5.4) **Axial position,** R ≠ 0, Measuring without reversal, 4 measurements necessary **_MVAR=4002** Direction of measurement for determining length L2 opposite to traversing direction, measuring procedure as for _MVAR=x3002 but with different traversing motion Notes: Length measurements for L2 (MP1,MP2) are performed here at the same measuring point – without rotating the spindle by 180 degrees. The same cutting edge is always measured (starting angle _STA1). The width of the milling tool must be considered when selecting the start position or dimension a!	L1=... L2=... R=...	L1 L2 R L1= (MP3x + MP4x)/2 L2= (MP1z + MP2z)/2 R= ABS(MP3x-MP4x)/2	 MP1 to MP4 measurements a - distance to starting position  MP1 to MP4 measurements a - distance to starting position  Start positions of the milling tool at cycle start and various radial positions



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Example: (as of SW 5.4) **Radial position,** R ≠ 0, Measuring without reversal, 4 measurements necessary **_MVAR=14002** Direction of measurement for determining length L1 opposite to traversing direction, measuring procedure as for _MVAR=13002 but with different traversing motion Notes: Length measurements for L2 (MP3 MP4) are performed here at the same measuring point – without rotating the spindle by 180 degrees. The same cutting edge is always measured (starting angle _STA1). The width of the milling tool must be considered when selecting the start position or dimension a!	L1=... L2=... R=...	L1 L2 R L1= (MP3x + MP4x)/2 L2= (MP1z + MP2z)/2 R= ABS(MP3z-MP4z)/2	 MP1 to MP4 measurements a - distance to starting position  MP1 to MP4 measurements a - distance to starting position  Start positions of the milling tool at cycle start and various radial positions

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.3.4 Incremental calibration (SW 6.2 and higher)

Function

The cycle uses the calibration tool to ascertain the current distance dimensions between the machine zero and the probe trigger point and automatically loads them into the appropriate data area in the GUD6 module. They are always calculated without empirical or mean values.

Precondition

The lateral surfaces of the probe cube must be aligned parallel to the machining axes abscissa and ordinate.

The coordinates of the tool probe regarding the machine zero are not known before starting calibration (data field _TP[_PRNUM-1,0] to _TP[_PRNUM1,3] contains invalid values).

Length 1 and 2
and the radius of the calibration tool must be known exactly and stored in a tool offset data block.

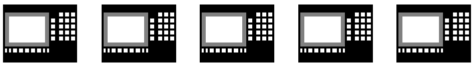
This tool offset must be active when the cycle is called. A turning tool must be specified as the tool type (type 5xy). The tool edge position must be 3.

The calibration tool (tool tip) must, before CYCLE982 is started, have a position that causes the probe to switch in the specified direction _MD for the measuring axis _MA within path 2 · _FA [mm].
Careful when positioning manually!
Damage to the probe must be avoided.

Parameters

_MVAR	100000	(6 decimal places)	Calibrate tool probe incrementally
_MA	1, 2		Measuring axis
_MD	0, 1		Measurement direction 0 - positive, 1 - negative
_PRNUM	INT		Probe number

These following additional parameters are also valid:
_VMS, **_FA** and **_NMSP**.
See Sections 2.2 and 2.3.



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Procedure

Position before the cycle is called

The calibration tool must be prepositioned as shown in the figure.
The "tip" of the calibration tool in the **measuring axis** _MA is within distance
2 · _FA [mm] **in front of** the measuring surface (dimension always with reference to the radius - like DIAMOF).
The center of the calibration tool tip in the **other axis** (offset axis) must be in the center of the probe.
The measuring cycle starts measuring in the specified axis (_MA) and direction (_MD) immediately on starting.

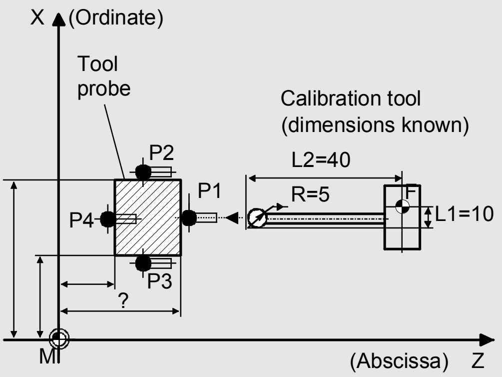
Position after the cycle has terminated

When the calibration procedure is completed the calibration tool is positioned on the starting position again.

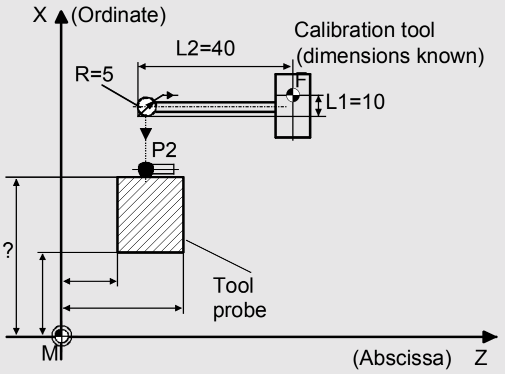
Notes

A special tool is used as the calibration tool and is entered as a turning tool (5xy) with cutting edge 3. It is usually shaped (bent) such that it is also possible to approach point P4 for calibration (_MA=1, _MD=0). However, it is not necessary to calibrate all 4 points for **incremental** measurement. The points that are used for incremental measurement are sufficient. That does not apply to automatic measurement. In that case, all 4 points must be calibrated so that the tool to be gauged can be centered automatically.

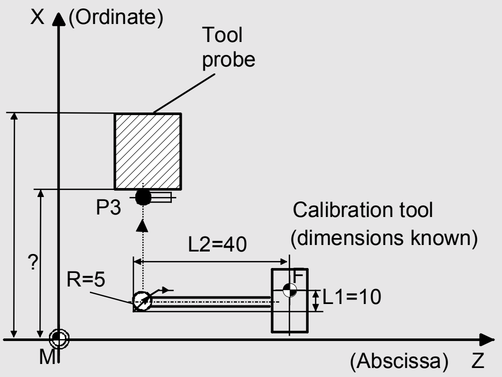
P1 to P4: possible points for calibration



Calibrate probe using calibration tool
P1: negative Z direction (_MA=1, _MD=1)



Calibrate probe using calibration tool
P2: negative X direction (_MA=2, _MD=1)



Calibrate probe using calibration tool
P3: positive X direction (_MA=2, _MD=0)

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Calibrate tool probe incrementally

The tool probe is stationary but provides a switching signal. The calibration tool is positioned with the turret.
Values of the calibration tool in T7 D1 in this example:

Type	500
Tool edge position	3
L1	10
L2	40
R	5

Values of tool probe 1 in module GUD6 before calibration:

_TP[0,0] = ?
_TP[0,1] = ?
_TP[0,2] = ?
_TP[0,3] = ?

INCR_CALIBRATION	
N10 T7 D1 G94	Calibration tool is active, starting position reached
N20 _MVAR=100000 _MA=2 _MD=1 _FA=20 _PRNUM=1 _VMS=0 _NMSP=1	Parameters for calibration cycle
N30 CYCLE982	Calibration in minus X direction
N99 M2	

Explanation

Before the program is started, the "tip" of the calibration tool T7 is in measuring axis X in a range 2 · _FA=40 mm (dimension with reference to radius) in front of the probe. In axis Z, the probe tip center is centered with respect to the probe.
When CYCLE982 is started, measurement starts in the negative X direction (_MA=2, MD=1) with measuring velocity 300 mm/min (_VMS=0, _FA>1). The switching signal is expected by the probe 1 (_PRNUM=1) within a distance of 2 · _FA=40 mm. Otherwise, an alarm will be triggered. Measurement is performed once (_NMSP=1).
After successful measurement, the "tip" of T7 is in the starting position again.
The calculated probe value is entered in _TP[0,2].
Calibration with the measuring process has been completed in minus X.

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

6.3.5 Incremental measurement (SW 6.2 and higher)

Function

The lengths of turning tools (type 5xy) or milling tools/drills (type 1xy / 2xy) can be measured on lathes.
In the case of milling tools, the miller radius offset can also be measured.
With milling tools, the measurement is further specified with the 3rd to 5th decimal places of parameter _MVAR.
The calculated offsets are entered in the active D number. The offset is entered in the geometry data and the wear data are reset.
Only the offset value that is in the measuring axis _MA can be determined in a measurement.

If _CHBIT[20]=1, positioning of the milling spindle at the value of _STA1 can be suppressed.
This is possible with the following miller measurement variants:
_MVAR= xxx001
(with x : 0 or 1, no other values).

General preconditions
For incremental measurement, the tool probe must be calibrated in the measuring axis and direction in which measuring will be performed.
The tool T to be measured must be called with tool length offset (D number).
The tool type is entered in the offset data.

For mills/drills, setting data SD 42950: TOOL_LENGTH_TYPE =2 must be set (length calculation as for turning tool). For milling tools, the tool spindle must be declared the master spindle.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Parameters

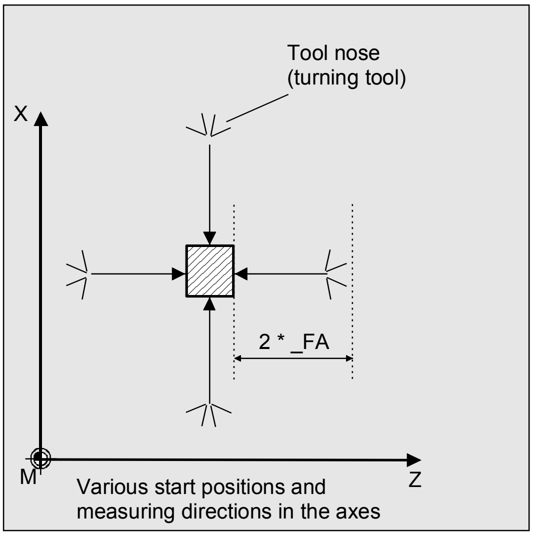
_MVAR	1xxxx1 (6 decimal places)	Measuring a tool incrementally More precise specification for milling tools via the 3rd to 5th decimal places
_MA	1, 2	Measuring axis
_MD	0, 1	Measurement direction 0 - positive, 1 - negative
_STA1		For milling tools only: Starting angle of the milling spindle
_CORA		Only for milling tools and measurement with reversal: Offset angle position of the milling spindle after reversal

These following additional parameters are also valid:
_VMS, _FA, _PRNUM and **_NMSP.**
See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called

Before the cycle is called, a starting position must be approached - as shown in the figure for **turning tools**, e.g.: by traversing in JOG mode.
The "tip" of the tool in the **measuring axis** _MA is within the distance
2 · _FA [mm] **in front of** the measuring surface (dimension always with reference to the radius - like DIAMOF).
The center of the cutting edge radius on the turning tool in the **other axis** is in the center of the probe. If the cutting edge radius =0, it is the tool nose.
For **milling tools**, the axial or radial position of the tool must be specified in _MVAR;
as must **measurement with reversal:**
First the measuring point is measured in the selected axis and in a milling spindle position according to starting angle _STA1. The tool (milling spindle) is then rotated through 180 degrees plus the value in _CORA and measured again. The average value is the measured value.
If the milling spindle is activated when the cycle is started, measurement will be performed with a **rotating spindle**. In that case, the user must exercise special care when selecting the speed, direction of rotation, and feedrate!

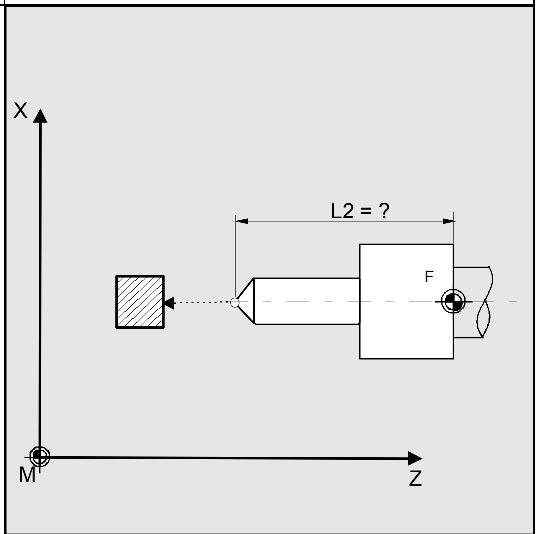
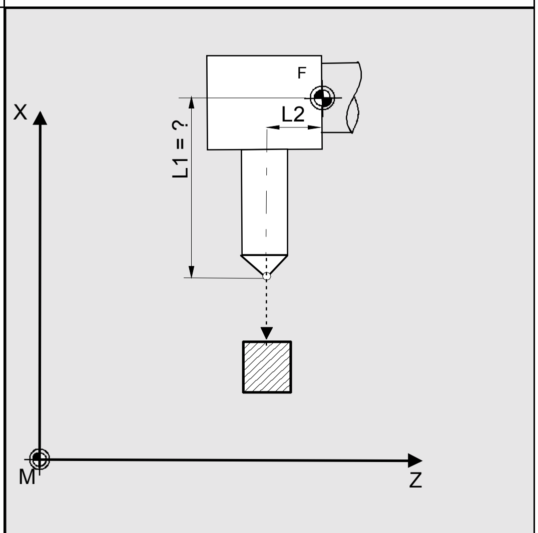


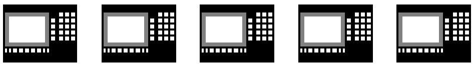
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

If _CHBIT[20]=1, selected measurement variants are possible for a milling cutter without taking the starting angle _STA1 into account.

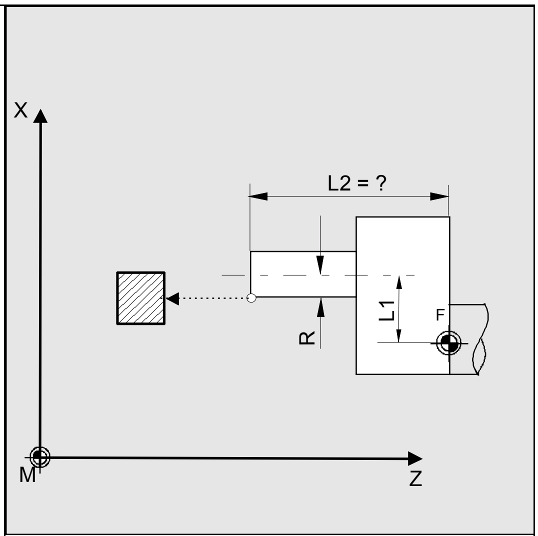
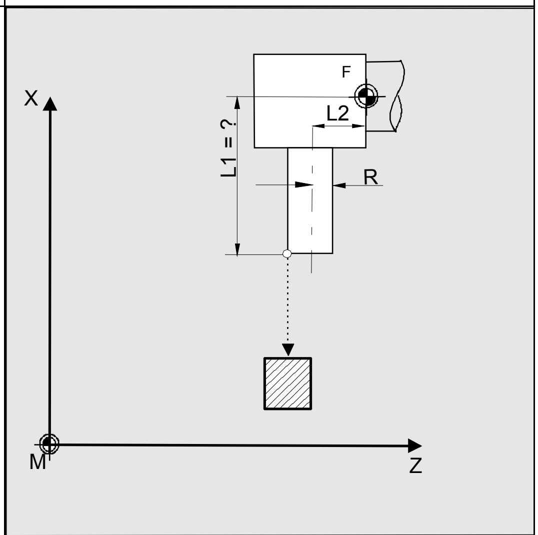
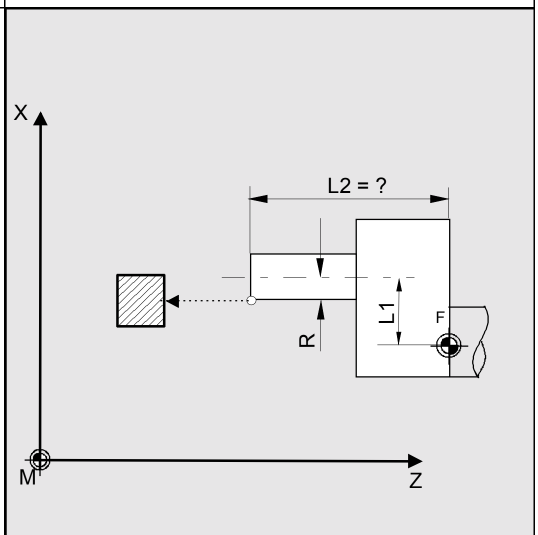
Position after the cycle has terminated

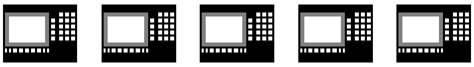
When the cycle is completed, the tool tip is back in the starting position.

Measurement variant	Specified geometry	Offset in	Milling tools, drills
Example: **Axial position,** drill, R=0, incremental measurement without reversal, calculation of the length in Z **_MVAR=100001** **_MA=1** Values L1 ≠ 0 are also possible. But always position the drill tip in the center of the probe!	L1=0 L2=... R=0	L2	
Example: **Radial position,** drill, R=0, measuring without reversal, calculation of the length in X **_MVAR=110001** **_MA=2**	L1=... L2=... R=0	L1=...	

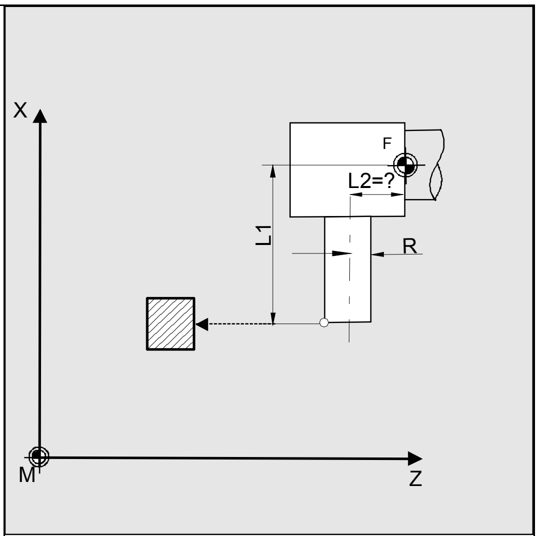
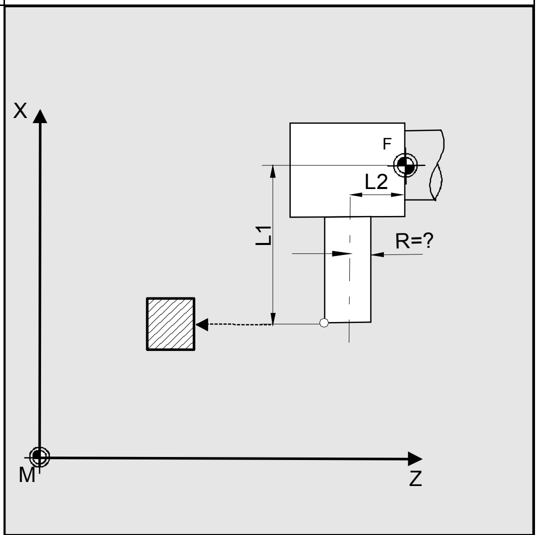


840 D	840 D	FM-NC	810 D	840 Di
NCU 571	NCU 572 NCU 573			

Example: **Axial position,** miller, R ≠ 0, measuring without reversal, calculation of the length in Z **_MVAR=100001** **_MA=1**	L1= -... L2=... R=...	L2	
Example: **Radial position,** miller, R ≠ 0, measuring without reversal, calculation of the length in X **_MVAR=110001** **_MA=2**	L1=... L2=... R=...	L1	
Example: **Axial position,** miller, R ≠ 0, measuring with reversal, calculate radius **_MVAR=101101** **_MA=2** In this case, L1 must be known	L1=.... L2=... R=...	R	

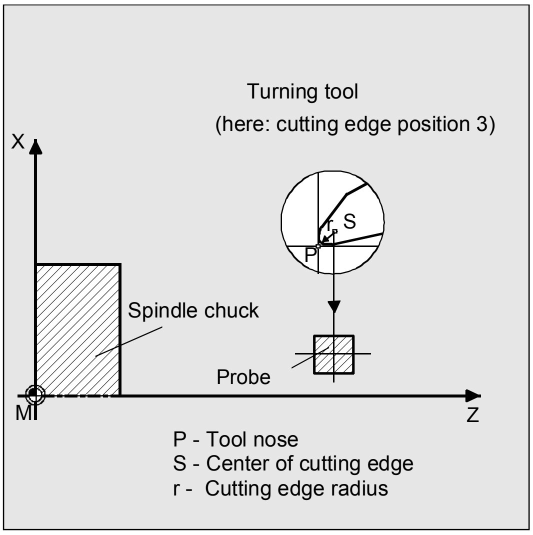


840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Example: **Radial position,** miller, R ≠ 0, measuring with reversal, calculation of the length in Z **_MVAR=110101** **_MA=1** In this case, R must be known	L1=... L2=... R=...	L2	
Example: **Radial position,** miller, R ≠ 0, measuring with reversal, calculate radius **_MVAR=111101** **_MA=1** In this case, L2 must be known	L1=... L2=... R=...	R	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example



INCR_MEASUREMENT	
N10 T3 D1 G94	Turning tool T3 is active, starting position reached
N20 _MVAR=100001 _MA=2 _FA=20 _MD=1 _PRNUM=1 _VMS=0 _NMSP=1	Parameters for measuring cycle
N30 CYCLE982	Measurement in minus X direction
N99 M2	

Explanation

The probe has been calibrated in minus X.
Before the program is started, the "tip" of the tool T3 is in measuring axis X in a range 2 · _FA=40 mm (dimension with reference to radius) in front of the probe. In axis Z, the center of the cutting edge is centered with respect to the probe. If the cutting edge radius =0, it is the tool nose.

When CYCLE982 is started, measurement starts in the negative X direction (_MA=2, MD=1) with measuring velocity 300 mm/min (_VMS=0, _FA>1). The switching signal is expected by the probe 1 (_PRNUM=1) within a distance of 2 · _FA = 40 mm. Otherwise, an alarm will be triggered. Measurement is performed once (_NMSP=1).
After successful measurement, the "tip" of T3 is in the starting position again.
The calculated length L1 (tool type 5xy, _MA=2, _MVAR=xx**0**xxx) is entered in D1 of T3 in the geometry. The associated wear component is reset.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.3.6 Milling tool: suppression of starting angle positioning with _STA1 (SW 6.2 and higher)

Function

To accept the angular position of the milling spindle (cutting edge of the miller contacting the probe) unchanged into the cycle and thus suppress the starting angle positioning with the value in _STA1, you can set

_CHBIT[20]=1

However, this only permits the simple miller measurement variants that do not have to access the starting angle in _STA1, e.g.: no 2nd measurement or repositioning after measurement with reversal. Otherwise, those miller measurement variants are possible that are also permitted during incremental measurement.

If the machine does not feature an SPOS-capable milling spindle, it is also possible to measure millers with these measurement variants and _CHBIT[20]=1.

Permissible measurement variants with miller and _CHBIT[20]=1:
xxx001 (with x : 0 or 1, no other values)

Other measurement variants with a miller will be rejected with an alarm message.

For measurement with a rotating spindle and _CHBIT[20]=1, these are also the only measurement variants permitted. Measurement with reversal is not permitted.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.4 CYCLE973 Calibrate workpiece probe

Programming

CYCLE973

Function

With this cycle the workpiece probe can be calibrated either in a reference groove or on a surface.

Result parameters

Measuring cycle CYCLE973 returns the following result values in the GUD5 module:

_OVR [4]	REAL	Actual value probe ball diameter
_OVR [5]	REAL	Difference probe ball diameter
_OVR [8]	REAL	Trigger point in minus direction, actual value, abscissa
_OVR [10]	REAL	Trigger point in plus direction, actual value, abscissa
_OVR [12]	REAL	Trigger point in minus direction, actual value, ordinate
_OVR [14]	REAL	Trigger point in plus direction, actual value, ordinate
_OVR [9]	REAL	Trigger point in minus direction, difference, abscissa
_OVR [11]	REAL	Trigger point in plus direction, difference, abscissa
_OVR [13]	REAL	Trigger point in minus direction, difference, ordinate
_OVR [15]	REAL	Trigger point in plus direction, difference, ordinate
_OVR [20]	REAL	Positional deviation abscissa
_OVR [21]	REAL	Positional deviation ordinate
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVI [2]	INTEGER	Measuring cycle number
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Measurement variants

The measuring cycle CYCLE973 permits the following calibration variant that are defined via the parameter _MVAR.

The possible values of the parameter are between 0 ... 12113 and are formed as follows:

Digit					*Meaning*
5	**4**	**3**	**2**	**1**	
0					No position calculation
1					With position calculation only for calibration in groove
	1				1 axis direction (indicate measuring axis and axis direction)
	2				2 axis directions only for calibration in slot (specifying measuring axis)
		0			No calculation of probe ball
		1			Calculation of probe ball (only for calibration in groove)
			0	0	Any surface
			1	3	Groove
			0		With any data in the plane
			1		With reference data in the plane

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

6.4.1 CYCLE973 Calibrate in the reference groove (plane)

Function

With this measuring cycle, it is possible to calibrate the probe in a reference groove. Calibration in the reference groove is possible in the abscissa and ordinate.

The calculated setpoint/actual value difference is offset against the probe length. The newly calculated trigger values are then loaded into the corresponding data area of the module GUD6.DEF.

Calibration is only performed on one surface (axis direction) in the groove.

Precondition
The probe must be called **with** tool offset. Only probes with "tool edge position" 7 or 8 can be used (see Subsection 1.5.2).
The valid reference groove is selected with _CALNUM.

Parameters

_MVAR	see Section 6.4	Definition of calibration variant
_MA	1, 2	Measuring axis
_MD	0 positive axis direction 1 negative axis direction	Measuring direction (depends on the measurement variant)
_CALNUM	INT	Number of calibration groove
_PRNUM	INT	Probe number

These following additional parameters are also valid:
_VMS, _TZL, _TSA, _FA and **_NMSP**.

See Sections 2.2 and 2.3.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

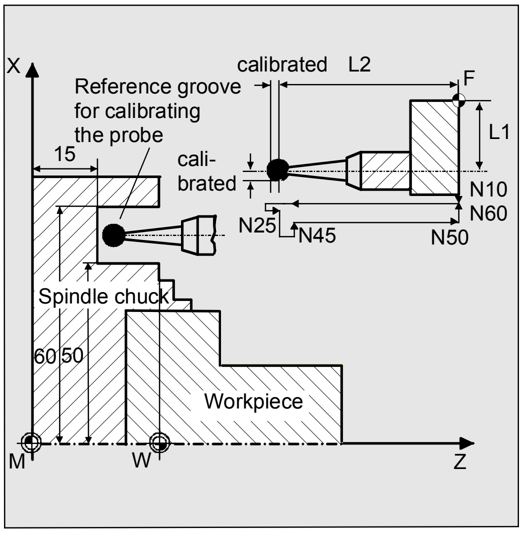
Procedure

Position before the cycle is called
A starting point must be selected from which the cycle can position the selected probe automatically to the relevant calibration groove, along the shortest path with paraxial movements.

Position after the cycle has terminated
On completion of the calibration process, the probe is positioned facing the calibration surface at a distance corresponding to _FA · 1 mm.

Programming example

Calibrate in the reference groove
The probe lengths L1 and L2 refer to the center point of the probe and must be entered in the tool offset memory (T8 D1 in the example), before the cycle is called.



CALIBRATE_IN_GROOVE	
N10 G0 SUPA G90 X95 Z125 T8 D1 DIAMOF	Position in front of the cycle call and select tool offset for the probe (tool type 500, SL 7)
N15 _MVAR=13 _MA=1 _MD=1 _CALNUM=1 _TZL=0 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=3	Set parameters for calibration in minus Z direction
N25 CYCLE973	Cycle call
N35 _MA=2	Set parameters for calibration in minus X direction
N45 CYCLE973	Cycle call
N50 G0 SUPA Z125	Retraction in Z
N60 SUPA X95	Retraction in X
N90 M30	

The new trigger values are stored in the corresponding global data of probe 1 _WP[0,1] and _WP[0,3].

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.4.2 CYCLE973 Calibrate on a random surface

Function

With this measuring cycle, you can calibrate the probe on a random surface, e.g. on the workpiece, to determine the trigger points.

Precondition

The probe is called up **with** tool offset and positioned opposite the calibrating surface. 500 should be entered as the tool type. Measuring cycle SW 6.2 and higher also allows you to enter tool type 580 (probe). Tool edge positions 5 to 8 are permitted.

For calibration in the plus direction below the turning center or to the left of the workpiece zero, the setpoint _SETVAL must be specified as a negative value.

Parameters

_MVAR	0	Calibration variant: Calibration on random surface
_SETVAL	REAL	Setpoint referred to the workpiece zero, for facing axis in the diameter
_MA	1, 2, 3[1)]	Measuring axis
_MD	0 positive axis direction 1 negative axis direction	Measurement direction
_PRNUM	INT	Probe number

1) As of measuring cycles SW 5.4, it is also possible to calibrate in the 3rd axis (Y in G18), provided that this axis exists.

These following additional parameters are also valid: **_VMS, _TZL, _TSA, _FA** and **_NMSP**.

See Sections 2.2 and 2.3.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

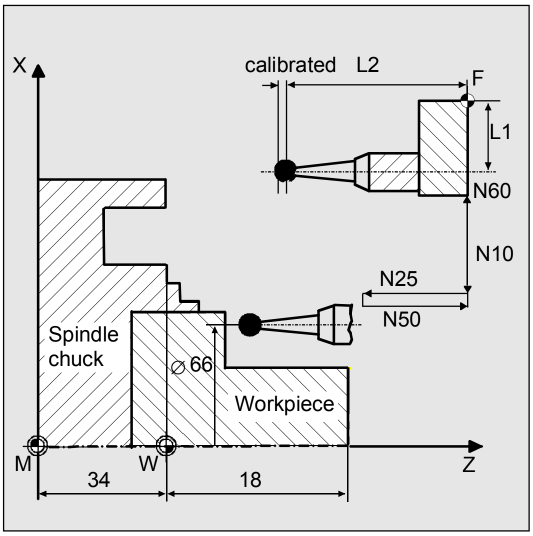
Procedure

Position before the cycle is called
The starting point is a random position opposite the calibration surface.

Position after the cycle has terminated
On completion of the calibration process, the probe is positioned facing the calibration surface at a distance corresponding to _FA · 1 mm.

Programming example

Calibrating a probe at a random surface in the minus Z direction
The probe lengths L1 and L2 refer to the center point of the probe and must be entered in the tool offset memory (T9 D1 in the example), before the cycle is called. The tool type is 500, the tool edge positions is 7.



CALIBRATE_IN_Z	
N10 G54 G0 X66 Z90 T9 D1 DIAMON	Position in front of the cycle call and select tool offset for the probe (tool type 500, SL 7)
N15 _MVAR=0 _SETVAL=18 _MA=1 _MD=1 _TZL=0 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=3	Set parameters for calibration in minus Z direction
N25 CYCLE973	Cycle call
N50 G0 Z90	Retraction in Z
N60 X146	Retraction in X
N90 M30	

The new trigger value in -Z is entered in the data of probe 1 _WP[0,1].

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.5 CYCLE974 Workpiece measurement

Programming

CYCLE974

Function

The measuring cycle ascertains the actual value of the workpiece in the selected measuring axis with reference to the workpiece zero and calculates the setpoint/actual value difference.
Both an empirical value stored in the GUD5 module and a mean value over several parts can be considered. The cycle checks that a set tolerance range for the measured deviation is not violated and automatically corrects the ZO memory or tool offset memory selected in _KNUM. Measurement is possible in all directions permitted by the tool edge positions of the probe.

Result parameters

Depending on the measurement variant, measuring cycle CYCLE974 returns the following result values in the GUD5 module:

_OVR [0]	REAL	Setpoint for measuring axis
_OVR [1]	REAL	Setpoint for abscissa
_OVR [2]	REAL	Setpoint for ordinate
_OVR [3]	REAL	Setpoint for applicate
_OVR [4]	REAL	Actual value for measuring axis
_OVR [8] [1)]	REAL	Upper tolerance limit for measuring axis
_OVR [12] [1)]	REAL	Lower tolerance limit for measuring axis
_OVR [16]	REAL	Difference for measuring axis
_OVR [20] [1)]	REAL	Offset value
_OVR [27] [1)]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVR [29] [1)]	REAL	Dimensional difference
_OVR [30]	REAL	Empirical value
_OVR [31] [1)]	REAL	Mean value
_OVI [0]	INTEGER	D number or ZO number

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

_OVI [2]	INTEGER	Measuring cycle number
_OVI [4] 1)	INTEGER	Weighting factor
_OVI [5]	INTEGER	Probe number
_OVI [6] 1)	INTEGER	Mean value memory number
_OVI [7]	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number
_OVI [11] 2)	INTEGER	Status offset request

1) For workpiece measurement with tool offset only
2) For measuring cycle SW 6.2 and higher; only for zero offset

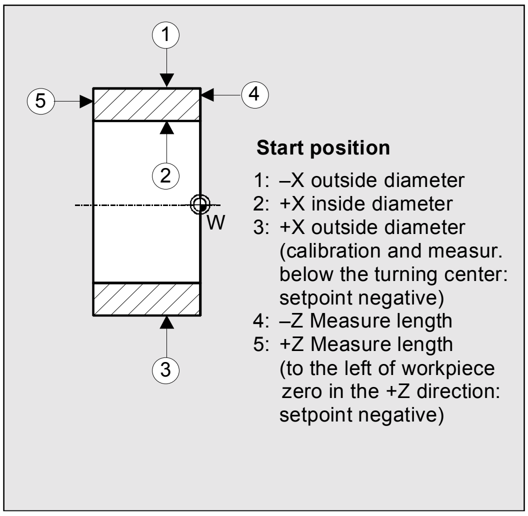
Measurement variants

Measuring cycle CYCLE974 permits the following measurement variants that are specified in the parameter _MVAR.

Value	*Meaning*
0	Single-point measurement
100	Single-point measurement ZO calculation
1000	Single-point measurement with reversal

Starting positions for the measurement variants

The starting positions before the cycle is called depend on the measurement variant selected.



840 D	840 D	FM-NC	810 D	840 Di
NCU 571	NCU 572			
	NCU 573			

6.5.1 CYCLE974 Single-point measurement ZO calculation

Function

With this measurement variant, the actual value of a blank is acquired with reference to the workpiece zero in the selected measuring axis.

An empirical value from the GUD5 module can be included with the correct sign.

The automatic offset in the ZO memory is **additive** depending on the value of the parameter _KNUM. If a fine offset is active (MD 18600: MM_FRAME_FINE_TRANS), an additive ZO will be implemented in it, otherwise it is implemented in the coarse offset.

Precondition

If necessary, the workpiece must be positioned in the correct angular spindle position with SPOS before the cycle is called.

The probe must be calibrated in the measuring direction and called **with** tool offset. The tool type is 500. Measuring cycle SW 6.2 and higher also allows you to enter tool type 580 (probe). The tool edge position can be 5 to 8.

The maximum diameter to be measured depends on the traverse range of the turret slide in the plus X direction.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Parameters

_MVAR	100	Measurement variant: Single-point measurement ZO calculation
_SETVAL	REAL[1)]	Setpoint, with reference to the workpiece zero
_MA	1, 2, 3[1)]	Measuring axis
_KNUM	0 No automatic ZO compensation; 1...99 Automatic ZO compensation in G54...G57, G505...G599	With/without automatic ZO calculation
Measuring cycles SW 4.4 and higher	1000 Automatic ZO compensation in basic frame G500[2)]	
Measuring cycles SW 6.2 and higher[2)]	1011...1026 automatic ZO compensation in 1st to 16th basic frame (channel)	($P_CHBFR[0]...$P_CHBFR[15])
	2000 automatic ZO compensation in system frame	scratching system frame ($P_SETFR)
	9999 automatic ZO compensation in an active frame	settable frames G54..G57, G505...G599 or for G500 in last active basic frame according to $P_CHBFRMASK (most significant bit)

1) As of measuring cycles SW 5.4, measuring is also possible in the 3rd axis of the plane (Y in G18), provided that this axis exists. Moreover, for measurement in the 3rd axis of the plane with active G18 (measurement in the Y axis), the same setpoint parameterization can be used as for measurement in the X axis (transverse axis), if CHBIT[19]=1 is set in module GUD6. The offset is then stored in the X component of the selected ZO memory.
2) As of measuring cycles SW 5.3, compensation is carried out in the last basic frame (per MD 28081: MM_NUM_BASE_FRAMES) if more than one is available. If measuring cycles higher than SW 5.3 are used at a control with SW 4, parameter _SI[1] in the GUD 6 module must be set to 4!
3) **Measuring cycles version SW 6.2 and higher can only be used with NCK-SW 6.3 and higher.**

The following additional parameters are also valid:
_VMS, _TSA, _FA, _PRNUM, _EVNUM and **_NMSP**.
See Sections 2.2 and 2.3.
If the parameter _VMS has value 0, the default value of the measuring cycle is used for the variable measuring velocity.

6.5 CYCLE974 Workpiece measurement

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Procedure

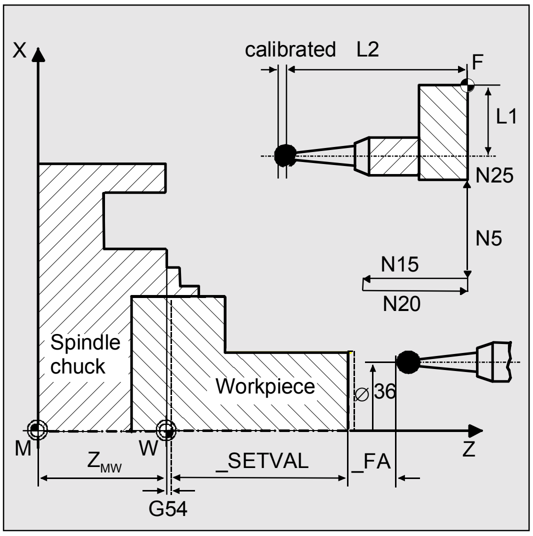
Position before the cycle is called
The probe must be positioned opposite the surface to be measured.

Position after the cycle has terminated
On completion of the measuring process, the probe is positioned facing the calibration surface at a distance corresponding to _FA · 1 mm.

Notice!
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane and measuring velocity are the same for both measurement and calibration. If the probe is used in the spindle for a powered tool, the orientation of the spindle must also be considered. Deviations can cause additional measuring errors.

Programming example

ZO calculation at a workpiece



ZO_CALCULATION_1	
N01 G18 T8 D3	Call probe (tool type 500, SL 7)
N05 G0 G90 G54 X36 Z100	Starting position before the cycle is called
N10 _MVAR=100 _SETVAL=60 _MA=1 _TSA=1 _KNUM=1 _EVNUM=0 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	Parameters for cycle call
N15 CYCLE974	Measurement in the Z direction
N20 G0 Z100	Retraction in Z
N25 X114	Retraction in X
N90 M30	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.5.2 CYCLE974 Single-point measurement

Function

With this measurement variant the actual value of a workpiece is acquired with reference to the workpiece zero in the selected measuring axis.

An empirical value from the GUD5 module can be included with the correct sign. A mean value derivation over several parts is possible as an option.

The automatic tool offset is **additive** depending on the value of the parameter _KNUM; observance of set tolerance ranges is checked.

Precondition

If necessary, the workpiece must be positioned in the correct angular spindle position with SPOS before the cycle is called.

The probe must be calibrated in the measuring direction and called **with** tool offset. The tool type is 500. Measuring cycle SW 6.2 also allows you to enter tool type 580 (probe). The tool edge position can be 5 to 8.

The maximum diameter to be measured depends on the traverse range of the turret slide in the plus X direction.

6.5 CYCLE974 Workpiece measurement

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Parameters

_MVAR	0	Measurement variant: Single-point measurement
_SETVAL	REAL[2)]	Setpoint (according to drawing)
_MA	1, 2, 3[1)]	Measuring axis
_KNUM	0 no automatic tool offset > 0 automatic tool offset (D number)	With/without automatic tool offset
_TNUM	1, 2, 3, ...	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)

1) As of measuring cycles SW 5.4, it is also possible to carry out measurement in the 3rd axis of the plane (with G18 in Y), provided that this axis exists.
2) Setting _CHBIT[19]=1 in module GUD6 enables the same setpoint parameterization to be used for measurement in the Y axis (3rd axis of the plane) with active G18 as for measurement in the X axis (transverse axis). The tool offset is then also in L1 (active length in X) if not specified differently in _KNUM.

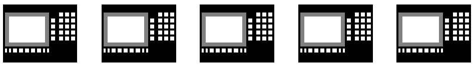
The following additional parameters are also valid:
_VMS, _TZL, _TMV, _TUL, TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP, and **_K**.

See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called
The probe must be positioned opposite the surface to be measured.

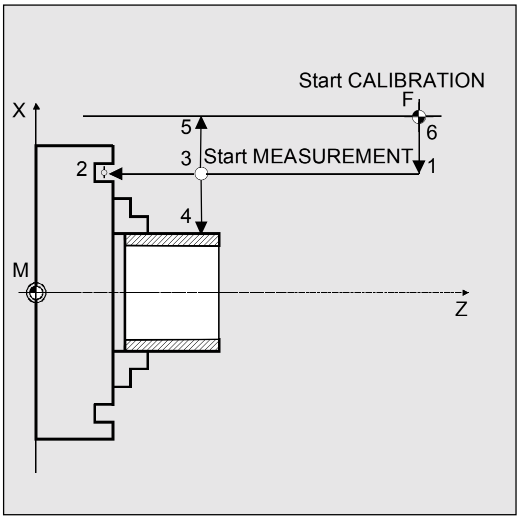
Position after the cycle has terminated
On completion of the measuring process, the probe is positioned facing the measuring surface at a distance corresponding to _FA · 1 mm.



Notice!
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane and measuring velocity are the same for both measurement and calibration. If the probe is used in the spindle for a powered tool, the orientation of the spindle must also be considered. Deviations can cause additional measuring errors.

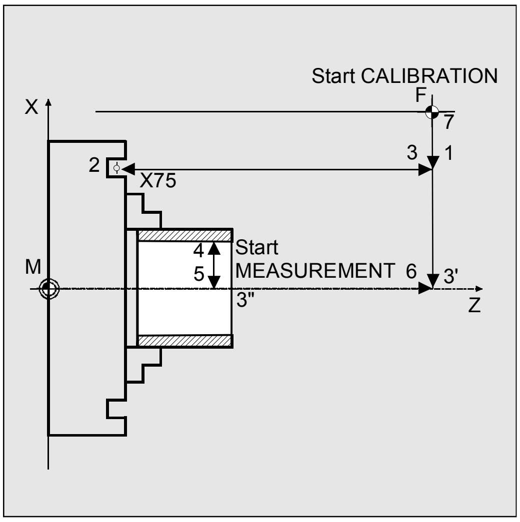
Procedure for external measurement (with calibration) for a probe with tool edge position 7:

1, 2 Self-generated approach paths for calibration
3 Retraction paths for position Z
4 Self-generated approach path for measuring on the outside diameter
5 Retraction paths to the initial point or approach another measuring point



Procedure for internal measurement (with calibration) for a probe with tool edge position 7:

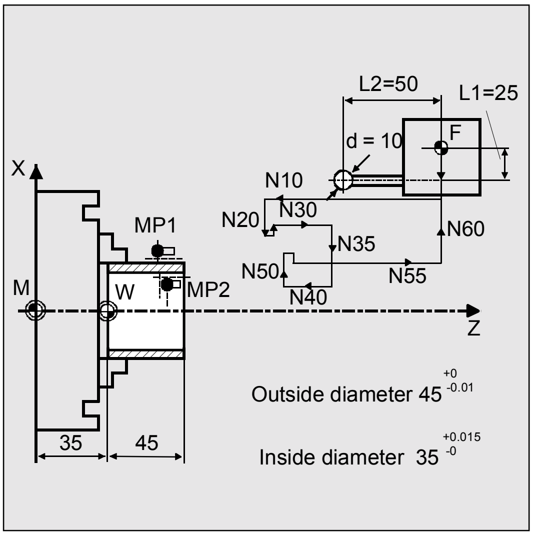
1, 2 Self-generated approach paths for calibration
3 Retraction paths for positions in Z and X
4 Self-generated approach path for measuring on the inside diameter
5, 6 Retraction paths to the initial point or approach another measuring point



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Single-point measurement at outside and inside diameters



SINGLE_POINT_MEASUREMENT	
N05 G18 T1 D1 DIAMON	Call probe (tool type 500, SL 7)
N10 G0 G90 G54 Z30 X90	Preposition probe
N15 _MVAR=0 _SETVAL=45 _TUL=0 _TLL=-0.01 _MA=2 _TNUM=7 _KNUM=1 _EVNUM=13 _K=2 _TZL=0.002 _TMV=0.005 _TDIF=0.04 _TSA=0.5 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	Parameters for cycle call
N20 CYCLE974	Measurement on the outside diameter
N30 G0 Z60	Position probe opposite MP2
N35 X0	
N40 Z40	
N45 _SETVAL=35 _TUL=0.015 _TLL=-0 _TNUM=8 _EVNUM=14	
N50 CYCLE974	Measurement on the inside diameter
N55 G0 Z110	Retraction in Z
N60 X90	Retraction in X
N65 M30	

840 D	840 D	FM-NC	810 D	840 Di
NCU 571	NCU 572			
	NCU 573			

6.5.3 CYCLE974 Single-point measurement with reversal

Function

With this measurement variant the workpiece actual value is ascertained with reference to the workpiece zero in the measuring axis by acquiring two opposite points on the diameter. Before taking the first measurement, the workpiece is positioned at the angular position programmed in parameter _STA1 with SPOS and the 180° reversal is automatically generated by the cycle before the second measurement.

An empirical value from the GUD5 module can be included with the correct sign. A mean value derivation over several parts is possible as an option.

The automatic tool offset is **additive** depending on the value of the parameter _KNUM; observance of set tolerance ranges is checked.

Precondition

The probe must be calibrated in the measuring direction and called **with** tool offset. The tool type is 500. Measuring cycle SW 6.2 and higher also allows you to enter tool type 580 (probe). The tool edge position can be 5 to 8.

The maximum diameter to be measured depends on the traverse range of the turret slide in the plus X direction.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Parameters

_MVAR	1000	Measurement variant: Single-point measurement with reversal
_SETVAL	REAL2)	Setpoint (according to drawing)
_MA	1, 2, 31)	Measuring axis
_STA1	REAL, positive	Initial angle
_KNUM	0 no automatic tool offset > 0 automatic tool offset (D number)	With/without automatic tool offset
_TNUM	1, 2, 3, ...	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)

1) As of measuring cycles SW 5.4, it is also possible to carry out measurement in the 3rd axis of the plane (with G18 in Y), provided that this axis exists.
2) Setting _CHBIT[19]=1 in module GUD6 enables the same setpoint parameterization to be used as for measurement in the X axis (transverse axis) for measurement in the Y axis (3rd axis of the plane) with active G18. The tool offset is then also in L1 (active length in X) if not specified differently in _KNUM.

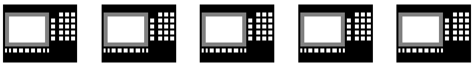
The following additional parameters are also valid:
_VMS, _TZL, _TMV, _TUL _TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP, and **_K**.

See Sections 2.2 and 2.3.

Procedure

Position before the cycle is called
The probe must be positioned opposite the surface to be measured.

Position after the cycle has terminated
On completion of the measuring process, the probe is positioned facing the measuring surface at a distance corresponding to _FA · 1 mm.



840 D
NCU 571

840 D
NCU 572
NCU 573

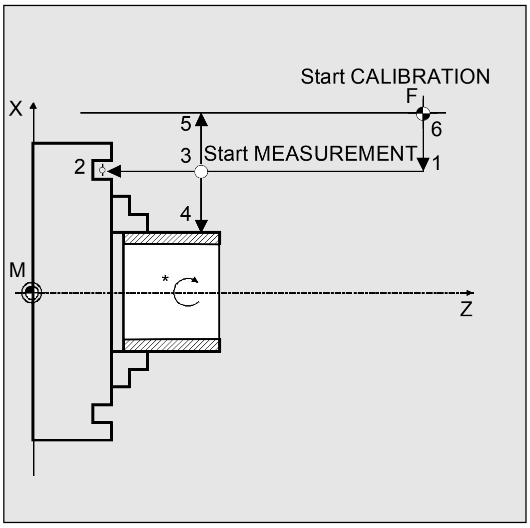
FM-NC

810 D

840 Di

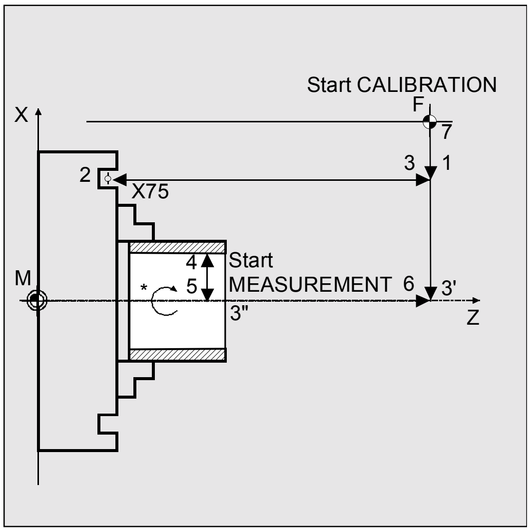
Notice!
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane and measuring velocity are the same for both measurement and calibration. If the probe is used in the spindle for a powered tool, the orientation of the spindle must also be considered. Deviations can cause additional measuring errors.

Procedure for external measurement (with calibration) for a probe with tool edge position 7:



1, 2 Self-generated approach paths for calibration
3 Retraction paths for position Z
4 Self-generated approach path for measuring on the outside diameter
5 Retraction paths to the initial point or approach another measuring point
- Retraction to 4, 180° reversal
  2nd approach of 4 automatically by cycle

Procedure for internal measurement (with calibration) for a probe with tool edge position 7:

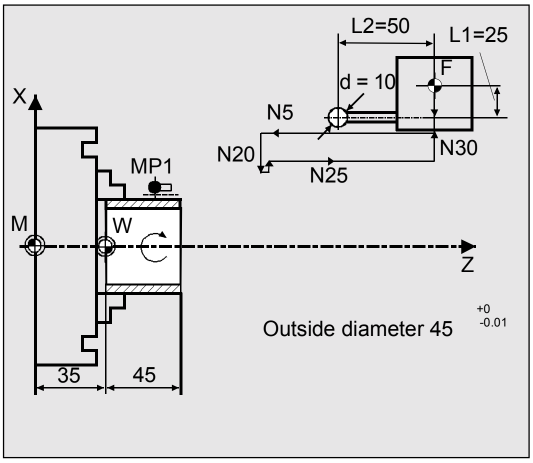


1, 2 Self-generated approach paths for calibration
3 Retraction paths for positions in Z and X
4 Self-generated approach path for measuring on the inside diameter
5, 6 Retraction paths to the initial point or approach another measuring point
- Retraction to 4, 180° reversal
  2nd approach of 4 automatically by cycle

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Single-point measurement at outside diameter



REVERSAL_MEASUREMENT	
N01 G18 T1 D1 DIAMON	Call probe (tool type 500, SL 7)
N05 G0 G90 G54 Z30 X90	Preposition probe
N10 _MVAR=1000 _SETVAL=45 _TUL=0 _TLL=-0.01 _MA=2 _STA1=0 _KNUM=2 _TNUM=11 _EVNUM=20 _K=1 _TZL=0.002 _TMV=0.04 _TDIF=0.2 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=3	Parameters for cycle call
N20 CYCLE974	Measuring cycle call
N25 G0 Z110	Retraction in Z
N30 X90	Retraction in X
N35 M30	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.6 CYCLE994 Two-point measurement

Programming

CYCLE994

Function

The measuring cycle ascertains the actual value of the workpiece with reference to the workpiece zero and calculates the setpoint/actual value difference. This is done automatically by approaching two opposite measuring points on the diameter.

The sequence of measurements defined in the cycle – 1st measuring point above on the diameter, 2nd measuring point below arises because a protection zone that can be programmed in parameters _SZA and _SZO is taken into account.

An empirical value stored in the GUD5 module and a mean value derivation over several parts can also be taken into account as options.
The cycle checks that a set tolerance range for the measured deviation is not violated and automatically corrects the tool offset memory selected in _KNUM.

Precondition
If necessary, the workpiece must be positioned in the correct angular spindle position with SPOS before the cycle is called.

The probe must be called **with** tool offset. Tool type 500 must be specified. Measuring cycle SW 6.2 and higher also allows you to enter tool type 580 (probe). The tool edge position can be 5 to 8.
The diameter to be measured depends on the traverse range of the turret slide in the negative direction and on the length offsets of the probe.

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Extension on measuring cycles SW 5.4 and higher
The measuring cycle can now be used for measurement without previous calibration. In place of the trigger values, the probe tip diameter entered in the data field of the probe _WP[_PRNUM-1,0] is then used in the calculation.
The function is controlled with bit:
_CHBIT[7] = 1: without inclusion of the trigger values
_CHBIT[7] = 0: as previously

Measurement variants

Measuring cycle CYCLE994 permits the following measurement variants that are specified in the parameter _MVAR.

Value	*Meaning*
1	Two-point measurement with programmed protection zone (for inside measurement only)
2	Two-point measurement with programmed protection zone (for inside measurement without protection zone)

Result parameters

Measuring cycle CYCLE994 returns the following result values in the GUD5 module:

_OVR [0]	REAL	Setpoint for diameter/radius	
_OVR [1]	REAL	Setpoint diameter/radius in abscissa	with _MA=1 only
_OVR [2]	REAL	Setpoint diameter/radius in ordinate	with _MA=2 only
_OVR [4]	REAL	Actual value for diameter/radius	
_OVR [5]	REAL	Actual value diameter/radius in abscissa	with _MA=1 only
_OVR [6]	REAL	Actual value diameter/radius in ordinate	with _MA=2 only
_OVR [8]	REAL	Upper Tolerance limit for diameter/radius	
_OVR [12]	REAL	Lower tolerance limit for diameter/radius	
_OVR [16]	REAL	Difference for diameter	
_OVR [17]	REAL	Difference diameter/radius in abscissa	with _MA=1 only
_OVR [18]	REAL	Difference diameter/radius in ordinate	with _MA=2 only
_OVR [20]	REAL	Offset value	
_OVR [27])	REAL	Zero offset area	
_OVR [28]	REAL	Safe area	
_OVR [29]	REAL	Dimensional difference	
_OVR [30]	REAL	Empirical value	
_OVR [31]	REAL	Mean value	
_OVI [0]	INTEGER	D number	

840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

_OVI [2]	INTEGER	Measuring cycle number
_OVI [4]	INTEGER	Weighting factor
_OVI [5]	INTEGER	Probe number
_OVI [6]	INTEGER	Mean value memory number
_OVI [7]	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number

Parameters

_MVAR	1 or 2	Measurement variant: Two-point measurement with/without programmed protection zone
_SETVAL	REAL[2)]	Setpoint (according to drawing)
_MA	1, 2, 3[1)]	Measuring axis
_SZA	REAL	Protection zone on workpiece abscissa[2)]
_SZO	REAL	Protection zone on workpiece ordinate[2)]
_KNUM	0 no automatic tool offset > 0 automatic tool offset (D number)	With/without automatic tool offset
_TNUM	1, 2, 3, ...	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)

1) As of measuring cycles SW 5.4, it is possible to carry out measurement in the 3rd axis of the plane (with G18 in Y), provided that this axis exists.
2) For measurement in the 3rd axis (for G18 in Y, _SZO applies in this axis, _SZA applies unchanged in the 1st axis in the plane (Z axis for G18), reversal is performed in the 1st axis of the plane (Z axis for G18).
Setting _CHBIT[19]=1 in module GUD6 enables the same setpoint and protection zone parameterization to be used for measurement in the 3rd axis (measurement in the Y axis) with active G18 as for measurement in the X axis (transverse axis). The tool offset is then also in L1 if not specified differently in _KNUM.

The following additional parameters are also valid:
_VMS, _TZL, _TMV, _TUL _TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP, and **_K**.
See Sections 2.2 and 2.3.

840 D NCU 571	840 D NCU 572 NCU 573	FM-NC	810 D	840 Di

Procedure

Position before the cycle is called

The probe must be positioned opposite the surface to be measured.

Position after the cycle has terminated

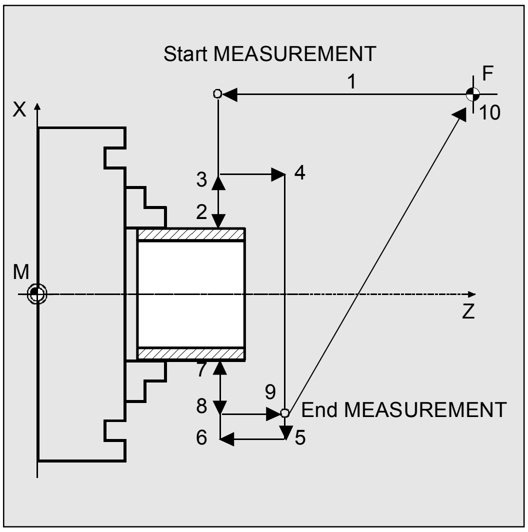
After measuring has been completed, the probe is outside the protection zone.

Notice!

Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane and measuring velocity are the same for both measurement and calibration. If the probe is used in the spindle for a powered tool, the orientation of the spindle must also be considered. Deviations can cause additional measuring errors.

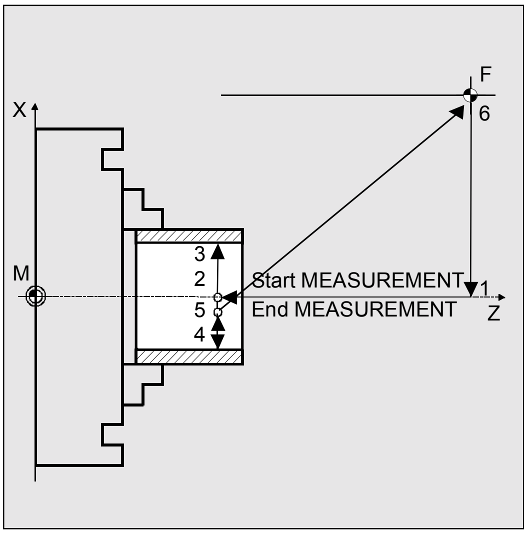
Procedure for outside measurement:

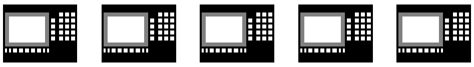
1 Approach path outside diameter
2–9 Self-generated traverse paths for measurement on the outside diameter
10 Retraction to the initial point



Procedure for inside measurement:

1,2 Approach paths for inside diameter
3–5 Self-generated traverse paths for measurement on the inside diameter
6 Retraction paths to the initial point

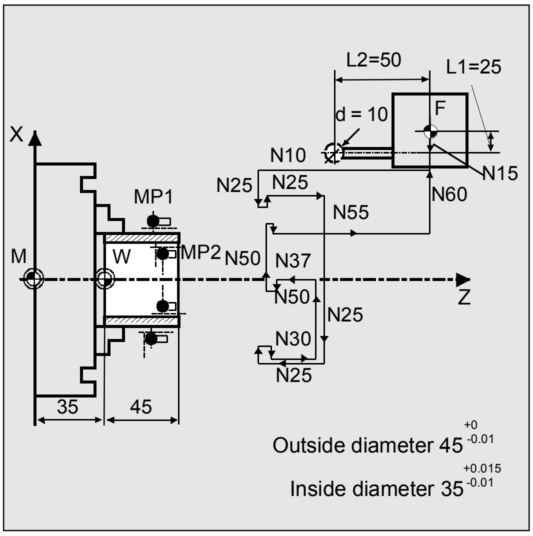




840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Two-point measurement, outside and inside



TWO_POINT_MEASUREMENT	
N03 T1 D1 DIAMON	Call of probe
N10 G0 G54 Z30 X60	Preposition probe opposite MP1 and ZO selection
N15 _MVAR=2 _SETVAL=45 _TUL=0 _TLL=-0.01 _MA=2 _SZA=55 _SZO=55 _TNUM=8 _KNUM=3 _EVNUM=3 _K=3 _TZL=0.002 _TMV=0.005 _TDIF=0.04 _TSA=0.5 _VMS=0 _NMSP=1 _FA=2	Parameter assignment for 1st cycle call
N25 CYCLE994	Two-point measurement outside with protection zone (MP1)
N30 G0 Z55	Position probe opposite MP2
N35 X20	
N37 Z30	
N40 _SETVAL=35 _TUL=0.015 _TNUM=9 _KNUM=4 _EVNUM=4	Parameter assignment for 2nd cycle call
N50 CYCLE994	Two-point measurement inside without protection zone (MP2)
N55 G0 Z110	Retraction in Z
N60 X60	Retraction in X
N65 M30	

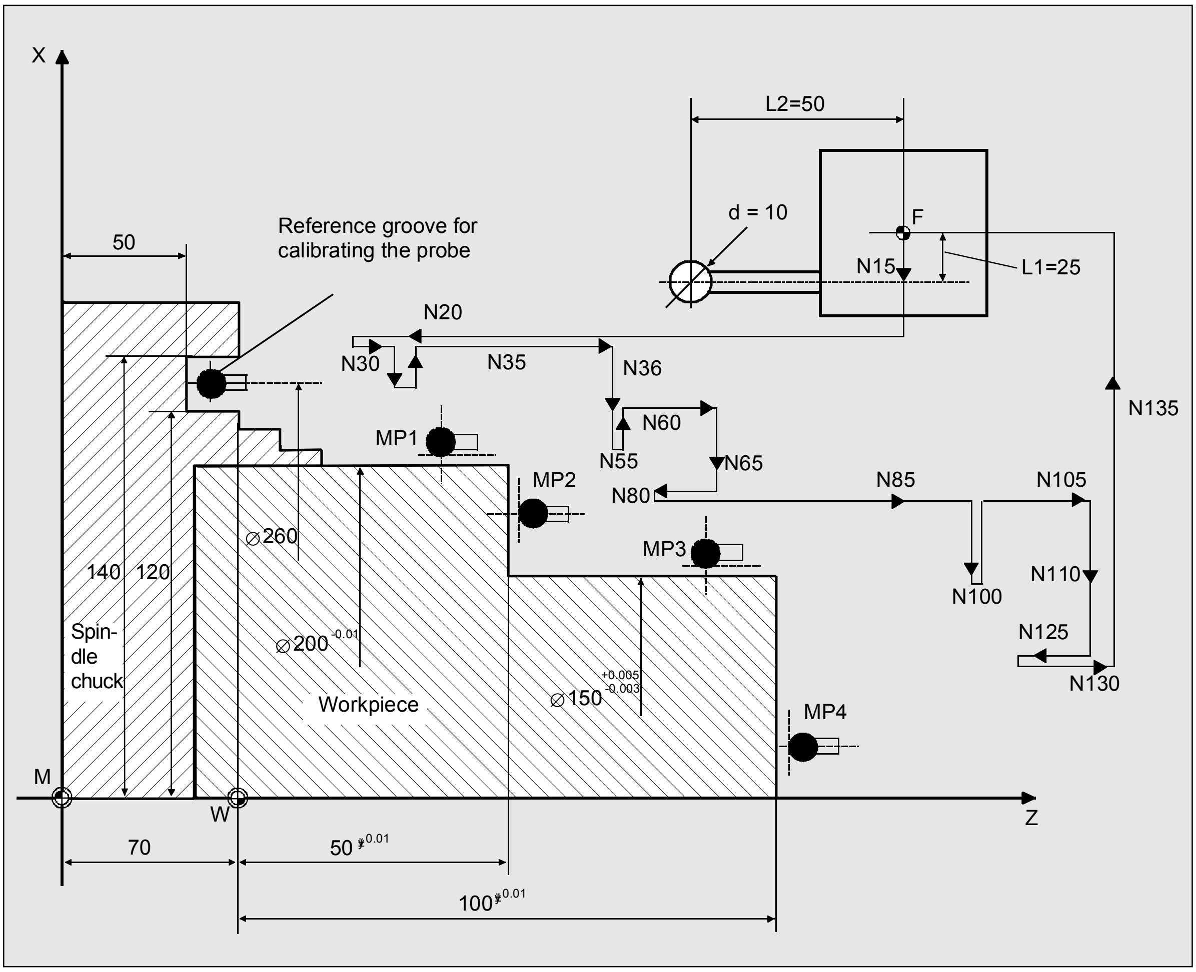
840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

6.7 Complex example for workpiece measurement

(CYCLE973, CYCLE974)

Explanation

The workpiece shown in the figure is to be measured with a probe.



840 D NCU 571 | 840 D NCU 572 NCU 573 | FM-NC | 810 D | 840 Di

Programming example

Calibrate workpiece probe, measure workpiece with CYCLE973 and CYCLE974

PART_1_MEASUREMENT

N05 T1 D1 DIAMON	Select probe
N06 SUPA G0 X300 Z150	Approach starting position in X and Z, from which it is possible to approach the reference groove for calibration without collision
N10 _MVAR=13 _MA=1 _MD=1 _CALNUM=1 _TZL=0 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	Parameters for calibration in reference groove
N20 CYCLE973	Calibrate probe in the minus direction
N25 _MA=2	Other measuring axis
N30 CYCLE973	Calibrate probe in the minus direction
N35 G54 G0 Z40 **N36 X220**	Select zero offset and position probe opposite measuring point 1
N40 _MVAR=0 _SETVAL=200 _TUL=0 _TLL=-0.01 _MA=2 _KNUM=8 _TNUM=3 _K=2 _TZL=0.002 _TMV=0.005 _TDIF=0.2 _TSA=0.3 _PRNUM=1	Define parameters for measurement
N55 CYCLE974	Measure MP1
N60 G0 Z70	Position probe opposite MP2
N65 X175	
N70 _MA=1 _SETVAL=50 _TUL=0.01 _KNUM=9 _TNUM=4	Define parameters for measurement in another axis
N80 CYCLE974	Measure MP2
N85 G0 Z180	Position probe opposite MP3
N90 _MA=2 _SETVAL=150 _TUL=0.005 _TLL=-0.003 _KNUM=1 _TNUM=5	Define parameters for measurement
N100 CYCLE974	Measure MP3
N105 G0 Z150	Position probe opposite MP4
N110 X50	
N115 _MA=1 _SETVAL=100 _TUL=0.01 _TLL=-0.01 _KNUM=2 _TNUM=6	Define parameters for measurement
N125 CYCLE974	Measure MP4
N130 G0 SUPA Z250	Retraction in Z
N135 SUPA X280	Retraction in X
N140 M30	

■

840 D
NCU 571

840 D
NCU 572
NCU 573

FM-NC

810 D

840 Di

Notes

Miscellaneous Functions

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

7.1 Logging of measuring results

In SW 4.3 and higher, the standard measuring cycles support logging of measuring results in a file in the control.

There are no special hardware requirements for logging measurement results. It is executed solely by the software.

7.1.1 Storing the log

Function

The log file is stored in the directory where the calling program is located. You can specify the file name for the log file. The restrictions that apply to program names also apply here. So, only letters, numbers and underscores are permitted, and the name must commence with two letters or a letter followed by an underscore. The file always has the extension "MPF".

The maximum length of the log file is set in MD 11420. If the system detects during writing that a data record is too long, another log file is automatically created. Underscore and a digit are added to the name specified in _PROTNAME[1] and the message
"New log file has been created" output.

In this way, up to 10 subsequent logs can be stored in the control.
After the 10th log operation is halted and the message **"Please specify new log name"** output.
After restart, operation is continued. If a log file with the same name already exists before logging is started, then it is deleted before writing is started.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

7.1.2 Handling of log cycles

Function

The log is enabled and disabled via the program (CYCLE100/CYCLE101). This requires a cycle call without setting any parameters.
After disabling the log function, the log files must be unloaded from the part program memory ("Part program" directory) (MMC 102/103) or read out via RS-232-C.

Print out the log file in

- Word or WordPad (Courier font)
- WINDOWS 95 editor
- MS DOS editor

Procedure

When used with the measuring cycles, it is sufficient to activate and deactivate the logging with CYCLE100 and CYCLE101 respectively. Logging is carried out with the parameters described in Subsections 7.1.3 to 7.1.5.
The logging sequence is implemented in CYCLE105, CYCLE106, CYCLE113 and CYCLE118. These cycles are called internally in the context of measuring cycles.

The log cycles may be used independently of the measuring cycles. CYCLE100 and CYCLE101, and CYCLE105 and CYCLE106 are called explicitly in this context. CYCLE113 and CYCLE118 are called internally. You can also call them separately for other purposes.

CYCLE100 Log ON
After the log is enabled, an existing file with the specified name is automatically deleted in the control. All follow-up logs with _PROTNAME[1]_digit are only deleted when the preceding logs overflow.
The log is reopened and the header is entered. The internal status variables are set.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

CYCLE101 Log OFF

Disables the logging function and resets the internal flag.

CYCLE105(int par1) Generate log contents

This cycle generates up to 4 lines of log contents (lines of values) according to the entries in the GUD variables.
It allows you to generate only value lines or only the log header depending on the setting for par1.

Transfer parameters:	0 output value block
	1 output header

CYCLE106(int par1)
Log sequential controller

This cycle controls how logging is executed.

Transfer parameters:	1 output header
	2 output value block.

The cycle is called by CYCLE100 automatically when the log is activated. It deletes all old log files with the same name as required, creates follow-up log files and monitors the page layout of the log.

CYCLE113(int par1,string[10] par2)
Read time and date from system

par1 = 1	Read date and return it in par2
par1 = 2	Read time and return it in par2

CYCLE118(real par1,int par2,string[12] par3, int par4, int par5)

This cycle formats the numerical values according to the places after the decimal point specified in parameter_DIGIT.

par1	Real value which is to be formatted
par2	Number of decimal places
String[12] par3	Formatted return value
par4	Control value
par5	Set to 0

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

7.1.3 Selecting the log contents

Function

The measurement result log contains parts that are fixed and some that can be set. It always contains:

- Measuring cycle
- Measurement variant (cycle name, value of _MVAR)

The following additional data can be included in a log:

- Time (specification _TIME)
- Axis names of the corresponding measuring axes
- Specification _AXIS:
  The axis name is entered automatically according to the measuring axis entered in _MA.
- Specification _AXIS1...3:
  - AXIS: Axis name of abscissa in selected plane
  - AXIS: Axis name of ordinate in selected plane
  - AXIS: Axis name of applicate in selected plane
- All result data provided by the measuring cycle in the _OVR field.
- R parameters
- Comment texts

The logging values to be selected must correspond to the measuring cycle and the selected measurement variant. This makes for versatile adaptation of the contents of the log to meet your requirements.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Procedure

Specification of the log contents is conducted via the variable `_PROTVAL[ ]`.
The strings stored in `_PROTVAL[0]` and `_PROTVAL[1]` are used as header lines for the log (see example in Subsection 6.1.7, Lines 8 – 10).
`_PROTVAL[2] ... [5]` specify the line contents of the individual log lines.

If you change the measuring cycle or the measurement variant, you may have to adapt `_PROTVAL[2] ... [5]` (see example in Subsection 6.1.7).

Up to 4 lines can be defined.
You can log the

- R parameters,
- _OVR[],
- axis names,
- times,
- free comments and
- strings saved in _TXT[] (GUD6).

Commas are used as separators.

Example

_PROTVAL[2]="R27,_OVR[0],_OVR[4],_OVR[8],_OVR[12],_OVR[16],_TIME"
_PROTVAL[3]="_AXIS,_OVR[1],_OVR[5],_OVR[9],_OVR[13],_OVR[17], INCH"
_PROTVAL[4]="_AXIS,_OVR[2],_OVR[6],_OVR[10],_OVR[14],_OVR[18], Metro"

In this example R27 stands for a variable freely entered into the log. The texts "INCH" and "Metro" at the end of the second and third line are examples for comment texts. This makes it easy, for example, to append dimensions after the measurement results.
Logging of variables always has priority, i.e. when specified format limits are exceeded they are modified and an alarm without terminating execution is generated.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

7.1.4 Log format

Programming

The following values can be specified for the log format:

_PROTFORM[0]	Number of line per page with log header
_PROTFORM[1]	Number of characters per line
_PROTFORM[2]	First page number
_PROTFORM[3]	Number of header lines
_PROTFORM[4]	Number of value lines in the log
_PROTFORM[5]	Column width/variable column width
_PROTSYM[0]	Separators between the values in the log
_PROTSYM[1]	Special characters for identification when tolerance limits are exceeded
_DIGIT	Number of decimal places

Explanation

The number of decimal places can be set via the variable _DIGIT in GUD6 (display precision).

The value set in parameter _PROTFORM[0] determines when a log header with title lines is output again. If this parameter is set to zero, the log only contains a header at the beginning.

Default settings exist for all these parameters which are set when the GUD modules are read in (see Subsection 6.1.6).

The value of parameter _PROTFORM[5] determines the column width of the log. If the parameter=0, the column width of each column is derived from the string lengths (number of characters between the commas) of the 1st header line (_PROTVAL[0]). This makes it possible to individually define the width of each column. If the value>0, each column is formatted to this value if the string length allows it.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

7.1.5 Log header

Function

The log header can be customized by the operator or a log header prepared by the standard measuring cycles can be used.

Procedure

The header is selected via the measuring cycle data bit `_CBIT[11]`. However, the standard log also allows you to customize up to three lines.

The contents of the header are stored in an array of string variables `_HEADLINE[10]`, which are automatically output when logging (CYCLE100) is enabled. The maximum number of header lines can be changed during measuring cycle start-up (`_PROTFORM[3]`).

Each field element contains a line for the log header.

Explanation

Customized log header
The contents of the string array `_HEADLINE[ ]` are entered in line 1 ff. The number of header lines can be defined by the user (according to the length of the `_HEADLINE` array).

Predefined log header
All variable parts are in bold formatting, that is:
Page number, program name,
Line 5, 6, 7 (`_HEADLINE[0-2]`) ff. and
Line 9 (`_PROTVAL[0]`)
Line 10 (`_PROTVAL[1]`)

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Line 1	Date: 98/09/15			Time: 10:05:30		Page: **1**
Line 2						
Line 3	Program: **MEASPROGRAM_1**					
Line 4						
Line 5	**Part number: 123456789**					
Line 6	**Job number: 6878**					
Line 7	**Supervised by: Smith**		**Tel.: 1234**			
Line 8	--					
Line 9	**Measuring**	**, Axis**	**, Set**	**, Actual**	**, Difference**	**, Time**
Line 10	**point**		**value**	**value**		
Line 11	--					

When filling in the standard log header shown above the following program lines must be inserted in the main program before the measuring cycle is called:

```
DEF INT PARTNUM, JOBNUM
_CBIT[11]=0                                         ;Log with default header
PARTNUM=123456789   JOBNUM=6878  _PROTNAME[0]="MEASPROGRAM_1"
_PROTNAME[1] ="MY_LOG1"
_HEADLINE[0]="Part number:   "<<PARTNUM
_HEADLINE[1]="Job number:"<<JOBNUM
_HEADLINE[2]="Supervisor: Smith Tel.: 1234"
_PROTVAL[0]="Measurement , Axis , Set , Actual value , Difference , Time"
_PROTVAL[1]="point   ,   , value
```

7.1.6 Variable for logging

In the measuring cycle, data logging is controlled via the following data bit:

`_CBIT[11]=`	0	Standard log header
	1	User-defined log header

The following variables describe the contents of the measurement log:

Variable	*Type*	*Default value*	*Contents*
`_PROTNAME[2]`	STRING[32]	Blank string	`_PROTNAME[0]` = Name of the main program from which the log is written
		"SMC:PROT”	`_PROTNAME[1]` = Name of the log file
`_HEADLINE[6]`	STRING[80]	Blank string	`_HEADLINE[0] ... _HEADLINE[5]` The user can enter customized texts in these strings; they are included in the log

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

_PROTFORM[6]	INTEGER	60	_PROTFORM[0] = Number of lines per page
		80	_PROTFORM[1] = Number of characters per line
		1	_PROTFORM[2] = First page number
		5	_PROTFORM[3] = Number of header lines
		4	_PROTFORM[4] = Number of value lines in the log
		12	_PROTFORM[5] = Number of characters per column
_PROTSYM[2]	CHAR	";"	_PROTSYM[0] = Separators between the values in the log
		"#"	_PROTSYM[1] = Special characters for identification when tolerance limits are exceeded
_PROTVAL[13]	String[80]	See Example	_PROTVAL[0] = Contents of the header line (line 9)
			_PROTVAL[1] = Contents of the header line (line 10)
			_PROTVAL[2]...[5] = Specification of the values to be logged in successive lines

7.1.7 Example of measuring result log

Line 1	Date: 96/11/15			Time: 10:05:30		Page: **1**
Line 2						
Line 3	Program: **MEASPROGRAM_1**					
Line 4						
Line 5	**Part number: 123456789**					
Line 6	**Job number: 6878**					
Line 7	**Supervised by: Smith**		**Tel.: 1234**			
Line 8	---					
Line 9	**Measuring**	**, Axis**	**, Set**	**, Actual**	**, Difference**	**, Time**
Line 10	**point**		**value**	**value**		
Line 11	---					
Line 12	CYCLE978	, _MVAR	, 100			
Line 13	1	, Z	, 80.000	, 79.987	, -0.013	, 09:35,12
Line 14						
Line 15						
Line 16	CYCLE977	, _MVAR	, 102			
Line 17	2	, X	, 64.000	, 64.009	, 0.009	, 09:36,45
Line 18		, Y	, 38.000	, 37.998	, -0.002	, 09:37,35

Programming

The log shown above is created using the following program.
The example shows the user how to handle the log.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

```
%_N_MEASPROGRAM_1_MPF
;$PATH=/_N_MPF_DIR
;Measure ring inside and outside with measurement log
DEF INT PARTNUM, JOBNUM
; -----------   Set parameters for log   -----------------
_CBIT[11]=0                                       ;Log with default header
; -------------------------   Log header   -----------------------

PARTNUM=123456789    JOBNUM=6878                  ;Name of calling program
_PROTNAME[0]="MEASPROGRAM_1"
_PROTNAME[1] ="MY_LOG1"                           ;Name of log file
_HEADLINE[0]="Part number:   "<<PARTNUM
_HEADLINE[1]="Job number:"<<JOBNUM
_HEADLINE[2]="Supervisor: Smith Tel.: 1234"
; ----------------------   Log format   ----------------------    Formats: Default values from GUD5
_PROTSYM[0] =" , "   _PROTSYM[1] = " * "          ;Define separators and special characters
_PROTFORM[4]=2                                    ;Two value lines
; ----------------------   Log contents   ----------------------
; Header lines
_PROTVAL[0]="Measurement   , Axis   , Set   , Actual value    , Difference
_PROTVAL[1]="point   ,   , value"
; ----------------   Other value assignments   -----------------
R27=1                                             ;Assign counter for measurement log
; --------   Perform measurements with log   -------------
N100 G0 G17 G90 T3 D1 Z100 F1000                  ;Approach start position for measurement
N110 X70 Y90
;
_MVAR=100 _SETVAL=80 _MA=3 _TSA=2 _FA=2           ;Set measuring cycle parameters
...                                               ;Measurement variant: Measure surface with
; Contents of the value lines
_PROTVAL[2]="R27,_AXIS,_OVR[0],_OVR[4],_OVR[16],_TIME"
N150 CYCLE100                                     ;Activate log
N160 CYCLE978                                     ;Measure surface
N170 Z200                                         ;Retraction in Z
N180 X64 Y38                                      ;Position above shaft center
N185 Z130                                         ;Lower in Z
;
_MVAR=102  _SETVAL=70  _FA=2  _TSA=2  _ID=-20     ;Set measuring cycle parameters
...                                               ;Measurement variant: Measure shaft with ZO
_PROTVAL[2]="R27,_AXIS1,_OVR[1],_OVR[5],_OVR[17],_TIME"
_PROTVAL[3]=" ,_AXIS2,_OVR[2],_OVR[6],_OVR[18],_TIME"
R27=R27+1                                         ;Increase user-def. counter for
N190 CYCLE977                                     ;Measure shaft
N210 CYCLE101                                     ;Deactivate log
N220 Z200                                         ;Retraction in Z
N290 M2
```

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

7.2 Cycle support for measuring cycles

Function

In SW 4.3 and higher, cycle support for measuring cycles in the ASCII editor is provided as for the standard cycles.
With this support function, the parameters described as mandatory parameters are input for each measuring cycle. For the additional parameters the last values input are retained. Furthermore, it is possible to change the additional parameters.

The measuring cycles are selected in the editor by using the vertical soft keys. The soft key bar is divided according to measuring tasks, e.g. "Calibrate" and "Calibrate in hole" or "Tool probe". In this manner there is no 1:1 assignment between the soft keys and the measuring cycles.

In MMC SW 5 and higher, measuring cycle support is provided by the soft keys
[>] → [Support] → [New cycle] → [Measuring cycles]
in the extension menu of the editor.

In the edited program there are calls with parameter list, e.g.
CYCLE_976(...) for calibrating in hole,
CYCLE_CAL_TOOLSETTER(...) for calibrating the tool probe.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

7.2.1 Files for cycle support

Function

Measuring cycle support requires the following files:

- cov.com
  Configuring the soft keys for cycle selection
- sc.com
  Configuring the input screens for the individual parameters
- Auxiliary cycle*.spf
  Additional cycles with parameter list, which transfer the input parameters to the measuring cycle GUD variables and call the measuring cycles.

These files are combined in the following two archives on the measuring cycle diskette:

- mcsupp_1.com
- mcsupp_2.com.

7.2.2 Loading the cycle support

Function

The files mcsupp_1.com and mcsupp_2.com are loaded from diskette or via RS-232-C (V24) with "Data in" into the "Services" menu.

With the MMC 102/103 the auxiliary cycle programs (see list Subsection 6.2.3) must be transferred to the NCU with "Load".

The Power ON is executed.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

7.2.3 Assignment of calls and measuring cycles

Function

The following table provides an overview of:
Measuring task, Measuring Cycle, Call

Measuring task, function	Measuring cycle	Call in the program
Calibrate tool probe	CYCLE971, CYCLE972, CYCLE982	CYCLE_CAL_TOOLSETTER(...)
Calibrating a workpiece probe on surface	CYCLE973, CYCLE976	CYCLE_CAL_PROBE(...)
Calibrate workpiece probe in reference groove	CYCLE973	CYCLE_973(...)
Calibrate workpiece probe in hole	CYCLE976	CYCLE_976(...)
Measure milling tool on milling machines	CYCLE971	CYCLE_971(...)
Measure turning tool	CYCLE972	CYCLE_972(...)
Measure turning and milling tools on turning machines (measuring cycles SW 5.4 and higher)	CYCLE982	CYCLE_982(...)
Measure hole/shaft parallel to axis/at an angle	CYCLE977, CYCLE979	CYCLE_977_979A(...)
Measure groove/web parallel to axis/at an angle	CYCLE977, CYCLE979	CYCLE_977_979B(...)
Measure rectangle inside/outside parallel to axis	CYCLE977	CYCLE_977_979C(...)
Single-point measurement milling machine	CYCLE978	CYCLE_978(...)
Angle measurement	CYCLE998	CYCLE_998(...)
Measure corner	CYCLE961	CYCLE_961_W CYCLE_961_P
Single-point measurement turning	CYCLE974	CYCLE_974(...)
Two-point measurement	CYCLE994	CYCLE_994(...)
Additional parameters	-	CYCLE_PARA(...)

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

7.2.4 Description of parameterization cycles

Function

The individual parameterization cycles of the measuring cycles together with their input parameters are described below.
The parameter names in the table directly refer to the defining parameters of the measuring cycle in question in the GUD variables. If no parameter is given, it is a selection field in the input screenform for particular functions.

Calibrating in hole – CYCLE_976

With CYCLE_976
Soft keys [Measurem. milling] → [Probe calibrat.]
CYCLE976 can be parameterized to calibrate a reference hole.

Parameters

_SETVAL	REAL	Setpoint
	INTEGER	Selection: Angular position 0...Paraxial calibration/1...calibration at an angle
	INTEGER	Selection: Positional deviation 0...without/1...with specification of positional deviation
	INTEGER	Selection: Number of axes Number of axes to be calibrated, 1, 2 or 4
	INTEGER	Selection: Ball calculation 0...without/1...with calculation of probe ball diameter
_MA	INTEGER	Number of measuring axis
_MD	INTEGER	Determining the measuring direction 0...in positive direction/1...in negative direction
_STA1	REAL	Angle
_PRNUM	INTEGER	Probe number
		Selection: Hole type 0... hole center known/1...unknown

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Calibration in groove – CYCLE_973

With CYCLE_973

Soft keys [Measurem. turning] → [Probe calibrat.]

CYCLE973 can be parameterized to calibrate a reference groove.

Parameters

_SETVAL	REAL	Setpoint
	INTEGER	Selection: Positional deviation 0...without/1...with specification of positional deviation
	INTEGER	Selection: Number of axes Number of axes to be calibrated, 1, 2
	INTEGER	Selection: Ball calculation 0...without/1...with calculation of probe ball diameter
_MA	INTEGER	Number of measuring axis
_MD	INTEGER	Determining the measuring direction 0...in positive direction/1...in negative direction
_CALNUM	INTEGER	Selection of calibration groove with number
_PRNUM	INTEGER	Probe number

Calibration on surface – CYCLE_CAL_PROBE

With CYCLE_CAL_PROBE

Soft keys [Measurem. turning] → [Probe calibrat.] → [Plane]

or

Soft keys [Measurem. milling] → [Probe calibrat.] → [Plane]

measuring cycles CYCLE973 and 976 can be parameterized to calibrate a surface.

Parameters

	INTEGER	Selection: Cycle number 976... for CYCLE976 (milling machine), 973... for CYCLE973 (turning machine)
_SETVAL	REAL	Calibration setpoint with respect to workpiece zero
_MA	INTEGER	Number of measuring axis
_MD	INTEGER	Measurement direction
_PRNUM	INTEGER	Probe number
_MVAR	INTEGER	Selection: Measuring variant (for CYCLE976 only) 0: Calibration on any surface 10000: Calibration in 3rd axis with calculation of probe length.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Calibrating a tool measuring probe – CYCLE_CAL_TOOLSETTER

With CYCLE_CAL_TOOLSETTER

Soft keys [Measurem. turning] → [Calibrat. TL probe] (CYCLE971)

or

Soft keys [Measurem. milling] → [Calibrat. TL probe] (CYCLE982)

measuring cycles CYCLE971, 972 and CYCLE982 can be parameterized to calibrate a tool measuring probe.

Parameters

	INTEGER	Selection: Cycle number 971... for CYCLE971 (milling machine), 972... for CYCLE972 (turning machine) 982... for CYCLE982 (turning machine, turning and milling tools)
_MA	INTEGER	Number of measuring axis and for CYCLE972 also the offset axis
_PRNUM	INTEGER	Probe number
	INTEGER	**only for CYCLE971** Selection: Measurement variant 0...absolute calibration/1...incremental calibration
_FA	REAL	Measuring path

Measuring turning tools – CYCLE_972

CYCLE_972 can be used to parameterize CYCLE976 to gauge tools.

No longer parameterized by new measuring cycle support in measuring cycles SW 6.2 and higher.

Parameters

_MA	INTEGER	Number of measuring axis

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Measuring milling tools – CYCLE_971

With CYCLE_971

Soft keys [Measurem. milling] → [Tool measurem.]

CYCLE971 can be parameterized for tool measurement.

Parameters

_MVAR	INTEGER	Measurement variant
_MA	INTEGER	Number of measuring axis
_ID	REAL	Offset
_PRNUM	INTEGER	Probe number
_MFS[0]	REAL	Feed 1st probing (only with _CBIT[12]=1)
_MFS[1]	REAL	Speed 1st probing
_MFS[2]	REAL	Feed 2nd probing
_MFS[3]	REAL	Speed 2nd probing
_MFS[4]	REAL	Feed 3rd probing
_MFS[5]	REAL	Speed 3rd probing

Tool measurement turning and milling tools for turning machines – CYCLE_982 (measuring cycles SW 5.4 and higher)

With CYCLE_982

Soft keys [Measurem. turning] → [Tool measurem.]

CYCLE982 can be parameterized for tool measurement.

Parameters

_MVAR	INTEGER	Measurement variant
_MA	INTEGER	Number of measuring axis
_STA1	REAL	Starting angle on milling tools
_CORA	REAL	Offset angular position after reversal for milling tools

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Measuring a shaft hole – CYCLE_977_979A

With CYCLE_977_979A

Soft keys [Measurem. milling] → [Tool measurem.] → [Hole Shaft]

measurement variants xxx1 and xxx2 of measuring cycles CYCLE977 and CYCLE979 can be parameterized.

Parameters

	INTEGER	Selection: Angular position 977...paraxial measurement / 979...measurement at an angle
_MVAR	INTEGER	Measurement variant
_SETVAL	REAL	Setpoint
_ID	REAL	Infeed path
_SZA	REAL	Protection zone
_TNUM	REAL	**Measurement only:** Tool number for automatic offset
_TNAME	STRING	**Alternatively, measurement only:** Tool name with active tool management
_KNUM	INTEGER	Offset number D number for measurement / ZO number for calculating zero offset
_CPA	REAL	Center 1st axis
_CPO	REAL	Center 2nd axis
_STA1	REAL	Initial angle
_INCA	REAL	Indexing angle
_PRNUM	INTEGER	Measuring probe number **CYCLE979 only:** The number of measuring points is assigned from the thousands digit; 0... 3 measuring points, 1... 4 measuring points

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Measure groove web – CYCLE_977_979B

With CYCLE_977_979B
Soft keys [Measurem. milling] → [Tool measurem.] → [Slot or web]
measurement variants xxx3 and xxx4 of measuring cycles CYCLE977 and CYCLE979 can be parameterized.

Parameters

	INTEGER	Selection: Angular position 977...paraxial measurement / 979...measurement at an angle
_MVAR	INTEGER	Measurement variant
_SETVAL	REAL	Setpoint
_ID	REAL	Infeed path
_MA	INTEGER	Number of measuring axis
_TNUM	REAL	**Measurement only:** Tool number for automatic offset
_TNAME	STRING	**Alternatively, measurement only:** Tool name with active tool management
_KNUM	INTEGER	Offset number D number for measurement / ZO number for calculating zero offset
_CPA	REAL	Center 1st axis
_CPO	REAL	Center 2nd axis
_STA1	REAL	Initial angle
_SZA	REAL	Protection range
_PRNUM	INTEGER	Probe number

Measure rectangle – CYCLE_977_979C

With CYCLE_977_979C
Soft keys [Measurem. milling] → [Tool measurem.] → [>>] → [Rectangle]
measurement variants xxx5 and xxx6 of measuring cycle CYCLE977 can be parameterized.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Parameters

_MVAR	INTEGER	Measurement variant
_SETV[0]	REAL	Setpoint length
_SETV[1]	REAL	Setpoint width
_ID	REAL	Infeed path
_SZA	REAL	Protection zone length
_SZO	REAL	Protection zone width
_TNUM	REAL	**Measurement only:** Tool number for automatic offset
_TNAME	STRING	**Alternatively, measurement only:** Tool name with active tool management
_KNUM	INTEGER	Offset number D number for measurement / ZO number for calculating zero offset

Single point measurement – CYCLE_978

With CYCLE_978

Soft keys [Measurem. milling] → [Tool measurem.] → [Plane]

CYCLE978 can be parameterized.

Parameters

_MVAR	INTEGER	Measurement variant
_SETVAL	REAL	Setpoint
_MA	INTEGER	Measuring axis
_TNUM	REAL	**Measurement only:** Tool number for automatic offset
_TNAME	STRING	**Alternatively, measurement only:** Tool name with active tool management
_KNUM	INTEGER	Offset number D number for measurement / ZO number for calculating zero offset

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Angle measurement – CYCLE_998

With CYCLE_998

Soft keys Measurem. milling → Tool measurem. → Angle

CYCLE998 can be parameterized.

Parameters

_MVAR	INTEGER	Measurement variant
_SETVAL	REAL	Setpoint
_ID	REAL	Distance
_RA	INTEGE	Number of rotary axis
_MA	INTEGER	Number of measuring axis
_KNUM	INTEGER	ZO number
_STA1	REAL	Angle
_PRNUM	INTEGER	Probe number
_MD	INTEGER	Determining the measuring direction 0...in positive direction / 1...in negative direction
_INCA	REAL	Setpoint angle about 2nd axis of the plane
_SETV[0]	REAL	Distance between measuring points P1 and P3

Corner measurement 1 – CYCLE_961_W

With CYCLE_961_W

Soft keys Measurem. milling → Tool measurem. → Corner

measurement variants 105 ... 108 for CYCLE961
can be parameterized.

Parameters

	INTEGER	Selection: Outside or inside corner 0...inside corner/1...outside corner
	INTEGER	Selection: Number of measuring points, 3 or 4
_SETV[0]	REAL	Distance between starting point and measuring point 2, without sign
_SETV[1]	REAL	Distance between starting point and measuring point 4, without sign
_ID	REAL	Retraction path in 3rd axis (applicate), for outside corner only, without sign
_STA1	REAL	Enter approximate angle between 1st axis (abscissa) and 1st edge, in clockwise direction with negative sign
_INCA	REAL	Angle between 1st and 2nd edge of workpiece, in clockwise direction with negative sign
_KNUM	INTEGER	ZO number

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

_SETV[4]	REAL	Selection: Offset 1...measured corner entered as zero point 2...measured corner is entered in 1st axis offset by the value in _SETV[2] and as a zero point 3...measured corner is entered in both axes offset and as a zero point 4...measured corner is entered in 2nd axis offset by the value in _SETV[3] and as a zero point
_SETV[2]	REAL	**For 3 measuring points only:** Offset of coordinate origin in 1st axis (abscissa)
_SETV[3]	REAL	**For 3 measuring points only:** Offset of coordinate origin in 2nd axis (ordinate)
_PRNUM	INTEGER	Probe number

Corner measurement 2 – CYCLE_961_P

With CYCLE_961_P

Soft keys [Measurem. milling] → [Tool measurem.] → [Corner]

measurement variants 117 ... 118 for CYCLE961 can be parameterized.

Parameters

	INTEGER	Selection: Outside or inside corner 0...inside corner/1...outside corner
_ID	REAL	Infeed path of measuring probe to measuring height, without sign
_SETV[0]	REAL	Starting position for measuring the 1st point in the 1st axis (abscissa)
_SETV[1]	REAL	Starting position for measuring the 1st point in the 2nd axis (ordinate)
_SETV[2]	REAL	Starting position for measuring the 2nd point in the 1st axis (abscissa)
_SETV[3]	REAL	Starting position for measuring the 2nd point in the 2nd axis (ordinate)
_SETV[4]	REAL	Starting position for measuring the 3rd point in the 1st axis (abscissa)
_SETV[5]	REAL	Starting position for measuring the 3rd point in the 2nd axis (ordinate)
_SETV[6]	REAL	Starting position for measuring the 4th point in the 1st axis (abscissa)
_SETV[7]	REAL	Starting position for measuring the 4th point in the 2nd axis (ordinate)
_KNUM	INTEGER	ZO number
_PRNUM	INTEGER	Probe number

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Single-point measurement – CYCLE_974

With CYCLE_974

Soft keys Measurem. turning → Tool measurem. → 1-point measurem.

CYCLE974 can be parameterized.

Parameters

_MVAR	INTEGER	Measurement variant
_SETVAL	REAL	Setpoint
_MA	INTEGER	Number of measuring axis
_TNUM	REAL	**Measurement only:** Tool number for automatic offset
_TNAME	STRING	**Alternatively, measurement only:** Tool name with active tool management
_KNUM	INTEGER	Offset number D number for measurement / ZO number for calculating zero offset
_PRNUM	INTEGER	Probe number
_STA1	REAL	Initial angle

Two-point measurement – CYCLE_994

With CYCLE_994

Soft keys Measurem. turning → Tool measurem. → Two-point measurem.

CYCLE994 can be parameterized.

Parameters

_MVAR	INTEGER	Measurement variant
_SETVAL	REAL	Setpoint
_MA	INTEGER	Number of measuring axis
_TNUM	REAL	**Measurement only:** Tool number for automatic offset
_TNAME	STRING	**Alternatively, measurement only:** Tool name with active tool management
_KNUM	INTEGER	Offset number D number for measurement / ZO number for calculating zero offset
_SZA	REAL	Protection zone on workpiece, 1st axis (abscissa)
_SZO	REAL	Protection zone on workpiece, 2nd axis (ordinate)
_PRNUM	INTEGER	Probe number

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Setting additional parameters – CYCLE_PARA

On measuring cycles SW 6.2 and higher, measuring cycle support no longer supports CYCLE_PARA as an autonomous cycle.
If _MZ_MASK[2]=0 is set in the GUD field, the CYCLE_PARA call will be written in the NC program in front of each measuring cycle call.

Parameters

_FA	REAL	Measuring path in mm
_VMS	REAL	Variable measuring velocity
_NMSP	INTEGER	Number of measurements at the same location
_RF	REAL	**CYCLE979 only:** Feedrate at circular-path programming
_PRNUM	INTEGER	Probe number
_CORA	REAL	**Only if monoprobe is used:** Offset angle position
_TZL	REAL	Tolerance range for zero offset
_TMV	REAL	Select range for offset with mean value calculation, greater than _TZL
_TUL	REAL	Upper tolerance range workpiece, oversize acc. to drawing
_TLL	REAL	Lower tolerance range workpiece, undersize acc. to drawing
_TSA	REAL	Safe area for measuring result
_EVNUM	INTEGER	Number of empirical value memory that is calculated
_K	REAL	Weighting factor for mean value derivation
_TDIF	REAL	Tolerance range for dimensional difference check

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

7.3 Measuring cycle support in the program editor (≥ SW 6.2)

In measuring cycles SW 6.2 and higher, the program editor provides extended measuring cycle support, for inserting measuring cycle calls into the program, for Siemens measuring cycles.

Function

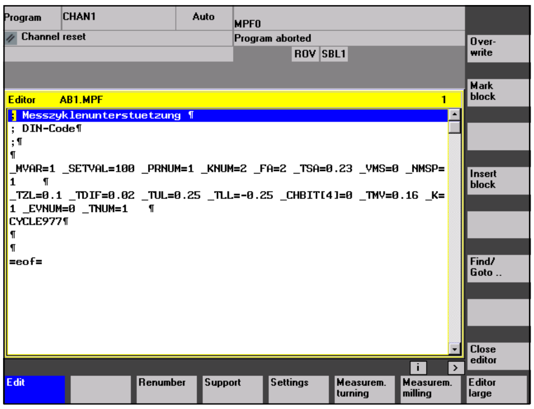
The measuring cycle support provides the following functionality:

- Measuring cycles can be selected using soft keys
- Input forms for parameter assignment with help displays
- Recompilable program code is generated from each screen form.

7.3.1 Menus, cycle explanation

Explanation

Selection of input forms for measuring cycles is technology-oriented using the horizontal soft keys 14 and 15 on the advancement menu.



Measurem. turning	Input forms for measuring cycles for turning technology
Measurem. milling	Input forms for measuring cycles for milling technology

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

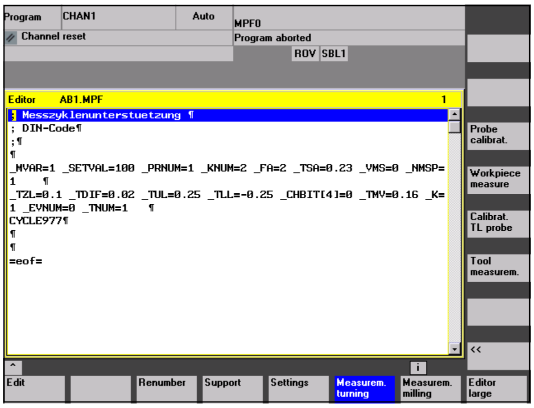
7.3.2 New functions of the input forms

Function

- With GUD field _MZ_MASK it is possible to adjust the input forms for measuring cycles to technological conditions and user requirements (see Section 7.3.3).
- In the measuring cycles, the measurement variant is controlled with parameter _MVAR. It often contains several settings encoded to form a single value. In the input forms of the new measuring cycle support, the separate settings are separated into different input fields that you can move between with the toggle key.
- The same applies to parameter _KNUM used for encoding the offset variants
- The input form changes depending on the settings of the NCK-global GUD field _MZ_MASK that is defined in module GUD6.
- The input forms change dynamically. Only those input fields that are necessary for the selected measurement variant or offset option are display, unnecessary input fields are hidden.
- If a form is displayed again, all fields will contain the values last entered as defaults.

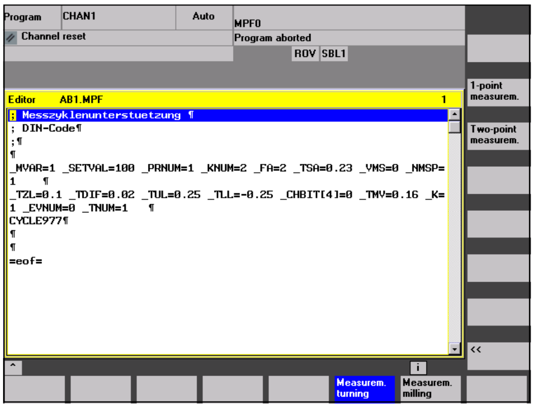
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Vertical soft key menu for turning technology



Probe calibrat.	Call screen form for CYCLE973 Calibrate workpiece probe for turning machines
Tool measurem.	Call new vertical soft key menu for "measure workpiece"
Calibrat. TL probe	Call screen form for CYCLE982 Calibrate tool probe for turning machines
Tool measurem.	Call screen form for CYCLE982 Gauge turning and milling tools for turning machines
<<	Jump back

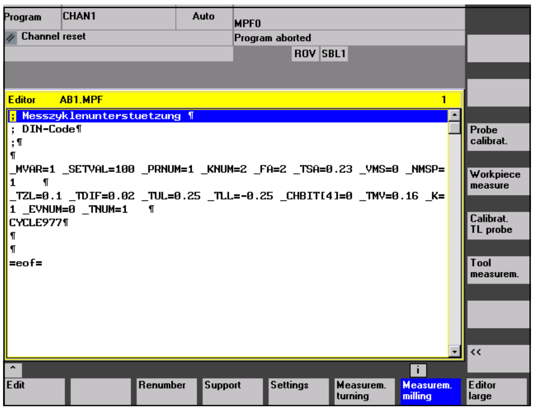
Vertical soft key menu for measure workpiece, turning



1-point measurem.	Call screen form Workpiece measurement for turning machines CYCLE974 1 point measurement
Two-point measurem.	Call screen form Workpiece measurement for turning machines CYCLE994 2 point measurement
<<	Jump back to selection menu turning

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Vertical soft key menu for milling technology



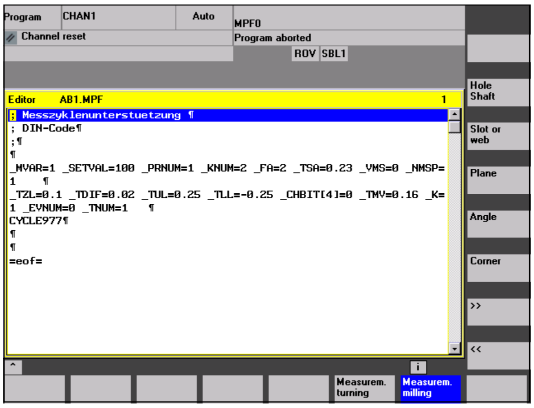
Probe calibrat.	Call screen form for CYCLE976 Calibrate workpiece probe for milling machines
Tool measurem.	Call new vertical soft key menu for selection "measure workpiece"
Calibrat. TL probe	Call screen form for CYCLE972 Calibrate tool probe for milling machines
Tool measurem.	Call screen form for CYCLE972 Gauge milling tools on milling machines
<<	Jump back

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Vertical soft key menu for workpiece measurement milling



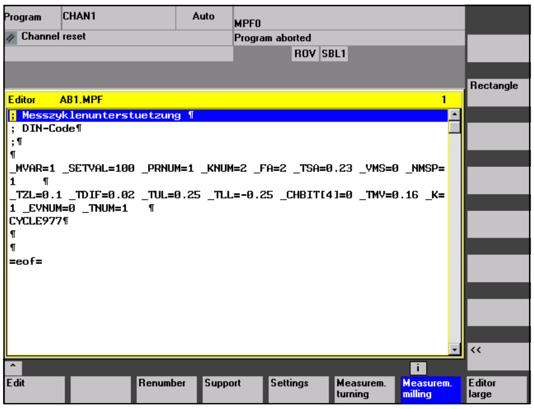
Hole Shaft	Call screen form for workpiece measurement for milling machines CYCLE974/979 drill-hole/shaft Drill-hole/shaft and paraxial/under angle switchover are performed in the screen form.
Slot or web	Call screen form for workpiece measurement for milling machines CYCLE974/979 slot/web Slot/web and paraxial/under angle switch-over are performed in the screen form.
Plane	Call screen form for workpiece measurement for milling machines CYCLE978 1 point measurement
Angle	Call screen form for zero measurement for milling machines CYCLE998 angle measurement 1 angle/2 angle switchover is performed in the screen form.
Corner	Call screen form automatic setup corner internal/external CYCLE961. Switchover between corner setup specifying distances and angle or points is performed within the form
>>	Call vertical advancement menu
<<	Jump back to selection menu milling

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Vertical advancement menu for workpiece measurement milling



Rectangle	Call screen form for workpiece measurement for milling machines CYCLE977 rectangle internal/external
<<	Jump back to selection list measure workpiece milling

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Programming example

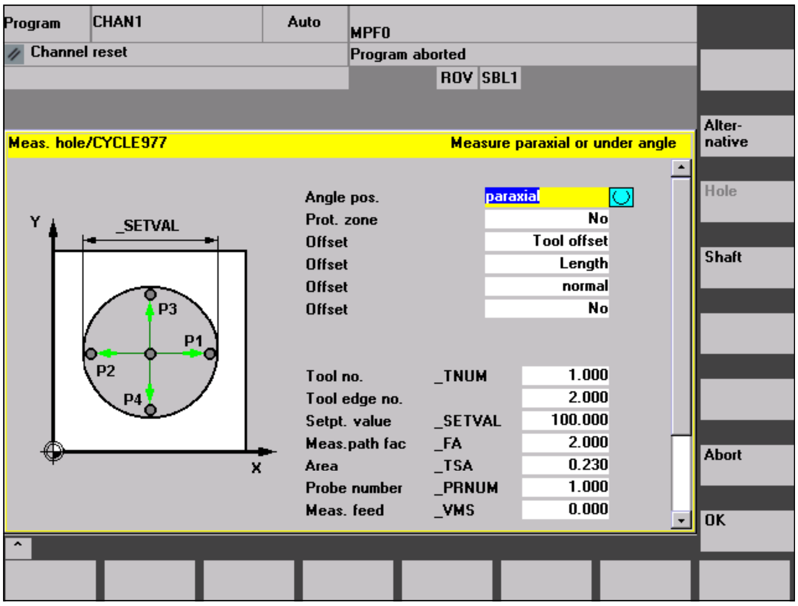
(generated with measuring cycle support)

`N100 G17 G0 G90 Z20 F2000 S500 M3`	Main block
`N110 T7 M6`	Change probes
`N120 G17 G0 G90 X50 Y50`	Position probe in X/Y plane at hole center
`N130 Z20 D1`	Position Z axis in hole
`; NC code generated by measuring cycle support` `; _MZ_MASK[0]=1`	

`N130 _MVAR=1001 _SETVAL=100.000 _PRNUM=101`
`_KNUM=2002 _FA=2 _TSA=0.23`
`_VMS=0 _NMSP=1 _ID=-20.000 _SZA=50.000`
`_CORA=0.03 _TZL=0.01 _TDIF=0.2 _TUL=0.065`
`_TLL=-0.065 _CHBIT[4]=0 _K=1 _EVNUM=2 –TNUM=1`
	Parameter passing to measuring cycle

`CYCLE977`	Call measuring cycle
`;* end of NC code generated by measuring cycle support`	

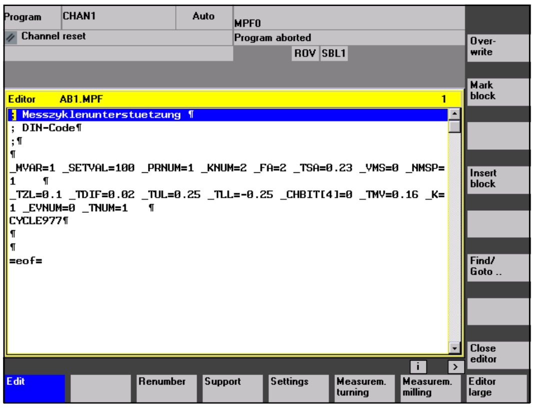
Input form for CYCLE977/979 measure drill-hole/shaft.



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

7.3.3 GUD variables for adaptation of measuring cycle support

Explanation



GUD variables to be taken into account

A field _MZ_MASK is declared in the GUD6 module in which the screen forms can be adapted:

- to the technological measurement conditions
- to the measurement variants

The settings for the field _MZ_MASK can be changed in a screen form in operating area "Setup".

Variable	*Value*	*Meaning*
_MZ_MASK[0]	0	An indirect measuring cycle call is inserted in the NC code. Example: CYCLE977/drill-hole CYCLE_PARA(...............) CYCLE_977_979A(977,........)
	1	A direct measuring cycle call is inserted in the NC code. Example: CYCLE977/drill-hole _MVAR=1 _KNUM=1 _PRNUM=1 ....CYCLE977
_MZ_MASK[1]	0	The workpiece screen forms contain the following selection options for zero offset and tool offset: Zero offset - default: • Settable zero offsets • Last channel-specific basic frame Tool offset – default: • Milling: Tool radius is corrected • Turning: Length offset in the measuring axis
	1	The workpiece screen forms contain the following selection options for zero offset and tool offset: Zero offset – extended: • Settable zero offsets • Last channel-specific basic frame • Offset in system frame • Offset in active frame • Offset in any basic frame (global or channel-spec.) Tool offset – extended: • Offset radius, length, or length selection (L1, L2, or L3) • Calculation of measurement results normal or inverted • Offset in setup/additive offset

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

_MZ_MASK[2]	0	Forms without input fields for parameters: • _VMS: Measuring velocity • _NMSP: Number of measurements at the same location The following default values are entered in the NC code for the parameters: • _VMS=0 corresponds to 150 mm/min or 5.9055 inch/min • _NMSP=1 number of measurements = 1
	1	Forms with parameter input for: • _VMS: Measuring velocity • _NMSP: Number of measurements at the same location
_MZ_MASK[3]	0	Screen forms for workpiece measurement with automatic tool offset and tool measurement do not contain an input field for the following parameters: • _EVNUM: Number of empirical value memory The following default value is entered in the NC code: • _EVNUM=0 No empirical value memory is taken into account.
	1	Screen forms for workpiece measurement with automatic tool offset and tool measurement contain input fields for the following parameters: • _EVNUM: Number of empirical value memory
_MZ_MASK[4]	0	Screen forms without input fields for the following parameters for mean value calculation with automatic tool offset: • _TMV: Select range for offset with mean value calculation • _K: Weighting factor for mean value derivation • EVNUM: Mean value memory number The following default values are entered in the NC code: • _TMV=ABS(_TUL-_TLL)/3 • _K=1 • _EVNUM=0 • _CHBIT[4]=0
	1	Screen forms contain input fields for the following parameters for mean value calculation with automatic tool offset: • _TMV: Select range for offset with mean value calculation • _K: Weighting factor for mean value derivation • _EVNUM: Mean value memory number • _CHBIT[4]=1
_MZ_MASK[5]	0	Probe type for workpiece measurement is a multiprobe
	1	Probe type for workpiece measurement is a monoprobe The relevant screen forms show an input field for the offset angle _CORA.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

`_MZ_MASK[6]`	0	The generated NC code does not contain a call for logging the measurement results.
	1	The generated NC code contains a call for logging the measurement results.
`_MZ_MASK[7]`	0	Screen form for CYCLE971 – tool measurement/milling does not contain input fields for feedrate and spindle speed. F and S are calculated within the cycle.
	1	Screen form for CYCLE971 – tool measurement/milling contains input fields for feedrate and spindle speed.

References: /BEM/, Operator's Guide HMI Embedded
/IAM/, Installation Guide HMI/MMC
IM2 "Installation HMI Embedded"

Recompilation

Recompilation of programs allows you to change existing programs using the cycle support.
When recompiling measuring cycle calls, please note that a field of defaults for programming is active (_MZ_MASK) in addition to the screen forms. If there has been a change in this settings between program creation and recompilation, the changes will also be included in the program.

Programs with measuring cycle calls cannot be recompiled after a change in the type of tool programming, i.e. change in the machine data setting

- MD 18102: MM_TYPE_OF_CUTTING_EDGE
- MD 18080: MM_TOOL_MANAGEMENT_MASK.

■

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Notes

Part 2: Description of Functions

Hardware, Software and Installation

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

8.1 Overview

Function

You can use measuring cycles for automatic measuring on CNC machines with SINUMERIK 840D and 810D controls.

For this, it is necessary to connect a touch-trigger probe to the control.

The measuring cycles and data blocks you require are loaded in the control via the RS-232-C interface.

You must adapt the measuring cycle data to the specific requirements of the individual machine, as well as assign initial values.

Measuring cycle Version 5.3 and higher also contains the package "Measurement in JOG" which permits semiautomatic "workpiece setup" and "tool measurement" in setup mode on milling machines.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

8.2 Hardware requirements

8.2.1 General hardware requirements

Axis assignment
For correct execution of the measuring cycles the machine axes must be assigned according to DIN 66217.

Applicable probes
See description in Section 4.1.

8.2.2 Probe connection

Explanation
The measuring cycles can also be used for SINUMERIK 840D and 810D. They operate with a touch-trigger probe which must be connected to the control.

Connection on 840D, 810D
On the SINUMERIK 840D and 810D, the probe is connected via the I/O interface X121 which is located on the front panel of the NCU module.

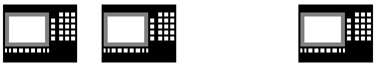
8.2.3 Measuring in JOG

Explanation

Measuring in JOG, available from measuring cycle 5.3 and higher can only be operated with

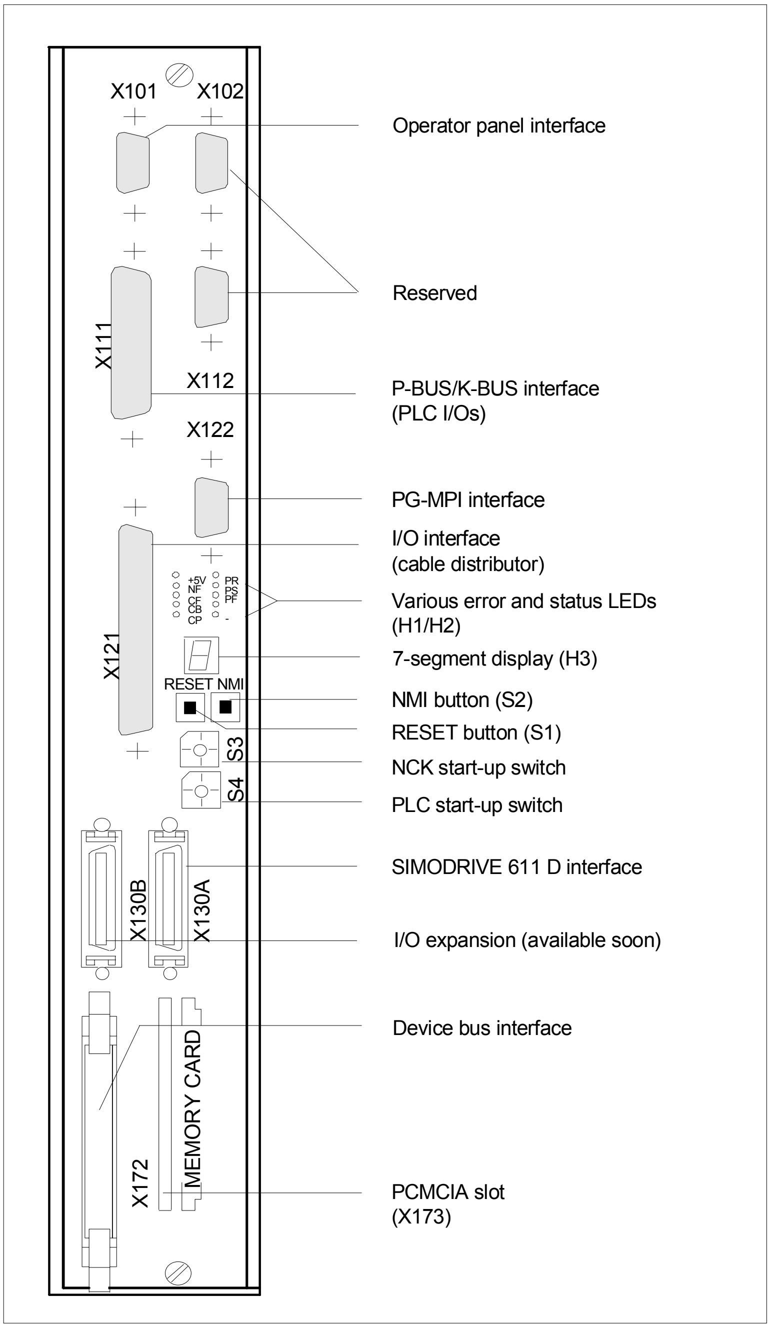
- SINUMERIK 840D with at least NCU 572
- SINUMERIK 810D
- MMC 103

8.2 Hardware requirements



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Interfaces, operator and display elements on the NCU module



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Explanation

Interface

- I/O interface
  37-pin subminiature D connector (X121),
  maximum 2 measuring probes can be connected;

  The 24 V external power supply for the binary inputs is also located on this connector.

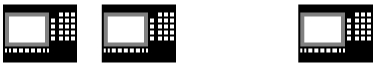
  Excerpt from PIN assignment table for front panel connector X121:

PIN		*Designation*
1	M24EXT	External power supply External ground
2	M24EXT	External ground
...	...	...
9	MEPUS 0	Connection probe 1 Measuring pulse signal input
10	MEPUC 0	Measuring pulse common input
...	...	...
20	P24EXT	External power supply External P24 V
21	P24EXT	External P24 V
...	...	...
28	MEPUS 1	Connection for probe 2 Measuring pulse signal input
29	MEPUC 1	Measuring pulse common input
...	...	...

For more detailed information and a description of the interfaces (e.g. pin assignment) please refer to the

References: /PHD/, Hardware Configuring Guide

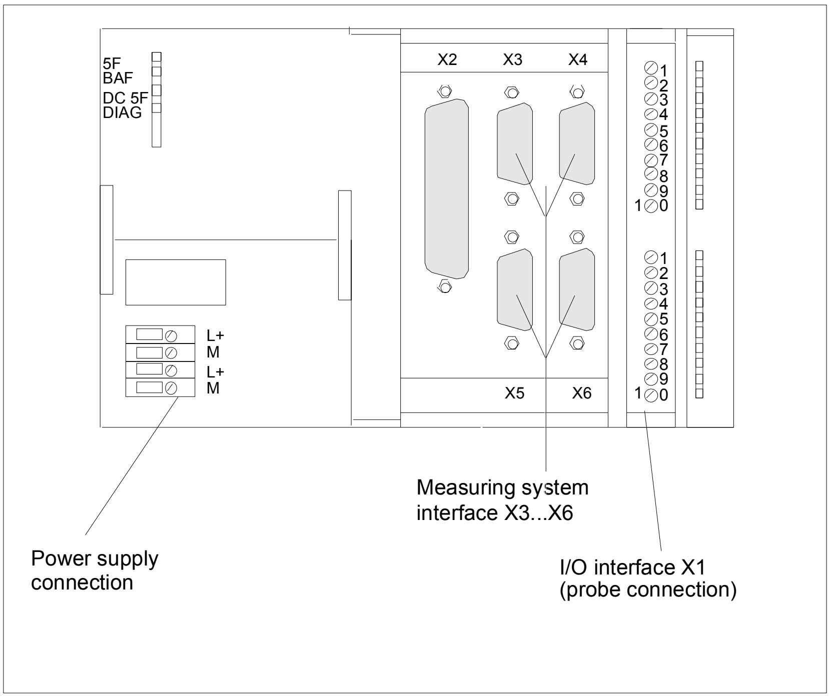
The same cable distributor is used as for SINUMERIK 840C.



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Connection to FM-NC NCU 570.2

The following figure shows the FM-NC interface for connecting the probe.



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

Explanation

Interface

- I/O interface
  - 20-way front connector (X1) for connecting the handwheels (maximum 2),
  - of the fast inputs, including probes, and
  - for wiring the NC-READY relay.

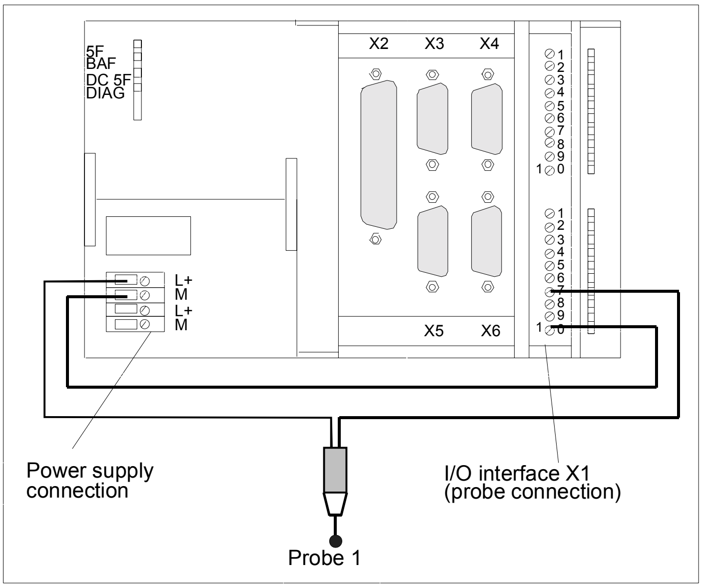
Excerpt from PIN assignment table for front panel connector X1:

PIN	MD 30120 CTRLOUT_NR	Designation
X1: Handwheel and I/O connection, 20-pin front panel connector		
...	...	...
17	-	Digit. input 3/measuring pulse input 1 (DE3/MEPU1)
18	-	Digit. input 3/measuring pulse input 2 (DE3/MEPU2)
20	-	M24EXT external ground

- Power supply connection
  4-pin screw terminal block (X10) for connecting the 24 V power supply

For more information please refer to the "SINUMERIK FM-NC Installation and Start-up Guide".

Example for connecting the probe to the FM-NC (NCU 570.2), probe 1



840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

8.3 Software requirements

Explanation

How the measuring cycles are supplied
The measuring cycles and data blocks you require for data definition are supplied on diskette in MS-DOS format.

The measuring cycles are read into the program memory of the control in the standard cycles directory via the RS-232-C interface.

8.3.1 General measuring cycles

Explanation

NC SW version
For correct execution of the measuring cycles, NC SW 3.2 or higher is required.

MMC SW
The functions measuring results display screen and parameter assignment via input dialog require MMC SW 3.2 or higher.

PLC program
The measuring cycles execute with the basic PLC program, it is not necessary to adapt them to the PLC user program. The measuring function is activated in the measuring cycles via the MEAS command.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

8.3.2 Measuring in JOG

Explanation

Options

"Measurement in JOG" can only be used if the "Inter-modal actions" option (ASUPs and synchronized actions in all modes) is active.

Measuring cycle version

For measuring in JOG, measuring cycle SW 5.3 or higher is required. To ensure correction functioning of measuring in JOG, a minimum installation of the following definition files, measuring and auxiliary cycles is required:

GUD5.DEF
GUD6.DEF in directory DEFINE on diskette1

CYCLE107.SPF
CYCLE108.SPF
CYCLE109.SPF
CYCLE110.SPF
CYCLE111.SPF
CYCLE114.SPF
CYCLE198.SPF
CYCLE199.SPF
CYCLE961.SPF
CYCLE971.SPF
CYCLE976.SPF
CYCLE977.SPF
CYCLE978.SPF is located in directory CYCLES on diskette 1

NC SW version

For correct operation of measuring in JOG, NC SW 5.3 and higher (810D SW 3.3 and higher) is required.

MMC SW

MMC SW 5.3 and higher is required for measuring in JOG.

PLC program

Measuring in JOG runs with the PLC basic program. No adaptations in the PLC user program are necessary.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

8.4 Function check

Function

Measurement command
The control has the command MEAS for generating a measuring block.

The measuring input number is set in the command parameters.

References: /PAZ/, Programming Guide

Measuring results
The results of the measurement command are stored in the system data of the NCK and can be accessed from the program.

These are:

$AC_MEA[<No.>]	Software switching signal for the probe No. stands for measuring input number
$AA_MW[<Axis>]	Measured value of the axis in workpiece coordinates Axis stands for the name of the measuring axis
$AA_MM[<Axis>]	Measured value of the axis in machine coordinates

References: /PAZ/, Programming Guide

PLC service display
The functional check of the probe is conducted via an NC program.

The measuring signal can be controlled via the diagnostics menu "PLC Status".

Status display for measuring signal

Probe 1 deflected	DB10	DB X107.0
Probe 2 deflected	DB10	DB X107.1

840 D
NCU 571

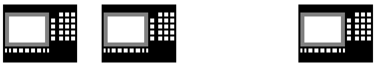
840 D
NCU 572
NCU 573

810 D

Example of functional check

%_N_CHECK_PROBE_MPF

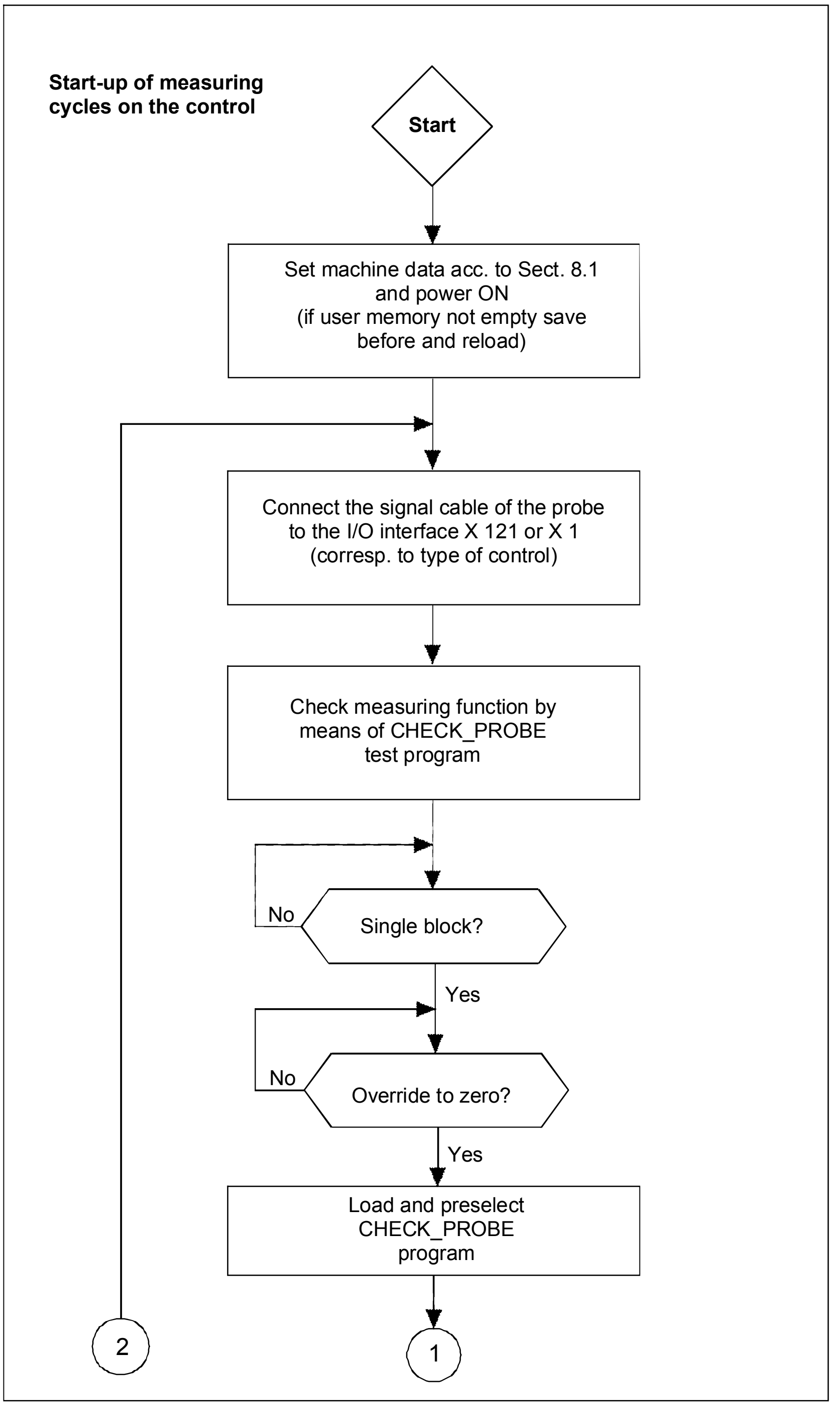
;$PATH=/_N_MPF_DIR		
;Test program for connecting the probe		
N05	DEF INT MTSIGNAL	;Flag for signal status
N10	DEF INT ME_NR=1	;Measuring input number
N20	DEF REAL MEAS.VALUE_IN_X	
N30	G17 T1 D1	;Preselect tool offset ;for probe
N40	_ANF: G0 G90 X0 F150	;Start position and measuring velocity
N50	MEAS=ME_NR G1 X100	;Measurement at measuring input 1 ;in the X axis
N60	STOPRE	
N70	MTSIGNAL=$AC_MEA[1]	;Read software switching signal ;at 1st measuring input
N80	IF MTSIGNAL == 0 GOTOF _ERR1	;Evaluation of signal
N90	MEAS.VALUE_IN_X=$AA_MW[X]	;Read in measured value in ;workpiece coordinate
N95	M0	
N100	M02	
N110	_ERR1: MSG ("Probe is not operating!")	
N120	M0	
N130	M02	

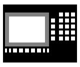
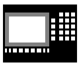
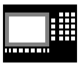


840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

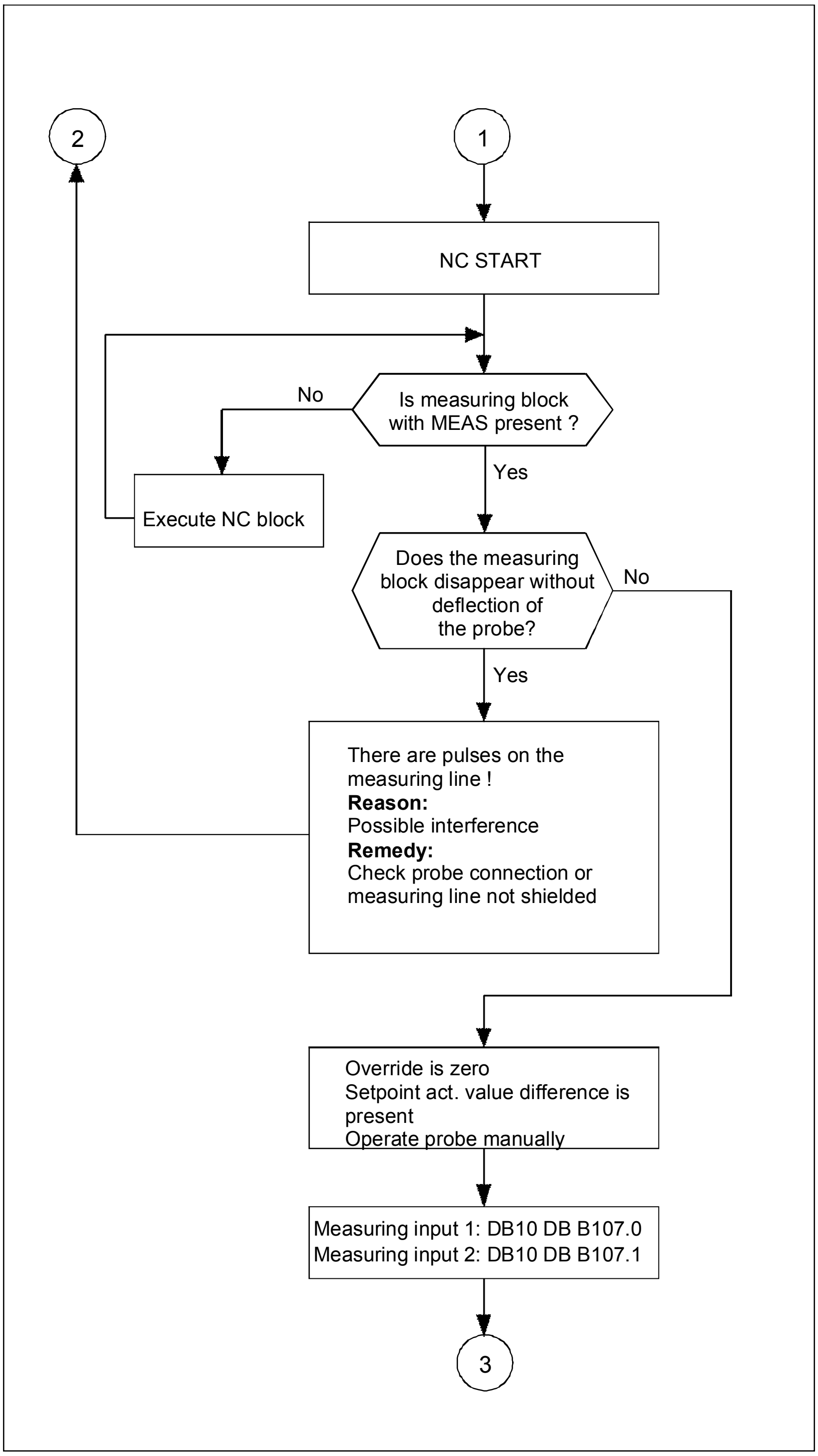
8.5 Start-up sequences

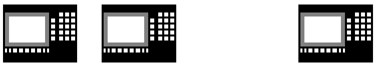
8.5.1 Start-up flowchart for measuring cycles and probe circuit



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

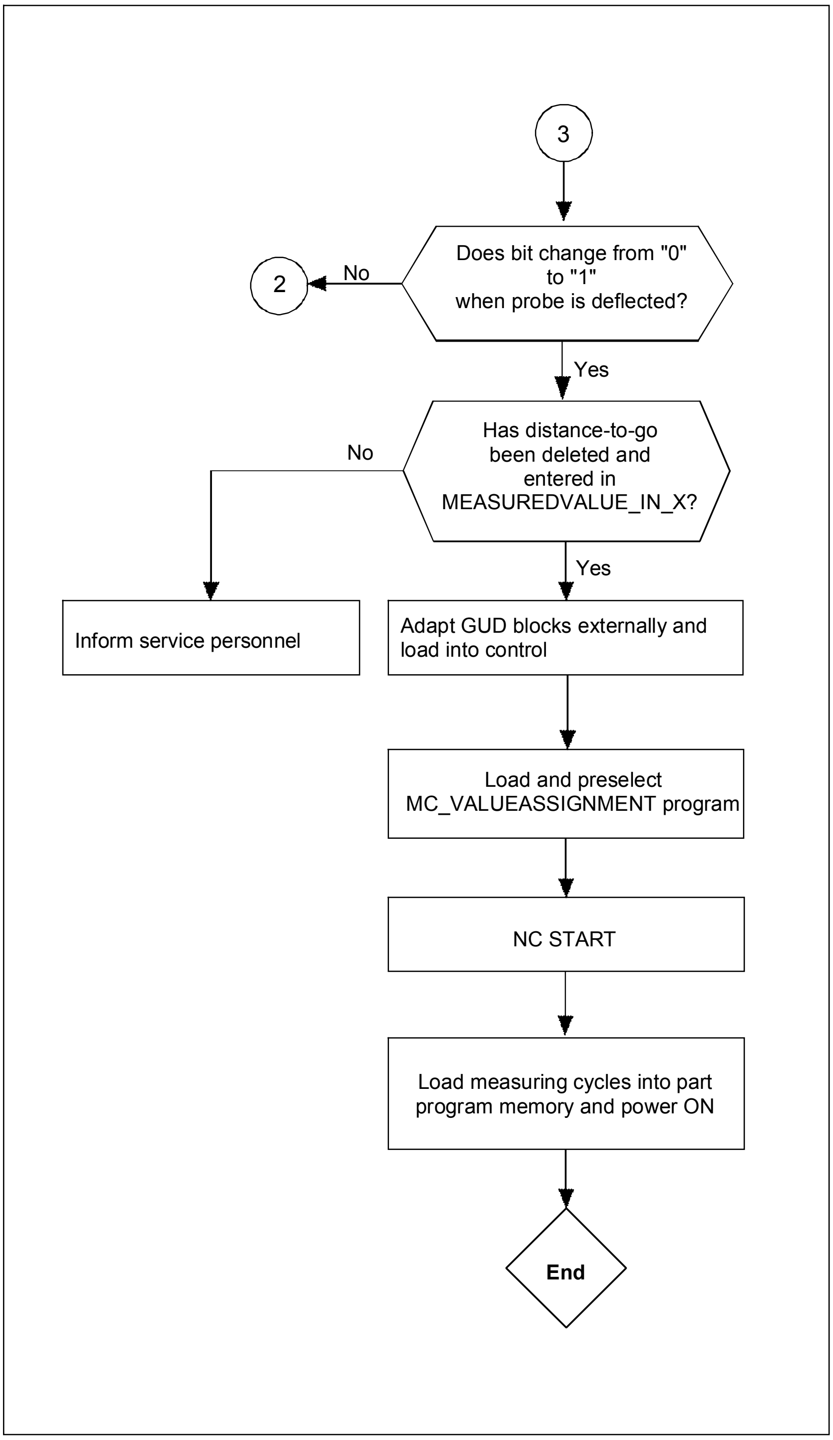




840 D
NCU 571

840 D
NCU 572
NCU 573

810 D



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D

8.5.2 Starting up the measuring cycle interface for the MMC 102

Function

In SW 3.2 and higher, the measuring cycles offer the option of displaying the measurement result screens and setting the input parameters via a dialog (call CYCLE103). These functions require adaptations in the MMC software on the control.

Explanation

MMC 102
In the "Start-up" operating area you can access the MMC file system via the softkeys "MMC" and "DOS-Shell".

In the file
c:\mmc2\comic.nsk
the comment has to be removed in the second line.

REM TOPIC(... ⇒ **TOPIC**(...

Then the MMC has to be started again.

Testing the measuring cycle interface
The cycle CYCLE103 can be activated and run in automatic mode.

When functioning properly, a screen is displayed with an overview of the measuring cycles; the dialog box for setting the measuring parameter cycles can be opened from here.

■

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

Notes

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Supplementary Conditions

There are no special conditions for the measuring cycles. However, the following memory capacity requirements should be taken into account.

Memory requirement

The measuring cycles require the following memory capacity in the NC program memory of the control:

	Memory requirements [in KB]
Full number of measuring cycles	approx. 190
Measuring cycles for milling machines	approx. 150
Measuring cycles for turning machines	approx. 120

■

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Notes

Data Description

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

10.1 Machine data for machine cycle runs

Function

Memory-configuring machine data

As the measuring cycles have to run with GUD and LUD variables (**G**lobal **U**ser **D**ata and **L**ocal **U**ser **D**ata), the following minimum settings must be made in the data for the memory configuration:

18118 MD number	**MM_NUM_GUD_MODULES** Number of data blocks	
Default setting: 7 **When using measuring cycles: 7**	min. input value: 1	max. input value: 9
Changes are validated by Power ON	Protection level: 2/7	Unit: -
Data type: DWORD	valid as of software version: SW 2	
Meaning:	Number of GUD files in the active file system (SRAM)	

18120 MD number	**MM_NUM_GUD_NAMES_NCK** Number of GUD variables in PLC	
Default setting: 10 **When using measuring cycles: 20**	min. input value: 0	max. input value: 400
Changes are validated by Power ON	Protection level: 2/7	Unit: -
Data type: DWORD	valid as of software version: SW 1	
Meaning:	Number of global user variables (SRAM)	

18130 MD number	**MM_NUM_GUD_NAMES_CHAN** Number of GUD variables per channel	
Default setting: 10 **When using measuring cycles: 100**	min. input value: 0	max. input value: 200
Changes are validated by Power ON	Protection level: 2/7	Unit: -
Data type: DWORD	valid as of software version: SW 1	
Meaning:	Number of channel-specific user variables (SRAM)	

18150 MD number	**MM_GUD_VALUES_MEM** Memory for values of the GUD variables	
Default setting: 12 **When using measuring cycles: 60**	min. input value: 0	max. input value: 50
Changes are validated by Power ON	Protection level: 2/7	Unit: KB
Data type: DWORD	valid as of software version: SW 1	
Meaning:	Memory for user variables (SRAM)	

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

18170 MD number	**MM_NUM_MAX_FUNC_NAMES** Number of special functions (cycles, DRAM)	
Default setting: 40 **When using measuring cycles: 70**	min. input value: 0.0	max. input value: plus
Changes are validated by Power ON	Protection level: 2/7	Unit: -
Data type: DWORD	valid as of software version: SW 1	
Meaning:	Number of cycles with input parameters	

18180 MD number	**MM_NUM_MAX_FUNC_PARAM** Number of special functions (cycles, DRAM)	
Default setting: 300 **When using measuring cycles: 600**	min. input value: 0.0	max. input value: plus
Changes are validated by Power ON	Protection level: 2/7	Unit: -
Data type: DWORD	valid as of software version: SW 1	
Meaning:	Number of additional parameters for cycles acc. to MD 18170	

28020 MD number	**MM_NUM_LUD_NAMES_TOTAL** Number of LUD variables in total (in all program levels)	
Default setting: 200 **When using measuring cycles: 200**	min. input value: 0	max. input value: 300
Changes are validated by Power ON	Protection level: 2/7	Unit: -
Data type: DWORD	valid as of software version: SW 3.2	
Meaning:	Number of local user variables (DRAM)	

These machine data are for configuring the supported memory area of the PLC.

Therefore, make sure that they **are set before initiating start-up**.

Otherwise, all data from the user program (NC program memory including cycles, tool offsets and R parameters) have to be backed up and read back in again.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Function

Machine data for adapting the probe

13200 MD number	**MEAS_PROBE_LOW_ACTIVE** Switching performance of the probe			
Default setting: 0 **When using measuring cycles: 0**	min. input value: FALSE		max. input value: TRUE	
Changes are validated by Power ON		Protection level: 2/7		Unit: -
Data type: BOOLEAN			valid as of software version: SW 2.2	
Meaning:	Value 0: (Default setting) Non-deflected state 0 V Deflected state 24 V Value 1: Non-deflected state 24 V Deflected state 0 V			

Machine data for adapting MMC commands in cycles

10132 MD number	**MMC_CMD_TIMEOUT** Monitoring time for MMC command in part program			
Default setting: 1 **When using measuring cycles: 3**	min. input value: 1		max. input value: 100	
Changes are validated by Power ON		Protection level: 2/7		Unit: s
Data type: DOUBLE			valid as of software version: SW 3.2	
Meaning:	Monitors the time until the MMC acknowledges a command from the part program.			

Machine data for logging

11420 MD number	**LEN_PROTOCOL_FILE** File size for log files (KB)			
Default setting: 1 **When using measuring cycles: 5**	min. input value: 1		max. input value: 1000	
Changes are validated by Power ON		Protection level: 0/0		Unit: -
Data type: DWORD			valid as of software version: SW 4.3	
Meaning:	Size for log file			

Machine data for configuring channel-specific system frames

28082 MD number	**MM_SYSTEM_FRAME_MASK** Configuration screen form for channel-specific system frames			
Default setting: 21hex **When using measuring cycles: Bit 0=1;** **at least Bit 5=1**	min. input value: 0		max. input value: 7Fhex	
Changes are validated by Power ON		Protection level: 2/7		Unit: -
Data type: INT			valid for measuring cycles SW 6.2. and higher	
Meaning:	Bit 0: 1 System frames for setting actual values and scratching Bit 5: 1 System frames for cycles			

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

10.2 Cycle data

10.2.1 Data concept for measuring cycles

Function

Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be suitably adapted to the problem at hand by means of parameter settings. They can be adapted for this purpose by means of so-called **defining parameters**.

They also return data such as measurement results. They are stored in **result parameters**.

Furthermore, the measuring cycles also require **internal parameters** for calculations.

Defining parameters

The defining parameters of the measuring cycles are defined as GUD variables.

They are stored in the nonvolatile memory area of the control, their setting values remain stored even when the control is switched off and on.

These data are contained in the data definition blocks
- GUD5.DEF and
- GUD6.DEF

which are supplied together with the measuring cycles.

These blocks must be loaded into the control during start-up. They must then be adapted according to the characteristics of the relevant machine by the machine manufacturer.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

The value for the defining parameters of the measuring cycles in module GUD5.DEF can be assigned in the program before the cycle is called; this is achieved by operator input or by starting CYCLE103, which controls an interactive dialog.

The data in the operating area "Parameters", "User data" can be selected via "Global user data" or "Channel-specific user data".

Result parameters

The results are also stored as GUD variables in the GUD5 module.

Internal parameters

LUD variables are used in the measuring cycles as internal arithmetic parameters. These are set up in the cycle and thus exist only for the duration of the run-time.

10.2.2 Data blocks for measuring cycles: GUD5.DEF and GUD6.DEF

Function

The measuring cycle data are stored in two separate definition blocks:

- GUD5.DEF Data module for measuring cycle operators
- GUD6.DEF Data module for machine manufacturers

The sizes of the fields for the empirical and mean values must also be configured by the machine manufacturer at measuring cycle start-up.
The sizes of the fields for the empirical and mean values must also be configured by the machine manufacturer at measuring cycle start-up. The preset values, however, are defined by the measuring cycle operator.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Excerpt from GUD5.DEF

For adapting the GUD5.DEF module, only the following section is relevant:

(An example is provided in Chapter 11).

```
%_N_GUD5_DEF
;$PATH=/_N_DEF_DIR
;<Version> , <Date>
...
N40 DEF CHAN REAL _EV[20]
N50 DEF CHAN REAL _MV[20]
...
N99 M02
```

Module GUD6.DEF

The general measuring cycle data are configured in the GUD6.DEF data module.

This module is supplied with the measuring cycles in its standard configuration and must be adapted to the specific requirements of the machine by the machine manufacturer.

(An example is provided in Chapter 11).

Contents of GUD6.DEF

This block is supplied with the measuring cycles, with the following contents for example (see also example in Chapter 11):

```
%_N_GUD6_DEF
;$PATH=/_N_DEF_DIR
;V05.04.06 , 14.12.2001
...
N10 DEF NCK INT _CVAL[4]=(3,3,3,0)
N11 DEF NCK REAL _TP[3,10]=(0,0,0,0,0,0,0,133,0,2)
N12 DEF NCK REAL _WP[3,11]
N13 DEF NCK REAL _KB[3,7]
N14 DEF NCK REAL _CM[8]=(60,2000,1,0.005,20,4,10,0)
N15 DEF NCK REAL _MFS[6]
N20 DEF NCK BOOL _CBIT[16]=(0,0,0,1,0,0,0,0,1,0,0,0,0,0,0,0)
N30 DEF NCK STRING[8] _SI[3]=("”,”5”,"”)
N40 DEF CHAN INT _EVMVNUM[2]=(20,20)
N41 DEF CHAN REAL _SPEED[4]=(50,1000,1000,900)
```

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

```
N50 DEF CHAN BOOL _CHBIT[20]=(0,1,1,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0)
N60 DEF NCK STRING[32] _PROTNAME[2]
N61 DEF NCK STRING[80] _HEADLINE[10]
N62 DEF NCK INT _PROTFORM[6]=SET(60,80,1,5,1,12)
N63 DEF NCK CHAR _PROTSYM[2]
N64 DEF NCK STRING[100] _PROTVAL[13]
N65 DEF NCK INT _PMI[4]
N66 DEF NCK INT _SP_B[20]
N67 DEF NCK STRING[12] _TXT[100]
N68 DEF NCK INT _DIGIT=3
...
N92 DEF CHAN INT _JM_I[5]=SET(0,1,1,17,0)
N93 DEF CHAN BOOL _JM_B[7]=SET(0,1,0,0,0,0,0)
M17
```

In the delivery status, the following settings are active:

- Number of data fields (N01), 3 data fields each for
  - tool probe (N11),
  - workpiece probe (N12)
  - calibrating piece (N13);
- Monitoring data for tool measurement with rotating spindle and cyclic calculation (N14):
  - max. grinding wheel surface speed 60 m/min,
  - max. speed 2000 rpm,
  - F_{min}=1 mm/min,
  - measuring accuracy 0.005 mm,
  - F_{max} for probing 20 mm/min,
  - direction of rotation M4,
  - double probing with feedrate factor 10 for first probing;
- Central bits (N20)
  - no measurement repetition or exceeding of dimensional difference and safe area,
  - no M0 for measurement repetition
  - no M0 for "Oversize", "Undersize", "Dimensional difference",
  - metric basic system,
  - tool measurement and calibration with CYCLE982 performed in the basic coordinate system (machine coordinate system with kinematic transformation switched off)
  - correction of monoprobe position with _CORA,
  - use of the standard log header,
  - cycle-internal calculation of speed and feedrate in tool measurement with rotating spindle,
  - length of the workpiece probe with reference to probe tip;

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

- Software status of the control – SW 5
- Channel-oriented values (N40)
  - 20 memories for empirical and mean values;
- Channel-oriented values (N41)
  - 50% rapid traverse velocity,
  - positioning feed in the plane 1000 mm/min
  - positioning feed in the infeed axis 900 mm/min;
- Channel-oriented bits (N50)
  - measurement input 1 for connecting a workpiece probe,
  - measurement input 2 for connecting a tool probe,
  - collision detection active in motion blocks generated by measurement cycles
  - entry of the tool data for tool measurement
    in the geo memory,
  - no mean value storage,
  - empirical value is subtracted from measured actual value,
  - in workpiece measurement with automatic TO
    additive offset is implemented in the wear memory
  - no measurement result screen display,
  - no coupling of spindle position with coordinate
    rotation in the plane,
  - max. 5 measurement attempts,
  - retraction from the measuring point at the same velocity
    as for intermediate positioning,
  - measuring feed on defined by _VMS;
- Central values for logging (N62)
  - 60 lines per page,
  - 80 characters per line,
  - start page number is 1,
  - number of header lines is 5,
  - number of value lines in log is 1,
  - number of characters per column is 12,
- Central values for logging (N68)
  - number of decimal places is 3
- Channel-oriented values for measurement in JOG (N92)
  - no entry of data field number for probe like in ShopMill
  - number of the data field for the workpiece probe is 1
  - number of the data field for the tool probe is 1
  - working plane for measurement in JOG is G17
  - active ZO number for measurement in JOG is 0 (G500)
- Channel-oriented bits for measurement in JOG (N93)
  - offset in geometry for tool measurement
  - 1 measurement attempt
  - retraction from the measuring point at the same velocity as
    for intermediate positioning
  - no fast measuring feed

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

10.2.3 Central values

	_CVAL Number of elements		
	min. input value: -	max. input value: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: INTEGER	valid as of software version: SW 3.2		
Meaning:			Default setting
	_CVAL[0]	Number of tool probes	3
	_CVAL[1]	Number of workpiece probes	3
	_CVAL[2]	Number of calibration probes	3
	_CVAL[3]	*Not currently assigned*	0

	_TP Tool probe		
	min. input value: -	max. input value: -	
Changes valid after value assignment	Protection level: 2/7	Unit: -	
Data type: REAL	valid as of software version: SW 3.2		
Meaning:			Default setting
	Index "x" stands for the number of the current probe - 1		
	Assignment for milling		
	_TP[x,0]	Trigger point in minus direction **X** (1st geometry axis)	0
	_TP[x,1]	Trigger point in plus direction **X** (1st geometry axis)	0
	_TP[x,2]	Trigger point in minus direction **Y** (2nd geometry axis)	0
	_TP[x,3]	Trigger point in plus direction **Y** (2nd geometry axis)	0
	_TP[x,4]	Trigger point in minus direction **Z** (3rd geometry axis)	0
	_TP[x,5]	Trigger point in plus direction **Z** (3rd geometry axis)	0
	_TP[x,6]	Edge length/disk diameter	0
	_TP[x,7]	Internal assignment	133
	_TP[x,8)	Probe type 0: Cube 101: Wheel in XY 201: Wheel in ZX 301: Wheel in YZ	0
	_TP[x,9]	Distance between upper edge of tool probe and lower edge of tool	2
	Assignment for turning		
	_TP[x,0]	Trigger point in minus direction of abscissa	0
	_TP[x,1]	Trigger point in plus direction of abscissa	0
	_TP[x,2]	Trigger point in minus direction of ordinate	0
	_TP[x,3]	Trigger point in plus direction of ordinate	0
	_TP[x,4] to	Without meaning	0
	_TP[x,9]	Without meaning	0

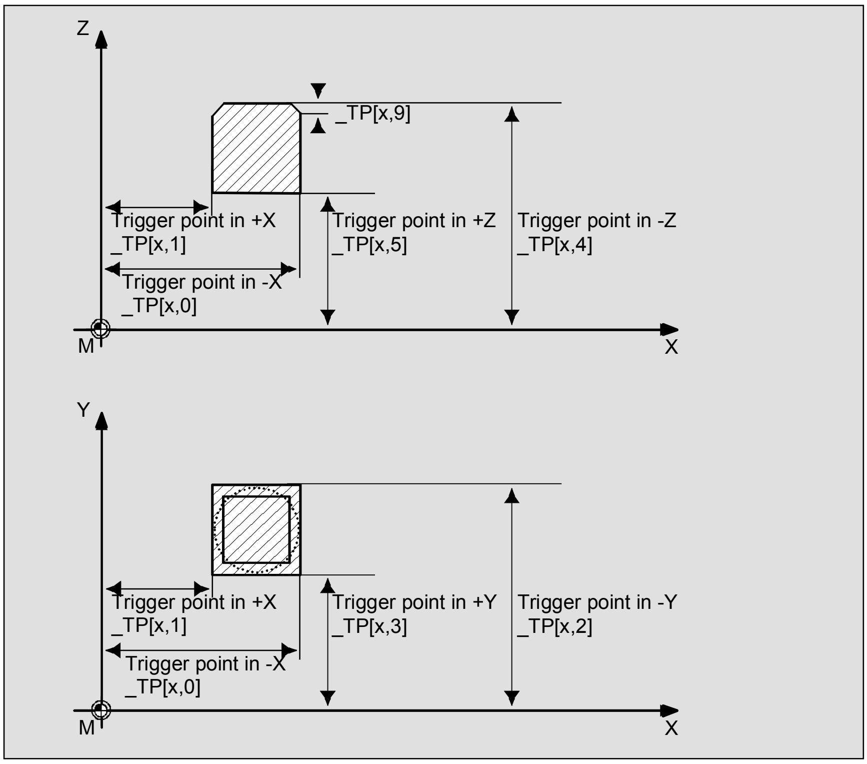
840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

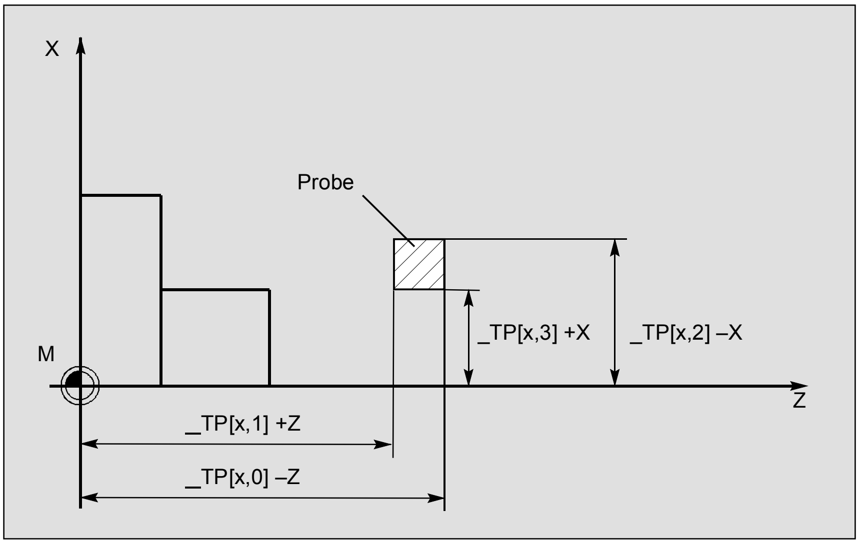
840 Di

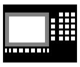
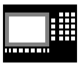
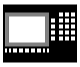
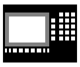
Tool probe on milling machine



Tool probe on turning machine

The representation refers to the working plane defined by G18.



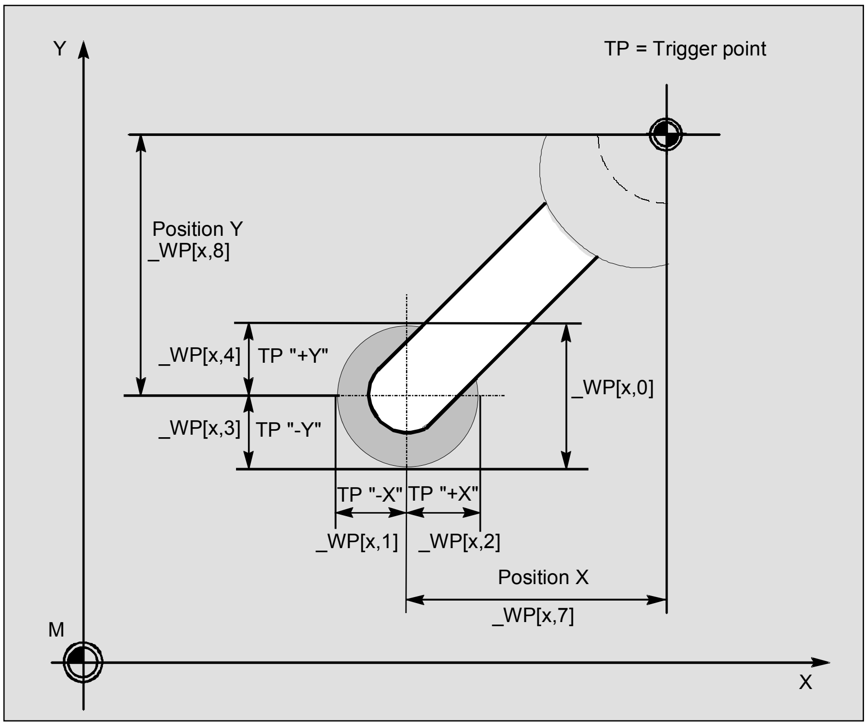
   

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

	_WP Workpiece probe		
	min. input value: -	max. input value: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: REAL	valid as of software version: 2/7		
Meaning:	Index "x" stands for the number of the current probe - 1		Default setting
	_WP[x,0]	Ball diameter of workpiece probe	0
	_WP[x,1]	Trigger point in minus direction of the abscissa	0
	_WP[x,2]	Trigger point in plus direction of the abscissa	0
	_WP[x,3]	Trigger point in minus direction of the ordinate	0
	_WP[x,4]	Trigger point in plus direction of the ordinate	0
	_WP[x,5]	Trigger point in minus direction of the applicate	0
	_WP[x,6]	Trigger point in plus direction of the applicate	0
	_WP[x,7]	Position of abscissa (deviation)	0
	_WP[x,8]	Position of ordinate (deviation)	0
	_WP[x,9]	Internal value	0
	_WP[x,10]	Internal value	0

Overview of probe data

The representation refers to the working plane defined by G17.

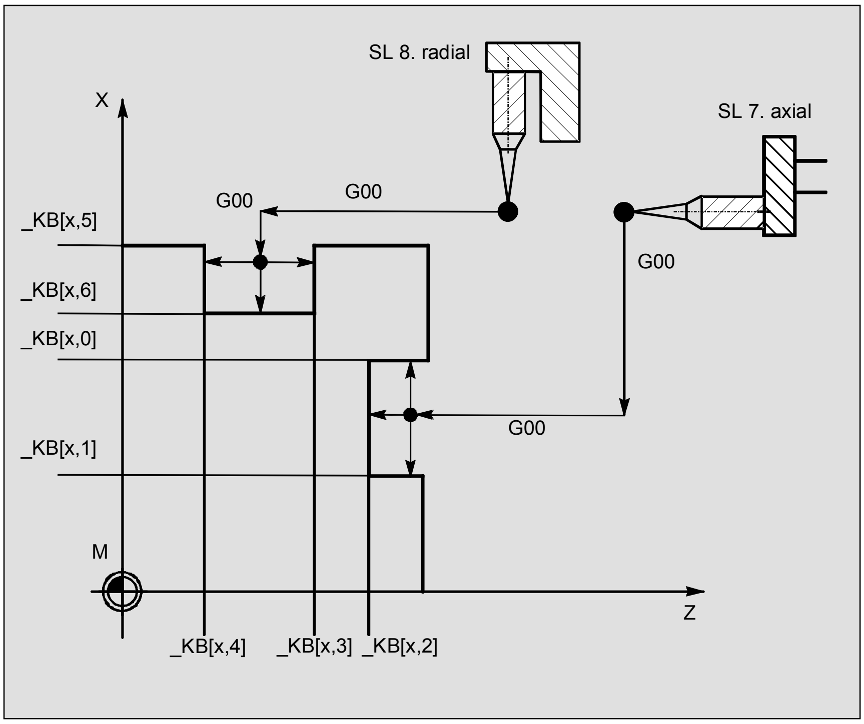


840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

	_KB Calibration block		
		min. input value: -	max. input value: -
Changes valid after value assignment		Protection level: -	Unit: -
Data type: REAL		valid as of software version: SW 3.2	
Meaning:	Index "x" stands for the number of the current calibration block - 1		Default setting
	Groove for calibrating a SL 7 probe (tool type 500, tool edge position 7)		
	_KB[x,0]	Groove edge in plus direction of the ordinate	0
	_KB[x,1]	Groove edge in minus direction of the ordinate	0
	_KB[x,2]	Groove bottom in abscissa	0
	Groove for calibrating a SL 8 probe (tool type 500, tool edge position 8)		
	_KP[x,3]	Groove edge in plus direction of the abscissa	0
	_KP[x,4]	Groove edge in minus direction of the abscissa	0
	_KP[x,5]	Upper edge of groove in ordinate	0
	_KP[x,6]	Groove bottom in ordinate	0

Overview of calibrating groove pair (only for turning)

The representation refers to the working plane defined by G18.



840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

	_CM[] Monitoring for tool measurement with rotating spindle, only effective with _CBIT[12]=0				
		min. input value: -		max. input value: -	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: REAL				valid as of software version: SW 4.3	

Meaning:

		Default setting
_CM[0]	Max. permissible grinding wheel surface speed [m/min]	60
_CM[1]	Max. permissible spindle speed [rpm]	2000
_CM[2]	Minimum feed for probing [mm/min]	1
_CM[3]	Required measuring accuracy [mm]	0.005
_CM[4]	Max. permissible feed for probing	20
_CM[5]	Direction of spindle rotation	4
_CM[6]	Feed factor 1	10
_CM[7]	Feed factor 2	0

	_MFS[] Feedrates and spindle speeds for measuring with rotating spindle, only effective with CBIT[12]=1				
		min. input value: -		max. input value: -	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: REAL				valid as of software version: SW 4.3	

Meaning:

		Default setting
_MFS[0]	Feed 1st probing	0
_MFS[1]	Speed 1st probing	0
_MFS[2]	Feed 2nd probing	0
_MFS[3]	Speed 2nd probing	0
_MFS[4]	Feed 3rd probing	0
_MFS[5]	Speed 3rd probing	0

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

10.2.4 Central bits

	_CBIT Central bits		
	min. input value: 0	max. input value: 1	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: BOOLEAN	valid as of software version: SW 3.2		
Meaning:			Default setting
	_CBIT[0]	Measurement repetition after exceeding dimensional difference and safe area	0
	_CBIT[1]	M0 on measurement repetition	0
	_CBIT[2]	No M0 on alarm "Oversize", "undersize", "Permissible dimensional difference exceeded"	0
	_CBIT[3]	Flag for basic system setting of the control	1
	_CBIT[4]	*Currently not assigned*	0
	_CBIT[6...7]	*Currently not assigned*	0
	_CBIT[8]	Offset for mono probe position	0
	_CBIT[9]	Internally assigned	0
	_CBIT[10]	Log destination	0
	_CBIT[11]	Log header	0
	_CBIT[12]	*(only relevant for measuring milling tools with rotating spindle)* 0: Calculation of feedrate and spindle speed through measuring cycle 1: Specified by user	0
	_CBIT[13]	1: Delete the measuring cycle fields in GUD6 _TP[],_WP[], _KB[], _EV[], _MV[]	0
Measuring cycles SW 4.5 and higher	_CBIT[14]	0: Length of the probe relative to the center of the probe ball 1: Length of the probe relative to the end (only for probe type 710 or 2xx)	0
	_CBIT[15]	0: No effect 1: Enter result of probe ball computation in the geometry memory of the probe (radius)	0
Measuring cycles SW 5.4 and higher	_CBIT[5]	0: Tool measurement and calibration of the tool probe is performed in the basic coordinate system (in the machine coordinate system with the kinematic transformation switched off (only for CYCLE982) 1: Tool measurement and calibration of the tool probe is performed in the active WCS (only for CYCLE982)	0

Measurement repeated

If _CBIT[0] is set and the calculated difference exceeds the values of the parameters for dimensional difference and safe area, the measurement is repeated.
An alarm is only displayed in the alarm line with measurement repetition if _CBIT[1] is set.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

M0 with measurement repetition

If _CBIT[1] is set, and the parameter limits for dimensional difference control and safe area were exceeded, the repetition of the measurement must be started with NC START.
An alarm is displayed in the alarm line; it requires no acknowledgment.

No M0 on alarm

If _CBIT[2] is not set, M0 is not generated if the alarms "Oversize", "Undersize" or "Permissible dimensional difference exceeded" are output.

Flag for basic system setting

When starting up the measuring cycles, this bit has to be set according to the basic settings of the PLC (MD 10240).

- 0: INCH
- 1: Metric

If modifying the basic settings of the PLC results in _CBIT[3] no longer matching MD 10240, measuring cycles software versions up to and including SW 4.4 will delete data fields _TP[], _WP[], _KB[] and _EV[] the first time a measuring cycle is called after the modification has been made, will output a message indicating this, and will terminate the measuring cycle.
The user must calibrate the tool probe or workpiece probe before measuring tasks can be solved again.

For measuring cycles SW 4.5 and higher, these data fields are not deleted but converted. This means that it is no longer necessary to recalibrate the tool probe or workpiece probe. The data for tool measurement with rotating spindle (_CM[], _MFS[]) are also converted.

Tool measurement and calibration in the WCS

(for use with CYCLE982 only)
Measuring cycles versions SW 5.4 and higher permit tool measurement and calibration of the probe in the active WCS if _CBIT[5] is set. This requires the same WCS preconditions for calibration and measurement. That also permits tool measurement with active TRAANG transformation.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Mono probe position offset

If _CBIT[8] is set, the probe position is offset by the value programmed in _CORA.

Log destination

The destination for the log procedure can be selected via bit _CBIT[10]. With _CBIT[10]=0 the log is sent to a device, for example a printer, via RS-232-C; with _CBIT[10]=1 the log is sent to a file (not yet implemented).

Log header

_CBIT[11] is for selecting the log header. The standard log header is selected with _CBIT[11]=0. With _CBIT[11]=1 you can use a customized log header.

Calculating feedrate and speed using measuring cycle

If _CBIT[12]=0 is set, feedrate and spindle speed is calculated for tool measurement of milling tools with rotating spindle via the measuring cycle. If _CBIT[12]=1, the user specifies the feedrate and the spindle speed in data field _MFS[6].

Deleting measuring cycle data fields in the GUD6 block

If _CBIT[13]=1, the data fields _TP[],_WP[], _KB[], EV[], _MV[] and _CBIT[13] are zeroed for the following measuring cycle call.

Length of the probe (only for tool type 710 or 2xx)

If _CBIT[14]=0, the length of the probe must be entered relative to the center of the probe ball.
If _CBIT[14]=1, the length of the probe must be entered relative to the end of the probe ball.

Enter the effect probe ball radius in the geometry memory (only for tool type 710 or 2xx)

If _CBIT[15]=1, the probe ball computation is calibrated by entering the active probe ball radius in the geometry memory of the probe.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

10.2.5 Central strings

	_SI Central strings		
	min. input value: -	max. input value: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: STRING	valid as of software version:		
Meaning:			Default setting
	_SI[0]	*Currently not assigned*	
	_SI[1]	Software version	4

Software version

Here you have to enter the first digit of the version of the NCU software on the control, e.g. for SW 03.06.02, enter 3.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

10.2.6 Channel-oriented values

	_EVMVNUM Number of empirical values and mean values	
	min. input value: 0	max. input value: -
Changes valid after value assignment	Protection level: -	Unit: -
Data type: INTEGER	valid as of software version: SW 3.2	
Meaning:	_EVMVNUM[0] Number of empirical values _EVMVNUM[1] Number of mean values	Default setting 20 20

	_EV Empirical values	
	min. input value: -	max. input value: -
Changes valid after value assignment	Protection level: -	Unit: -
Data type: REAL	valid as of software version: SW 3.2	
Meaning:	Index "x" stands for the number of the empirical value - 1 _EV[x] Number of empirical values	Default setting 0

	_MV Mean values	
	min. input value: -	max. input value: -
Changes valid after value assignment	Protection level: -	Unit: -
Data type: REAL	valid as of software version: SW 3.2	
Meaning:	Index "x" stands for the number of the mean value - 1 _MV[x] Mean value	Default setting 0

	_SPEED Traversing velocities for intermediate positioning	
	min. input value: 0	max. input value: 100
Changes valid after value assignment	Protection level: -	Unit: -
Data type: REAL	valid as of software version: SW 3.2	
Meaning:	_SPEED[0] Rapid traverse speed in % when collision monitoring is not active (values between 1 and 100) _SPEED[1] Positioning speed plane for collision monitoring is active _SPEED[2] Positioning speed in the applicate if active collision monitoring as of measuring cycles SW 4.5 _SPEED[3] Fast measuring feed	Default setting 50 1000 1000 900

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840 Di

Rapid traverse speed

The intermediate positions calculated by the measuring cycles are approached at the maximum axis speed specified in percent. With 0 the maximum axis speed is effective.

This value is only effective with deactivated collision monitoring.

Positioning speed

The intermediate positions calculated by the measuring cycles are approached at the specified speed.

The values are only effective with active collision monitoring and must be > 0, otherwise an alarm message is issued.

Fast measuring feed

As of measuring cycles SW 4.5, the measurement can be carried out with two different feedrates.
The fast measuring is only active if _CHBIT[17] is set and _FA>1.
When you switch on the probe, it is retracted 2 mm and the actual measurement carried out with the feedrate programmed in _VMS.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

10.2.7 Channel-oriented bits

	_CHBIT Channel bits		
	min. input value: -	max. input value: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: BOOLEAN	valid as of software version: SW 3.2		

Meaning:

		Default setting
_CHBIT[0]	Measuring input workpiece measurement	0
_CHBIT[1]	Measuring input tool measurement	1
_CHBIT[2]	Collision monitoring	1
_CHBIT[3]	Tool offset mode with tool measurement	0
_CHBIT[4]	Mean value memory	0
_CHBIT[5]	Reverse EV included	0
_CHBIT[6]	Tool offset mode Workpiece measurement with automatic tool offset	0
_CHBIT[7]	Measured value offset for CYCLE994	0
_CHBIT[8]	Switching edge measuring input 1 (0 → 0/L edge)[1]	0
_CHBIT[9]	Switching edge measuring input 2 (0 → 0/L edge)[1]	0
_CHBIT[10]	Display of measured result screen	0
_CHBIT[11]	Acknowledgment with NC start	0
_CHBIT[12]	*No assignment at present*	
_CHBIT[13]	Coupling of spindle position with coordinates in the plane	0
_CHBIT[14]	Adapt spindle position	0
For measuring cycle SW 4.5 and higher		
_CHBIT[15]	0: Max. 5 measuring passes 1: Only one measuring pass	0
_CHBIT[16]	0: Retraction like for intermediate positioning 1: Retraction of measuring point with rapid traverse (only active if _CHBIT[2]=1)	0
_CHBIT[17]	0: Measurement with feed in _VMS 1: 1st measurement with feed in _SPEED[3] 2nd measurement with feed in _VMS	0
_CHBIT[18]	0: No effect 1: Measurement result screen retained until next time measuring cycle is called on measuring cycles version	0
SW 5.4 and higher		
_CHBIT[19]	0: No effect 1: On measuring in the G18 plane in the applicate (Y-axis) parameterization is performed analogously to parameterization in the ordinate (X-axis), the ZO is implemented in the specified ZO memory in the ordinate part (X-axis), the TO is implemented in the length (L1) active in the ordinate (X-axis) if not via _KNUM.	0
For measuring cycles SW 6.2 and higher		
_CHBIT[20]	0: No effect 1: Suppression of the starting angle positioning _STA1 in CYCLE982	0

1) relevant only up to SW 3

Measurement input workpiece measurement

_CHBIT[0]=0: Measuring input 1 is activated for workpiece measurement.
_CHBIT[0]=1: Measuring input 2 is activated for workpiece measurement.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Measurement input tool measurement

_CHBIT[1]=0: Measuring input 1 is activated for tool measurement.
_CHBIT[1]=1: Measuring input 2 is activated for tool measurement.

Collision monitoring

If _CHBIT[2] is set, the intermediate positioning calculated and approached by the measuring cycles is canceled as soon as the probe returns a switching signal. When aborted (collision) an alarm message is displayed.

Tool offset mode with tool measurement

_CHBIT[3]=0: The determined tool data (length or radius) are written in the geometry data of the tool.
The wear is deleted.
_CHBIT[3]=1: The calculated difference is written in the appropriate wear data of the tool. The geometry data remain unchanged.

Mean value calculation

Relevant for workpiece measurement with automatic tool offset.
_CHBIT[4]=0: The formula used to calculate the mean value (see Section 1.7) uses 0 as the old mean value.
The mean value obtained is not stored!
_CHBIT[4]=1: When computing the mean value, the value is calculated from the mean value memory programmed via EVNUM and then stored with the new mean value determined in this mean value memory.

Reverse EV inclusion

_CHBIT[5]=0: Empirical value (EV) is subtracted from the measured actual value.
_CHBIT[5]=1: Empirical value is added to the measured actual value.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Tool offset mode for workpiece measurement with automatic tool offset

_CHBIT[6]=0: The calculated offset value is included as an added value in the wear memory calculation (length or radius) of the specified tools.

_CHBIT[6]=1: The length or radius of the specified tool is compensated by the calculated offset value, entered in the appropriate geo memory and the appropriate wear memory set to zero.

Measured value offset for CYCLE994 (as of measuring cycles SW 5.4)

_CHBIT[7]=0: The trigger values derived in _WP[_PRNUM-1,1...4) are used to determine the actual value.

_CHBIT[7]=1: The diameter stored in _WP[_PRNUM-1,0] is used to determine the actual value.

Display of measurement result screen

_CHBIT[10]=1: Following measurement or calibration, the measured result is displayed automatically.

Acknowledge with NC Start

_CHBIT[11]=0: The measurement result screen is automatically deactivated at the end of the cycle. For measuring cycle SW 4.5 and higher, _CHBIT[18] must be equal to 0; otherwise, the effect described for CHBIT[18]=1 is produced.

_CHBIT[11]=1: After the measurement result screen is displayed, continuation of the measuring cycle is initiated by cycle M0 and the screen is deactivated after NC Start.

Static display of measured result (for measuring cycle SW 4.5 and higher)

_CHBIT[18]=0: Effect is defined by _CHBIT[11].

_CHBIT[18]=1: The measured result display is retained until the next measuring cycle is called. The NC program processing is not interrupted, _CHBIT[10] must be set, _CHBIT[11] must be 0!

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Link between spindle position and coordinate rotation

_CHBIT[13]=0: If multiprobes are used, there is no link between the spindle position and possible active coordinate rotation in the plane.

_CHBIT[13]=1: If multiprobes are used, the spindle is positioned as a function of the active coordinate rotation in the plane (rotation around applicate (infeed axis) so that the probing is at the same points in calibration and measurement.

Notice If other rotations are active, this function has no effect!

Adapt spindle positioning

_CHBIT[14]=0: When multiprobes and spindle positioning (_CHBIT[13]=1) are used, the spindle positioning is carried out as standard.
Angle of coordinate rotation in the plane 0°:
Spindle positioning 0°
Angle of coordinate rotation in the plane 90°:
Spindle positioning 270°

_CHBIT[14]=1: Spindle positioning performed in the opposite direction.
Angle of coordinate rotation in the plane 0°:
Spindle positioning 0°
Angle of coordinate rotation in the plane 90°:
Spindle positioning 90°

A coordinate rotation in the plane consists of

- one rotation around the Z axis with G17,
- one rotation around the Y axis with G18 or
- one rotation around the X axis with G19,

Number of measuring passes (for measuring cycle SW 4.5 and higher)

_CHBIT[15]=0: A maximum of 5 measuring attempts are performed before the error "Probe not responding" is generated.

_CHBIT[15]=1: An unsuccessful attempt produces the error message "Probe not responding".

Return velocity (for measuring cycle SW 4.5 and higher)

_CHBIT[16]=0: The return from the measuring point is carried out at the same speed as for intermediate positioning.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

_CHBIT[16]=1: The return velocity is always carried out at the rapid traverse percentage defined in _SPEED[0] and is only active for active collision monitoring.

Measuring with different feedrates (for measuring cycle SW 4.5 and higher)

_CHBIT[17]=0: Measured with the feedrate programmed in _VMS.

_CHBIT[17]=1: The measuring feedrate _SPEED[3] is used for traversing initially, after switching the probe returns 2 mm from the measuring point and the actual measurement then begins with the feedrate in _VMS.
The feedrate in _SPEED[3] is used only for a measuring path > 2 mm.

Treatment of the Y-axis in measurement in G18 (measuring cycle SW 5.4 and higher)

_CHBIT[19]=0: No effect

_CHBIT[19]=1: The setpoint input and parameterization of a protection zone when measuring the applicate (Y-axis) is performed in the same way as for the ordinate (X-axis), i.e. like for a transverse axis.
The TO (CYCLE974 and CYCLE994) is performed in the length (L1) active in the ordinate (X-axis), if no length is set in _KNUM.
The ZO is implemented in the ZO set in the ordinate part (X-axis)

Suppressing positioning of the milling spindle (SW 6.2 and higher)

_CHBIT[20]=0: No effect

_CHBIT[20]=1: During measurement of milling cutters in CYCLE982 with simple measurement variants it is possible to suppress positioning of the milling spindle to the value of the starting angle _STA1. This is possible with the following miller measurement variants: _MVAR=xxx001
(with x: 0 or 1, no other values)

840 D
NCU 572
NCU 573

810 D

840 Di

10.3 Data for measuring in JOG

10.3.1 Machine data for ensuring ability to function

11602 MD number	**ASUP_START_MASK** Ignore reasons for stopping ASUB	
Default setting: 0 **for measuring in JOG: 1, 3** **(Bit0=1)**	min. input value: 0	max. input value: 3
Changes are validated by Power ON	Protection level: 2/4	Unit: -
Data type: DWORD	valid as of software version: SW 4.1	
Meaning:	Bit 0: 1 ASUB start possible in JOG	

11604 MD number	**ASUP_START_PRIO_LEVEL** Priorities for "ASUP_START_MASK" active	
Default setting: 0 **for measuring in JOG: 1 - 64H**	min. input value: 0	max. input value: 64H
Changes are validated by Power ON	Protection level: 2/4	Unit: HEX
Data type: INT	valid as of software version: SW 4.1	
Meaning:	"ASUP_START_MASK" included from ASUB priority "64H" to ASUB priority 1.	

20110 MD number	**RESET_MODE_MASK** Defining control default setting after power-up and RESET	
Default setting: 0 **for measuring in JOG: at least 4045H** **(Bit0=1, Bit2=1, Bit6=1, Bit14=1)**	min. input value: 0	max. input value: 07FFFhex
Changes valid after RESET	Protection level: 2/7	Unit: HEX
Data type: DWORD	valid as of software version: SW 2	
Meaning:	Bit 0: 1 Tool length compensation active Bit 2: 1 Bit 6: 1 } After Power ON the last tool length compensation to be selected is active Bit 14: 1 Current setting of the basic frame is retained	

20112 MD number	**START_MODE_MASK** Defining control default setting after part program start	
Default setting: 400H **for measuring in JOG 400H** **(Bit6=0)**	min. input value: 0	max. input value: 07FFFhex
Changes valid after RESET	Protection level: 2/4	Unit: HEX
Data type DWORD	valid as of software version: SW 3.2	
Meaning:	Bit 6: 0 Active tool compensation maintained	

840 D
NCU 572
NCU 573

810 D

840 Di

<table>
<tr><td colspan="2">20310
MD number</td><td colspan="4">TOOL_MANAGEMENT_MASK
Channel-specific activation of tool management</td></tr>
<tr><td colspan="2">Default setting: 0
for measuring in JOG: at least 4001H (Bit0=1, Bit14=1)</td><td colspan="2">min. input value: 0</td><td colspan="2">max. input value: 0FFFFhex</td></tr>
<tr><td colspan="3">Changes are validated by Power ON</td><td colspan="2">Protection level: 2/4</td><td>Unit: HEX</td></tr>
<tr><td colspan="3">Data type DWORD</td><td colspan="3">valid as of software version: SW 2</td></tr>
<tr><td>Meaning:</td><td colspan="5">Bit 0: 1 Tool management active
Bit 14: 1 Automatic tool change on RESET and START according to MD 20110: RESET_MODR_MASK</td></tr>
</table>

840 D
NCU 572
NCU 573

810 D

840 Di

10.3.2 Modifying the GUD7 data block

Function

Notice

The GUD7 data block does not have to be modified if ShopMill is installed in the control.

Select definition file GUD7.DEF in menu "Services" in directory "Definitions" with the arrow keys and unload it by pressing the softkey "Unload".

Then open file GUD7.DEF by pressing the Enter key. In the section "**Measure**", remove the semicolons at the beginning of each definition line with the DEL key. This concerns the definition lines.

```
DEF CHAN BOOL    E_MESS_MS_IN=0
DEF CHAN BOOL    E_MESS_MT_IN=1
DEF CHAN REAL    E_MESS_D=5
DEF CHAN REAL    E_MESS_D_M=50
DEF CHAN REAL    E_MESS_D_L=2
DEF CHAN REAL    E_MESS_D_R=1
DEF CHAN REAL    E_MESS_FM=300
DEF CHAN REAL    E_MESS_F=2000
DEF CHAN REAL    E_MESS_FZ=2000
DEF CHAN REAL    E_MESS_MAX_V=100
DEF CHAN REAL    E_MESS_MAX_S=1000
DEF CHAN REAL    E_MESS_MAX_F=20
DEF CHAN REAL    E_MESS_MIN_F=1
DEF CHAN REAL    E_MESS_MIN_D=0.01
DEF CHAN INT     E_MESS_MT_TYP[3]=SET(0,0,0)
DEF CHAN INT     E_MESS_MT_AX[3]=SET(133,133,133)
DEF CHAN REAL    E_MESS_MT_DL[3]
DEF CHAN REAL    E_MESS_MT_DR[3]
DEF CHAN REAL    E_MESS_MT_DZ[3]=SET(2,2,2)
DEF CHAN INT     E_MESS_MT_DIR[3]=SET(-1,-1,-1)
DEF CHAN REAL    E_MESS[3]
DEF CHAN REAL    E_MEAS
```

It may be necessary for the machine manufacturer to adapt the number of fields for connectable tool measuring probes to the actual conditions of the machine. In the version supplied, three fields are provided for tool measuring probes (E_MESS_MT_....). If the number is altered, the TP field for running measuring cycles in the GUD6 block must also be changed accordingly and the altered number of fields entered in _CVAL field _CVAL[0].

840 D
NCU 572
NCU 573

810 D

840 Di

Example:
A milling machine has a tool measuring probe. Tool measurement is only performed in G17.

To save memory space, the definition lines in GUD7.DEF are altered as follows.

DEF CHAN INT	E_MESS_MT_TYP[**1**]=SET(0)
DEF CHAN INT	E_MESS_MT_AX[**1**]=SET(133)
DEF CHAN REAL	E_MESS_MT_DL[**1**]
DEF CHAN REAL	E_MESS_MT_DR[**1**]
DEF CHAN REAL	E_MESS_MT_DZ[**1**]=SET(2)
DEF CHAN INT	E_MESS_MT_DIR[**1**]=SET(-1)

In file GUD6.DEF, the following definition lines are also adapted.

N10 DEF NCK INT _CVAL[4]=(**1**,3,3,0) ;*1 tool measuring probe
N11 DEF NCK REAL _TP[**1**,10]=(0,0,0,0,0,0,0,133,0,2)

After saving and closing the editor, activate file GUD7.DEF by pressing the softkey "Activate". The global channel-specific variables have now been written to and pre-assigned in the control memory and can be altered later if necessary.

In the delivery status, the following settings are active:

E_MESS_MS_IN=0	Workpiece measuring probe at measuring input 1 connected
E_MESS_MT_IN=1	Tool measuring probe at measuring input 2 connected
E_MESS_D=5	Internal data for measuring in JOG not relevant
E_MESS_D_M=50	Measuring path for manual measuring [mm] (in front of and behind the measuring point)
E_MESS_D_L=2	Measuring path for length measurement [mm] for tool measurement (in front of and behind the measuring point)

840 D
NCU 572
NCU 573

810 D

840 Di

E_MESS_D_R=1	Measuring path for radius measurement [mm] for tool measurement (in front of and behind the measuring point)
E_MESS_FM=300	Measuring feedrate [mm/min] for workpiece measurement and calibration
E_MESS_F=2000	Plane feedrate for collision monitoring [mm/min]
E_MESS_FZ=2000	Infeed feedrate for collision monitoring [mm/min]
E_MESS_MAX_V=100	Max. peripheral speed for measuring with rotating spindle [m/min]
E_MESS_MAX_S=1000	Max. spindle speed for measuring with rotating spindle [rpm]
E_MESS_MAX_F=20	Max. feedrate for measuring with rotating spindle [mm/min]
E_MESS_MIN_F=1	Min. feedrate for measuring with rotating spindle [mm/min]
E_MESS_MIN_D=0.01	Measuring accuracy for measuring with rotating spindle [mm/min]
E_MESS_MT_TYP[3]=SET(0,0,0)	Three fields for tool measuring probe; tool measuring probe type; cube
E_MESS_MT_AX[3]=SET(133,133,133)	Permissible axis directions for tool measuring probe in X and Y in plus and minus direction, in Z in minus direction only
E_MESS_MT_DL[3]	Active diameter of tool measuring probe for length measurement 0
E_MESS_MT_DR[3]	Active diameter of tool measuring probe for radius measurement 0
E_MESS_MT_DZ[3]=SET(2,2,2)	Distance between tool measuring probe upper edge and tool lower edge [mm] for tool radius measurement 2
E_MESS_MT_DIR[3]=SET(-1,-1,-1)	Approach direction in plane of tool measuring probe for tool measurement (minus direction in 1st plane axis) -1

Notice
Data fields E_MESS_MT_DL[] and E_MESS_DR[] (active diameter, width of tool measuring probe for length/radius measurement) must be assigned.

840 D
NCU 572
NCU 573

810 D

840 Di

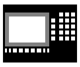
10.3.3 Settings in data block GUD6

Function

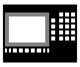
The channel specific data fields _JM_I[ ], and _JM_B[ ] in data block GUD6 are used for adaptation to the requirements of the machine

N92 DEF CHAN INT _JM_I[5]=SET(0,1,1,17,0)

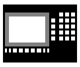
	_JM_I INT value field for JOG measurement		
	min. input value: -	max. input value: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: INT	valid as of software version: 5.3		
Meaning:			Default setting
	_JM_I[0]	Specified workpiece measuring probe number 0: Specified by _JM_I[1] 1: Specified by tool parameter (ShopMill)	0
	_JM_I[1]	Probe number and probe type for workpiece measurement Only active when _JM_I[0]=0	1
	_JM_I[2]	Measuring probe number for tool measurement	1
	_JM_I[3]	Working plane 17: Measurement in G17 plane 18: Measurement in G18 plane 19: Measurement in G19 plane All other values: Measurement in the plane defined in machine data.	17
	_JM_I[4]	Definition of the active ZO for measurement 0: Measurement with G500 1...99: Measurement with defined settable zero offset G54...G57 or G505...G599 where 1: G54...4: G57 5...99: G505...G599 100: Measurement with the ZO defined in machine data	0



840 D
NCU 572
NCU 573



810 D



840 Di

	__JM_B Bool value field for JOG measurement		
	min. input value: -	max. input value: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: BOOLEAN	valid as of software version: 5.3		
Meaning:			Default setting
	_JM_B[0]	Tool offset mode for tool measuring 0: Offset in Geo for tool measuring 1: Offset in wear	0
	_JM_B[1]	Number of measurement attempts 0: 5 measurement attempts 1: 1 measurement attempt	1
	_JM_B[2]	Retraction from measurement point 0: Retraction as for intermediate positioning 1: Retraction with rapid traverse	0
	_JM_B[3]	Fast measuring feedrate 0: Measure with measuring feedrate 1: 1. Measurement with feedrate in _SPEED[3] 2. Measurement with measuring feedrate	0
	_JM_B[4]	Not assigned	0
	_JM_B[5]	Not assigned	0
	_JM_B[6]	Internal date	0

840 D
NCU 572
NCU 573

810 D

840 Di

10.3.4 Loading files for measuring in JOG

Function

The files located on diskette 2 in directory
JOG_MESS

E_MS_CAL.SPF	For calibrating a workpiece measuring probe
E_MS_CAN.SPF	To measure a corner
E_MS_HOL.SPF	To measure a hole
E_MS_PIN.SPF	To measure a spigot/shaft
E_MT_CAL.SPF	For calibrating a tool measuring probe
E_MT_LEN.SPF	For length measurement of a tool
E_MT_RAD.SPF	For radius measurement of a tool
CYC_JM.SPF	Auxiliary cycle for measuring
CYC_JMC.SPF	Auxiliary cycle for corner calculation

are transferred to the control into directory "Standard cycles" from the diskette in menu "Services" after selection of softkey "Data in", "Diskette" and selection of the file in question and then pressing the "Start" softkey. They must then be loaded into the NC memory by pressing the softkey "Load". After the next Power on, they are known to the control.

The other files

JOG_MEAS.COM	Configuring file for measuring in JOG user interface
MA_JOG.COM	Configuring file for the softkeys for measuring in JOG in the JOG basic display
BMP_FILE.ARC	Help displays for measuring in JOG

must also be transferred to the control.

■

840 D
NCU 572
NCU 573

810 D

840 Di

Notes

Examples

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

11.1 Determining the repeat accuracy

Function

Test program

The program is used to determine the measuring scattering (repeat accuracy) of the entire measuring system (machine-probe-signal transfer to NC).

In the example, measurements are carried out 10 times in the X axis and the measured values are stored in workpiece coordinates.
Thus the so-called accidental measurement deviations, which are not subject to a trend, can be determined.

Example:

%_N_CHECK_ACCURATE_MPF			
;$PATH=/_N_MPF_DIR			
N05	DEF INT SIGNAL, II		;Variable definition
N10	DEF REAL MEAS.VALUE_IN_X[10]		
N15	G17 T1 D1		;Start conditions, preselect tool offset for ;probe
N20	ANF: G0 X0 F150	←	;Prepositioning in the measured axis
N25	MEAS=+1 G1 X100	←	;Measurement at 1st measurement ;input with switching signal not ;deflected, deflected in the X axis
N30	STOPRE	←	;Stop decoding for subsequent ;evaluation of result
N35	SIGNAL= $AC_MEA[1]		;Read software switching signal at ;1st measurement input
N37	IF SIGNAL == 0 GOTOF_FEHL1		;Check switching signal
N40	MEAS.VALUE_IN_X[II]=$AA_MW[X]		;Read measured value in workpiece ;coordinates
N50	II=II+1		
N60	IF II<10 GOTOB_ANF		
N65	M0		;Repeat 10 times
N70	M02		
N80	_FEHL1: MSG ("Probe does not switch")		
N90	M0		
N95	M02		

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

After selecting the parameter display (user-defined variables), the measurement results can be read from the array MEAS.VALUE_IN_X[10] while program execution is still active.

11.2 Adapting the data for a particular machine

Function

There are two main steps for adapting the data to a specific machine:

1. Adapting the data configuration in the GUD modules and loading them in the PLC.
2. Defining values for specific measuring cycle data.

Explanation

1. Adapting the data definition

The following example shows how to adapt the data blocks GUD5.DEF and GUD6.DEF to a machine with SINUMERIK 840D with the characteristics described below:

- SINUMERIK 840 D has software status 4xx
- 2 data fields for use with a tool probe with disc in XY and a disk diameter of 20 mm,
- 2 data fields for use with a tool probe,
- without calibration groove pair,
- 10 empirical values and mean values are to be used respectively.

Example:

%_N_GUD6_DEF

;$PATH=/_N_DEF_DIR

;27.04.01 adaptation to machine_1

...

N10 DEF NCK INT _CVAL[4]=(**2**,**2**,**0**,0)[1)]

N11 DEF NCK REAL _TP[**2**,10]=(0,0,0,0,0,0,0,133,0,2)[1)]

N12 DEF NCK REAL _WP[**2**,11][1)]

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

N13 DEF NCK REAL _KB[3,7][1)]
N14 DEF NCK REAL _CM[8]=(60,2000,1,0.005,20,4,10,0)
N15 DEF NCK REAL _MFS[6]
N20 DEF NCK BOOL _CBIT[16]=(0,0,0,1,0,0,0,0,1,0,0,0,0,0,0,0)
N30 DEF NCK STRING[8] _SI[3]=("",**4**,"")
N40 DEF CHAN INT _EVMVNUM[2]=(**10**,**10**)
N41 DEF CHAN REAL _SPEED[4]=(50,1000,1000,900)
N50 DEF CHAN BOOL _CHBIT[20]=(0,1,1,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0)
N60 DEF NCK STRING[32] _PROTNAME[2]
N61 DEF NCK STRING[80] _HEADLINE[10]
N62 DEF NCK INT _PROTFORM[6]=SET(60,80,1,5,1,12)
N63 DEF NCK CHAR _PROTSYM[2]
N64 DEF NCK STRING[100] _PROTVAL[13]
N65 DEF NCK INT _PMI[4]
N66 DEF NCK INT _SP_B[20]
N67 DEF NCK STRING[12] _TXT[100]
N68 DEF NCK INT _DIGIT
...
M17
%_N_GUD5_DEF
;$PATH=/_N_DEF_DIR
;27.04.01 adaptation to machine_1
...
N40 DEF CHAN REAL _EV[**10**][1)]
N50 DEF CHAN REAL _MV[**10**][1)]
...
N99 M02

1) Characters and numbers displayed in bold type indicate changes compared to the previous version

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Explanation

2. Adapting specific values

Value adaptation is achieved by loading a part program in the PLC and running it in AUTOMATIC mode.

The following adaptations are to be achieved:

- Retraction of the probe from the measuring point at 80% of the rapid traverse speed,
- Measurement repetition when the permissible dimensional difference or the safe area are exceeded, but without M0,
- Static display of measurement results
- No repetition of an unsuccessful attempted measurement

%_N_MZ_VALUE ASSIGNMENT_MPF	
;$PATH=/_N_MPF_DIR	
;27.04.01 Default measuring cycle data on machine_1	
N05 _TP[0,6]=20 _TP[1,6]=20 _TP[0,8]=101 _TP[1,8]=101	;Specification of disk diameter and type ;of tool probe
N10 _SPEED[0]=80	;Reduction of rapid traverse to 80 %
N20 _CBIT[0]=1	;Preset measurement repeat bit
N30 _CBIT[14]=1	;Length of workpiece probe relative to end ;of probe ball
N40 _CHBIT[10]=1 _CHBIT[11]=0 _CHBIT[18]=1	;Bits for static display of measurement ;result.
N50 _CHBIT[15]=1	;Measurement abort after unsuccessful ;attempt
N55 _CHBIT[16]=1	;Retraction from measuring point at % rapid ;traverse speed defined in _SPEED[0]
N99 M02	

■

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Notes

Data Fields, Lists

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

12.1 Machine data

Number	Identifier	Name	Reference
General ($MN_...)			
10132	MMC_CMD_TIMEOUT	Monitoring time for MMC command in part program	
11420	LEN_PROTOCOL_FILE	File size for log files (KB)	
13200	MEAS_PROBE_LOW_ACTIVE	Switching performance of the probe	M5
18102	MM_TYPE_OF_CUTTING_EDGE	Type of D number programming (SRAM)	W1
18118	MM_NUM_GUD_MODULES	Number of data blocks	S7
18120	MM_NUM_GUD_NAMES_NCK	Number of GUD variables in PLC	S7
18130	MM_NUM_GUD_NAMES_CHAN	Number of GUD variables per channel	S7
18150	MM_GUD_VALUES_MEM	Memory for values of the GUD variables	S7
18170	MM_NUM_MAX_FUNC_NAMES	Number of cycles with transfer parameters	S7
18180	MM_NUM_MAX_FUNC_PARAM	Number of special functions (cycles, DRAM)	S7
Channel-specific ($MC_...)			
28020	MM_NUM_LUD_NAMES_TOTAL	Number of LUD variables in total (in all program levels)	S7

12.2 Measuring cycle data

Explanation

The measuring cycle data are stored in modules GUD5 and GUD6.

Number	Identifier	Name	Reference
General			
	_CBIT[16]	Central measuring cycle bits	
	_CVAL[4]	Central values	
	_TP[3,6]	Tool probe	
	_WP[3,9]	Workpiece probe	
	_KB[3,7]	Calibration block	
	_SI[2]	Central measuring cycle strings	
	_CM[]	Monitoring for tool measurement with rotating spindle	
	_MFS[]	Feedrates and speeds for measuring with rotating spindle	
Channel-specific			
	_CHBIT[16]	Channel-specific measuring cycle bits	
	_EV[20]	Empirical values	
	_EVMVNUM[2]	Number of empirical values and mean values	
	_MV[20]	Mean values	
	_SPEED[3]	Traversing velocities for intermediate positioning	

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

12.3 Alarms

General notes

If faulty states are detected in the measuring cycles, an alarm is generated and execution of the measuring cycle is aborted.
In addition, the measuring cycles issue messages in the dialog line of the PLC. These messages do not interrupt execution.

Error handling in the measuring cycles

Alarms with numbers between 61000 and 62999 are generated in the measuring cycles. This number range is divided again into alarm reactions and delete criteria.
The error text which is displayed together with the alarm number provides more information on the error cause.

Alarm number	Delete criterion	Alarm reaction
61000 ... 61999	NC_RESET	Block preparation in NC is aborted
62000 ... 62999	Delete key	Program execution is not interrupted; display only.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Overview of the measuring cycle alarms

The following table displays the errors which occur in the measuring cycles, together with error location and tips for remedying the errors.

Alarm number	Alarm text	Source	Meaning, remedy
61016	"System frame for cycles missing"	All	MD 28082: MM_SYSTEM_FRAME_MASK, Set bit 5=1
61301	"Probe does not switch"	All	• Check measurement input • Check measurement path • Probe defective
61302	"Probe – collision"	All	There is an obstacle in the probe's traversing path.
61303	"Safe area" violated	All	• Check setpoint • Increase parameter _TSA
61306	"Permissible dimensional difference exceeded"	All	• Check setpoint • Increase parameter _TDIF
61307	"Incorrect measurement variant"	All	Parameter _MVAR has an illegal value.
61308	"Check measurement path 2a"	All	Parameter _FA is ≤ 0.
61309	"Check probe type"	All except CYCLE971 CYCLE972	Tool type of workpiece probe in TO memory is not allowed
		CYCLE971	Tool probe type entered in _TP[x,8] not allowed.
61310	"Scale factor is active"	All	Measurements are not possible when the scale factor is active.
61311	"No D number is active"	All	There is no tool offset selected for the probe (with workpiece measuring) or no tool offset selected for the active tool (with tool measuring).
61312	"Check measuring cycle number"	All	Measuring cycle called not permissible.
61313	"Check probe number"	All	The probe number is illegal (_PRNUM). Remedy: Correct _PRNUM or set up data field _TP[] or _WP[] for additio-nal tool and workpiece probes and adapt _CVAL[0]/_CVAL[1] ccordingly.
61314	"Check selected tool type"	CYCLE971 CYCLE972 CYCLE982	Tool probe not permitted for tool measurement/tool probe calibration.

840 D NCU 571	840 D NCU 572 NCU 573	810 D	840 Di

Alarm number	Alarm text	Source	Meaning, remedy
61315	"Check tool edge position"	CYCLE972 CYCLE973 CYCLE974 CYCLE982 CYCLE994	Check tool edge position of tool (measuring probe) in TO memory
61316	"Center point and radius cannot be determined"	CYCLE979	It is not possible to calculate a circle from the measured points.
61317	"Check parameter CYCLE116"	CYCLE979	Parameterization faulty; needs 3 or 4 points for calculating the center point
61318	"Check weighting factor _K"	CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE994 CYCLE998	Parameter _K is 0.
61319	"Check call parameter CYCLE114"	As 61318	Internal error measuring cycles.
61320	"Check tool number"	All	If tool management is active, parameter _TNUM=0**, and** parameter _TNAME is not assigned or the specified tool name for tool management is not known.
61321	"Check ZO memory number"	As 61318	The ZO with the number specified in _KNUM does not exist.
61322	"Check 4th digit in _KNUM"	As 61318 CYCLE114	4th digit position in _KNUM > 2
61323	"Check 5th digit in _KNUM"	As 61318 CYCLE114	5th digit position in _KNUM > 1
61324	"Check 6th digit in _KNUM"	As 61318 CYCLE114	6th digit position in _KNUM contains invalid value (permissible values 1, 2, 3, 4)
61325	"Check measuring axis/ offset axis"	All except CYCLE977 CYCLE979	Parameter for the measuring axis _MA has an incorrect value.
61326	"Check measuring direction"	CYCLE973 CYCLE976	Parameter for the measuring direction _MD has an incorrect value.
61327	"Program reset necessary"	All except CYCLE973 CYCLE976	NC reset necessary

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Alarm number	Alarm text	Source	Meaning, remedy
61328	"Check D number"	All	The D number in parameter KNUM is 0.
61329	"Check rotary axis"	CYCLE998	The axis number specified in parameter _RA is not assigned to a name (MD 20080) or the axis is not configured as a rotary axis (MD 30300).
61330	"Coordinate rotation active"	CYCLE972 CYCLE973 CYCLE974 CYCLE994	Measurements are not possible in a rotated coordinate system.
61331	"Angle too large, change measuring axis"	CYCLE998	Parameter _STA1 is too large for the specified measuring axis; select another measuring axis.
61332	"Change position of tool tip"	CYCLE971 CYCLE972 CYCLE982 E_MT_CAL E_MT_LEN E_MT_RAD	Position of tool is not correct; change starting point of measurement.
61333	"Check calibration block number"	CYCLE973	Parameter _CALNUM is too large: 1. Reduce _CALNUM to a permissible value 2. Increase maximum value _CVAL[2] in GUD6
61334	"Check protection zone"	CYCLE977	Parameter _SZA/_SZD too large or too small
61336	"geometry axes not available"	All	No geometry axes are configured; change machine data in MD 20060.
61338	"Positioning speed is zero"	All	Parameter _SPEED[1], _SPEED[2] in GUD6 is 0
61339	"Offset factor rapid traverse < 0"	All	Check parameter _SPEED[0] in GUD6.
61340	"Incorrect alarm number"	All	Internal error measuring cycles.
61341	"Probe in active plane not calibrated"	CYCLE974 CYCLE977 CYCLE978 CYCLE979	Calibrate probe before cycle call.
61342	"Software version entry in GUD6 incomplete or wrong format"	CYCLE110	_SI[1] in GUD6 has no value or a value < 3

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Alarm number	Alarm text	Source	Meaning, remedy
61343	"Tool for specified tool identifier does not exist"	All	Check name of tool identifier
61344	"Several tools are active"	All	Remove tool from other spindle.
61345	"Parameterized D number (_KNUM) too large"	All	Reduce D number in _KNUM, check software or shallow D number MD
61346	"Distance between starting point and measuring point _SETV[0] and _SETV[1] ≤ 0"	CYCLE961	Parameters _SETV[0] or _SETV[1] are not assigned or are less than 0
61347	"Angle 1st edge - 2nd edge is 0"	CYCLE961	Parameter _INCA is 0.
61349	"Distance between tool probe top edge and measuring position for tool radius measurement is 0"	CYCLE971	Parameter _TP[x,9] distance between tool probe top edge and bottom edge is 0; relevant for radius measurement
61350	"Feedrate, speed for tool measurement with rotating spindle not programmed in _MFS"	CYCLE971	Measurement feed and/or spindle speed for tool measurement with rotating spindle not specified in GUD variable _MFS[2].
61351	"Tool length or radius is 0"	CYCLE971	The length or radius for the active tool is zero.
61352	"Illegal path for log file"	CYCLE106	The path specification for the log file is incorrect.
61353	"Path for log file does not exist"	CYCLE106	The specified directory does not exist or the path indicated is incorrect.
61354	"Log file not found"	CYCLE106	No name was specified for the log file.
61355	"Incorrect file type for log file"	CYCLE106	The file extension for the log file is incorrect.
61356	"Log file already in use"	CYCLE106	The log file is already used by another NC program.
61357	"No resources available"	CYCLE106	Insufficient NC memory available, delete files.
61358	"Logging error"	CYCLE106	Internal error, contact hotline
61359	"Continue with RESET"	CYCLE106	Internal error, contact hotline

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Alarm number	Alarm text	Source	Meaning, remedy
61360	"Undefined logging job - press RESET to continue"	CYCLE106	The cycle CYCLE106 was called with an incorrect parameter.
61361	"Unable to log variable"	CYCLE105	The value specified in _PROTVAL[] cannot be logged.
61362	"Too many values"	CYCLE118	4th parameter for CYCLE118 is greater than 10.
61363	"Max. number of value lines exceeded"	CYCLE105	Reduce number of lines.
61364	"Check distance between measuring point 1 and measuring point 2.	CYCLE998	Parameter _ID is ≤ 0.
61365	"Check circular feed"	CYCLE979	Parameter _RF is ≤ 0.
61366	"Direction of rotation for tool measurement with rotating spindle in _CM[5] is not defined"	CYCLE971	Permissible values for data field _CM[5] in the GUD6 module are 3 (corresponds to M3) and 4 (corresponds to M4)
61367	"The points P1 and P2, P3 and P4 are identical".	CYCLE961	Various positions specified for the different positions of _SETV[0...7].
61368	"The straight line defined by P1 and P2 or P3 and P4 do not produce an intersection"	CYCLE961	Various positions specified for the different positions of _SETV[0...7].
61369	"Unable to uniquely determine position of corner, check parameter _SETV[0...7]"	CYCLE961	Define P1 and P2, or P3 and P4 so that the intersection of the straight lines through these points lies outside the section defined by P1 and P2 or P3 and P4.
61370	"_PROTVAL[0] - _PROTVAL [5] do not contain entries"	CYCLE105 CYCLE108	Assign values to _PROTVAL[0...5].
61371	"The log produced by the column width and number columns exceed 200 characters per line"	CYCLE105 CYCLE108	Reduce the column width or number of columns.
61372 (measuring cycle SW 6.2 and higher)	"Selected measurement variant requires an SPOS-capable spindle"	All	Change measurement variant or check machine equipment
61373 (measuring cycle SW 6.2 and higher)	"Mono probe requires an SPOS-capable spindle"	All	Check machine equipment
61401	"Probe is not responding, travel limitation by software end position"	CYCLE961 CYCLE971 CYCLE976 CYCLE977 CYCLE978 CYCLE998	Unable to reach setpoint position because software limit software end position exceeded.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Alarm number	Alarm text	Source	Meaning, remedy
61402	"Probe collision, travel limitation through software end position"	CYCLE977	The position path in the plane has been limited by the software end position for measurement variants shaft/web. Infeed in the infeed axis caused the sensor response.
61403	"Internal cycle error in frame calculation"	All	Call SIEMENS hotline
62303	"Safe area" violated	All	• Check setpoint • Increase parameter _TSA
62304	"Oversize"	CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE994	Actual/setpoint difference is greater than the upper tolerance level (parameter _TUL)
62305	"Undersize"	CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE994	Actual/setpoint difference is less than the lower tolerance level (parameter _TLL)
62306	"Permissible dimensional difference exceeded"	CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE994	Actual/setpoint difference is greater than the tolerance parameter _TDIF, tool data are not corrected.
62307	"Max. number of characters per line exceeded"	CYCLE105	• Number of characters per line not sufficient • Increase value in _PROTFILE[1]
62308	"Variable column width not possible"	CYCLE105	• No variable column widths can be generated because no header exists. • A fixed column width of 12 characters is used. • Remedy: Complete header in _PROTVAL[]
62309	"Column width not sufficient"	CYCLE105	• Value to be logged is greater than the column width. • Adapt _PROTFORM[5] or change header for variable column width.

■

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Notes

Appendix

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

A Overview of measuring cycle parameters

Parameter definition

Parameters must be defined

Parameter is not used in the cycle

The definition of the parameter depends on the measurement variant, other parameters, or on the machine configuration

CYCLE 961			Workpiece measurement					
Parameters GUD5	Type	Comparable parameters 840C	Automatic setup inside and outside corner for G17: in XY plane for G18: in ZX plane for G19: in YZ plane					
			Specifying distances and angles				Specifying 4 points	
			Corner inside 3 measuring points	Corner outside 3 measuring points	Corner inside 4 measuring points	Corner outside 4 measuring points	Corner inside	Corner outside
_CALNUM	INTEGER	R12						
_CORA	REAL	R13						
_CPA	REAL	R20						
_CPO	REAL	R21						
_EVNUM	INTEGER	R11						
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm! only included if calculated larger than internal value					
_ID	REAL	R19		Retraction in infeed axis, incremental for overtraveling corner if _ID=0 bypasses the corner		Retraction in infeed axis, incremental for overtraveling corner if _ID=0 bypasses the corner	Infeed of positioning depth to measuring depth (incremental)	
_INCA	REAL 179.5 ..179.5 degrees	R26	Angle from 1st edge to 2nd edge of the workpiece (clockwise negative)					
_K	INTEGER	R29						

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 961			Workpiece measurement					
Parameters GUD5	**Type**	**Comparable parameters 840C**	**Automatic setup inside and outside corner for G17: in XY plane for G18: in ZX plane for G19: in YZ plane**					
			Specifying distances and angles				**Specifying 4 points**	
			Corner inside 3 measuring points	**Corner outside 3 measuring points**	**Corner inside 4 measuring points**	**Corner outside 4 measuring points**	**Corner inside**	**Corner outside**
_KNUM	INTEGER >=0	R10	Without/with automatic offset of the ZO memory 0 without offset 1..99 autom. offset in ZO G54...G57 G505...G599 1000 autom. offset in basic frame G500					
_MA	INTEGER	R30						
_MD	INTEGER	R31						
_MVAR	INTEGER >0	R23	Measurement variant 105	106	107	108	117	118
_NMSP	INTEGER >0	R27	Number of measurements at the same location					
_PRNUM	INTEGER >0	R22	Probe number (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM-1]**)					
_RA	INTEGER	R31						
_RF	REAL	R31						
_SETVAL	REAL	R32						
_SETV[0]	REAL		Distance between starting point and measuring point 2 (positive only)				Coordinates of point P1 in the active workpiece coordinate system (abscissa)	
_SETV[1]	REAL		Distance between starting point and measuring point 4 (positive only)				Coordinates of point P1 in the active workpiece coordinate system (ordinate)	
_SETV[2]	REAL		Distance between measured and required corner point in abscissa only active if **_SETV[4]>1**				Coordinates of point P2 in the active workpiece coordinate system (abscissa)	
_SETV[3]	REAL		Distance between measured and required corner point in ordinate only active if **_SETV[4]>1**				Coordinates of point P2 in the active workpiece coordinate system (ordinate)	
_SETV[4]	REAL		1: Measured corner 2: Offset in absc. 3: Offset in absc. and ordinate 4: Offset in ord.				Coordinates of point P3 in the active workpiece coordinate system (abscissa)	

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 961			Workpiece measurement					
Parameters GUD5	Type	Com-parable para-meters 840C	Automatic setup inside and outside corner for G17: in XY plane for G18: in ZX plane for G19: in YZ plane					
			Specifying distances and angles				Specifying 4 points	
			Corner inside	Corner outside	Corner inside	Corner outside	Corner inside	Corner outside
			3 measuring points		4 measuring points			
_SETV[5]	REAL						Coordinates of point P3 in the active workpiece coordinate system (ordinate)	
_SETV[6]	REAL						Coordinates of point P4 in the active workpiece coordinate system (abscissa)	
_SETV[7]	REAL						Coordinates of point P4 in the active workpiece coordinate system (ordinate)	
_STA1	REAL 0...360 degrees	R24	Approx. angle of posit. direction of the abscissa with respect to 1st edge of the workpiece (reference edge), clockwise negative					
_SZA	REAL	R19						
_SZO	REAL	R18						
_TDIF	REAL	R37						
_TMV	REAL	R34						
_TNAME	STRING[]							
_TNUM	INTEGER	R9						
_TUL	REAL	R40						
_TLL	REAL	R41						
_TSA	REAL	R36						
_TZL	REAL	R33						
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)					

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 971			Tool measurement of milling tools on milling machines		
Parameters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible axes abscissa/ordinate/applicate for G17: X=1 Y=2 Z=3 for G18: Z=1 X=2 Y=3 for G19: Y=1 Z=2 X=3**		
			Calibrate tool probe	**Measure tool**	
_CALNUM	INTEGER	R12			
_CORA	REAL	R13			
_CPA	REAL	R20			
_CPO	REAL	R21			
_EVNUM	INTEGER >=0	R11		Empirical value memory number number of data field **GUD5:_EV[_EVNUM-1]**	
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm!		
			For incremental calibration, the direction of travel is specified by the sign of _FA		
_ID	REAL >= 0	R19		Normally 0, on multiple cutters the offset between the highest point of the cutting edge and the length for radius measurement (or the radius for length measurement).	
_INCA	REAL	R26			
_K	INTEGER	R29			
_KNUM	INTEGER	R10			
_MA	INTEGER >=1	R30	Measuring axis 1..3		
			1: Calibration in +/- direction in 1 (abscissa) 2: Calibration in +/- direction in 2 (ordinate) 3: Calibration in +/- direction in 3 (applicate) For calibration in plane also possible 102: a) Calculation of the center in 1 (abscissa) b) Calibration in 2 (ordinate) 201: a) Calculation of the center in 2 (ordinate) b) Calibration in 1 (abscissa) **Not for incremental calibration!**	1: Measurement of radius in direction 1 (abscissa) 2: Measurement of radius in direction 2 (ordinate) 3: Measurement of the length on center of the tool probe 103: Measurement of the length, offset about radius in 1 (abscissa) 203: Measurement of the length, offset about radius in 2 (ordinate)	
_MD	INTEGER	R31			

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 971			Tool measurement of milling tools on milling machines			
Parameters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible axes abscissa/ordinate/applicate** **for G17: X=1 Y=2 Z=3** **for G18: Z=1 X=2 Y=3** **for G19: Y=1 Z=2 X=3**			
			Calibrate tool probe	**Measure tool**		
_MVAR	INTEGER >=0	R23	Measurement variant 0 Calibration in measuring axis after previous positioning on center of the measuring cube 10000: Incremental calibration traversing movement only in the measuring axis	1: Measurement with stationary spindle of length or radius 2: Measurement with rotating spindle direction of rotation before cycle call is retained! With spindle stopped, direction of rotation off **_CM[5]**		
_NMSP	INTEGER >0	R27	Number of measurements at the same location			
_PRNUM	INTEGER >0	R22	Tool probe number (number of the data field assigned to the tool probe **GUD6:_TP[_PRNUM-1]**)			
_RA	INTEGER	R31				
_RF	REAL	R31				
_SETVAL	REAL	R32				
_SETV[8]	REAL					
_STA1	REAL 0...360 degrees	R24				
_SZA	REAL	R19				
_SZO	REAL	R18				
_TDIF	REAL >0	R37		Dimensions difference check		
_TMV	REAL	R34				
_TNAME	STRING[32]					
_TNUM	INTEGER	R9				
_TUL	REAL	R40				
_TLL	REAL	R41				
_TSA	REAL >0	R36	Safe area			
_TZL	REAL >=0	R33	Zero offset area			
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)			
_CM[] GUD6 data item	REAL	REAL		Cycle-internal calculation of **F**, **S** from monitoring data in **_CM[]** **Only active if _CBIT[12]=1**		
_MFS[] GUD6-data item	REAL	REAL		Specification of **F**, **S** by user in **_MFS[]** **Only active if _CBIT[12]=1**		

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Parameters GUD5	Type	Comparable parameters 840C	Calibrate tool probe	Measure tool	
CYCLE 972			**Tool measurement of turning tools with cutting edge 1 – 8 on turning machines**		
			Possible axes abscissa/ordinate for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2		
_CALNUM	INTEGER	R12			
_CORA	REAL	R13			
_CPA	REAL	R20			
_CPO	REAL	R21			
_EVNUM	INTEGER >=0	R11		Empirical value memory number number of data field **GUD5:_EV[_EVNUM-1]**	
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm!		
_ID	REAL	R19			
_INCA	REAL	R26			
_K	INTEGER	R29			
_KNUM	INTEGER	R10			
_MA	INTEGER >0	R30	Measuring axis 1..2		
_MD	INTEGER	R31			
_MVAR	INTEGER	R23	Measurement variant 0	1	
_NMSP	INTEGER >=1	R27	Number of measurements at the same location		
_PRNUM	INTEGER >=1	R22	Tool probe number (number of the data field assigned to the tool probe **GUD6:_TP[_PRNUM-1]**)		
_RA	INTEGER	R31			
_RF	REAL	R31			
_SETVAL	REAL	R32			
_SETV[8]	REAL				
_STA1	REAL	R24			
_SZA	REAL	R19			
_SZO	REAL	R18			
_TDIF	REAL	R37		Dimensions difference check	
_TMV	REAL	R34			
_TNAME	STRING[]				
_TNUM	INTEGER	R9			
_TUL	REAL	R40			
_TLL	REAL	R41			
_TSA	REAL >0	R36	Safe area		
_TZL	REAL >=0	R33	Zero offset area		
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)		

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 973			Workpiece measurement	
Para-meters GUD5	Type	Com-parable para-meters 840C	Possible axes abscissa/ordinate for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2	
			Workpiece probe calibration	
			with reference data	with any data
			Groove	Surface
_CALNUM	INTEGER >0	R12	Number of the gauging block (number of the data field assigned **GUD6:_KB[_CALNUM-1]**)	
_CORA	REAL	R13		
_CPA	REAL	R20		
_CPO	REAL	R21		
_EVNUM	INTEGER	R11		
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a" always 1mm!	
_ID	REAL	R19		
_INCA	REAL	R26		
_K	INTEGER	R29		
_KNUM	INTEGER	R10		
_MA	INTEGER >0	R30	Measuring axis	
_MD	INTEGER	R31	Measuring direction (0 = positive / 1 = negative)	
_MVAR	INTEGER >=0	R23	Measurement variant **5 4 3 2 1** digits 2/1: **0 0** Any surface; **1 3** Reference groove digit 3: **0** Without calculation probe tip; **1** With calculation probe tip (only for calibration in groove) digit 4: **1** 1 axis direction (specifying measuring axis and axis direction); **2** 2 axis directions (specifying measuring axis) digit 5: **0** without position calculation; **1** with position calculation (only for calibration in groove) 13	0
_NMSP	INTEGER >0	R27	Number of measurements at the same location	
_PRNUM	INTEGER >0	R22	Tool probe number (number of the data field assigned to the tool probe **GUD6:_WP[_PRNUM-1]**)	
_RA	INTEGER	R31		
_RF	REAL	R31		
_SETVAL	REAL	R32 R42		Calibration setpoint
_SETV[8]	REAL			
_SZA	REAL	R19		
_SZO	REAL	R18		

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 973			Workpiece measurement	
Para-meters GUD5	Type	Com-parable para-meters 840C	Possible axes abscissa/ordinate for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2	
			Workpiece probe calibration	
			with reference data	with any data
			Groove	Surface
_STA1	REAL	R24		
_TDIF	REAL	R37		
_TMV	REAL	R34		
_TNAME	STRING[32]			
_TNUM	INTEGER	R9		
_TUL	REAL	R40		
_TLL	REAL	R41		
_TSA	REAL >0	R36	Safe area	
_TZL	REAL >=0	R33	Zero offset area	
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)	

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 974 CYCLE 994			Workpiece measurement					
Para-meters GUD5	**Type**	**Com-parable Para-meters 840C**	**Possible measuring axes abscissa/ordinate for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2**					
			ZO calculation	**Measure**				
			CYCLE974	CYCLE974		CYCLE994		
			1 point	1 point	1 point with reversal	2 point on diameter		
_CALNUM	INTEGER	R12						
_CORA	REAL	R13						
_CPA	REAL	R20						
_CPO	REAL	R21						
_EVNUM	INTEGER >=0	R11	Empirical value memory number number of data field **GUD5:_EV[_EVNUM-1]**					
				Mean value memory number number of the data field **GUD5:_MV[_EVNUM-1]** Only active if **GUD6:_CHBIT[4]=1**				
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm!					
_ID	REAL	R19						
_INCA	REAL	R26						
_K	INTEGER >0	R29		Weighting factor k for mean value calculation				
_KNUM	INTEGER >=0	R10	Without/with automatic offset of the ZO memory **0** without offset	without / with automatic tool offset (D number) **0** without tool offset				
			1..99 automatic offset in ZO G54...G57 G505...G599 **1000** automatic offset in basic frame G500	Normal D number structure **6 5 4 3 2 1** |__ 1-digit D number |_|__ **0** not assigned[1)] |___ **0/1** length offset in measuring axis **2** radius offset |___ **0** offset normal **1** offset inverted |___ **0** offset corres. 4th digit **1** offset of L1 **2** offset of L2 **3** offset of L3 **4** radius compensation 1) If MD 18105 >9<1000, 3-digit D number		Surface D number structure[2)] **8 7 6 5 4 3 2 1** |_|_|_|_ |__ 5-digit D number |___ **0/1** length offset in measuring axis **2** radius offset |___ **0** offset normal **1** offset inverted |___ **0** offset corres. 4th digit **1** offset of L1 **2** offset of L2 **3** offset of L3 **4** radius compensation 2) If MD 18105 >999 valid also for normal D number structure			

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 974 CYCLE 994			Workpiece measurement					
Para-meters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible measuring axes abscissa/ordinate for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2**					
			ZO calculation	**Measure**				
			CYCLE974	CYCLE974		CYCLE994		
			1 point	1 point	1 point with reversal	2 point on diameter		
_MA	INTEGER >0	R30	Measuring axis 1..2					
_MD	INTEGER	R31						
_MVAR	INTEGER >0	R23	Measurement variant					
			100	0	1000	1 2		
_NMSP	INTEGER >0	R27	Number of measurements at the same location					
_PRNUM	INTEGER >0	R22	Workpiece probe number (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM-1]**)					
_RA	INTEGER	R31						
_RF	REAL	R31						
_SETVAL	REAL	R42 R32	Setpoint	Setpoint (acc. to drawing)				
_SETV[8]	REAL							
_STA1	REAL 0...360 degrees	R26			Initial angle			
_SZA	REAL	R19				Protection zone abscissa (LA)		
_SZO	REAL	R18				Protection zone ordinate (PA)		
_TDIF	REAL >0	R37		Dimensions difference check				
_TNAME	STRING[]			Tool name (alternative for "_TNUM" if tool management active)				
_TNUM	INTEGER >=0	R9		Tool number for automatic tool offset				
_TMV	REAL >0	R34		Offset range with mean value calculation only active if **GUD6:_CHBIT[4]=1**				
_TUL	REAL	R40		Upper tolerance limit (per drawing)				
_TLL	REAL	R41		Lower tolerance limit (per drawing)				
_TSA	REAL >0	R36	Safe area					
_TZL	REAL >=0	R33		Zero offset area				
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)					

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 976			Workpiece measurement					
Para-meters GUD5	Type	Com-parable Para-meters 840C	Possible measuring axes abscissa/ordinate for G17: X=1 Y=2 Z=3 for G18: Z=1 X=2 Y=3 for G19: Y=1 Z=2 X=3					
			Workpiece probe calibration					
			with reference data			with any data		
						Drill-hole with known center	Surface	Drill-hole with unknown center
_CALNUM	INTEGER	R12						
_CORA	REAL 0...359.5	R13	Offset angular position only active if monoprobe					
_CPA	REAL	R20						
_CPO	REAL	R21						
_EVNUM	INTEGER	R11						
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm!					
_ID	REAL	R19						
_INCA	0...360 degrees	R26						
_K	INTEGER	R29						
_KNUM	INTEGER	R10						
_MA	INTEGER >0	R30	Measuring axis					
_MD	INTEGER >0	R31	Measuring direction (0 = positive / 1 = negative)					
_MVAR	INTEGER >0	R23	Measurement variant (see below)					
						xxxx01	x0000	xxxx08

Calibration in plane

```
6 5 4 3 2 1
I I I I I I_____   1   Calibration in drill-hole with known center
I I I I I          8   Calibration in drill-hole with unknown center
I I I I I
I I I I I______    0   With any data in the plane
I I I I
I I I I________    0   Without calculation probe tip
I I I              1   With calculation probe tip (for measurement in plane)
I I I
I I I__________    0   4 Axis directions
I I                1   1 Axis direction (specifying measuring axis and axial direction)
I I                2   2 Axis directions (specifying measuring axis)
I I
I I____________    0   Without position calculation
I I                1   With position calculation
I
I______________    0   Calibration paraxial (in the plane)
                   1   Calibration at any angle (in the plane)
```

Calibration on any surface

_MVAR=0 Calibration on surface

_MVAR=10000 Calibration on surface with calculation of the probe length only permissible with _MA=3!

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

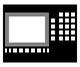
CYCLE 976			Workpiece measurement					
Para-meters GUD5	Type	Com-parable para-meters 840C	Possible measuring axes abscissa/ ordinate for G17: X=1 Y=2 Z=3 for G18: Z=1 X=2 Y=3 for G19: Y=1 Z=2 X=3					
			Workpiece probe calibration					
			with reference data			with any data		
						Drill-hole with known center	Surface	Drill-hole with unknown center
_NMSP	INTEGER >0	R27	Number of measurements at the same location					
_PRNUM	INTEGER >0	R22	Probe type/ workpiece probe number 3 2 1 | |_ |_______ 2-digit number | |____________ 1 Monoprobe 0 Multiprobe (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM(2-digit)-1]**)					
_RA	INTEGER	R31						
_RF	REAL	R31						
_SETVAL	REAL	R32				Calibration setpoint		
_SETV[8]	REAL							
_STA1	REAL	R24				Initial angle		Initial angle
_SZA	REAL	R19						
_SZO	REAL	R18						
_TDIF	REAL	R37						
_TMV	REAL	R34						
_TNAME	STRING[]							
_TNUM	INTEGER	R9						
_TUL	REAL	R40						
_TLL	REAL	R41						
_TSA	REAL >0	R36	Safe area					
_TZL	REAL >=0	R33	Zero offset area					
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)					

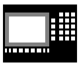
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

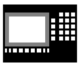
CYCLE 977			Workpiece measurement							
Para-meters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible measuring axes abscissa/ordinate for G17: X=1 Y=2 Z=3 for G18: Z=1 X=2 Y=3 for G19: Y=1 Z=2 X=3**							
			Measure				**ZO calculation**			
			Drill-hole	**Shaft**	**Groove**	**Web**	**Drill-hole**	**Shaft**	**Groo-ve**	**Web**
_CALNUM	INTEGER	R12								
_CORA	REAL 0...359.5	R13	Offset angular position only active if monoprobe							
_CPA	REAL	R20								
_CPO	REAL	R21								
_EVNUM	INTEGER >=0	R11	Empirical value memory number number of data field **GUD5:_EV[_EVNUM-1]**							
			Mean value memory number number of the data field **GUD5:_MV[_EVNUM-1]** Only active if **GUD6:_CHBIT[4]=1**							
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm!							
_ID	REAL	R19	Infeed applicate							
_INCA	REAL 0...360 degrees	R26								
_K	INTEGER >0	R29	Weighting factor k for mean value calculation							
_KNUM	INTEGER >=0	R10	without / with automatic tool offset (D number) **0** without tool offset				without/with automatic offset of the ZO memory **0** without offset			
			Normal D number structure **6 5 4 3 2 1** 1-digit D # **0** not assigned[1)] **0/1** length offset in measuring axis **2** radius offset **0** offset normal **1** offset inverted **0** offset corres. 4th digit **1** offset of L1 **2** offset of L2 **3** offset of L3 **4** Radius comp. 1) If MD 18105 >9<1000, 3-digit D number		Surface D number structure[2)] **8 7 6 5 4 3 2 1** 5-digit D # **0/1** length offset in measuring axis **2** radius offset **0** offset normal **1** offset inverted **0** offset corres. 4th digit **1** offset of L1 **2** offset of L2 **3** offset of L3 **4** radius comp. 2) If MD 18105 >999 also valid for normal D number structure		**1..99** automatic offset in ZO G54...G57 G505...G599 **1000** automatic offset in basic frame G500			
_MA	INTEGER >0	R30			Measuring axis 1..2				Measuring axis 1..2	
_MD	INTEGER	R31								
_MVAR	INTEGER >0	R23	Measuring variant 1xxx measurement with bypassing or consideration of a protection zone							
			1	2	3	4	101	102	103	104

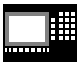
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 977			Workpiece measurement							
Para-meters GUD5	Type	Com-parable para-meters 840C	Possible measuring axes abscissa/ ordinate for G17: X=1 Y=2 Z=3 for G18: Z=1 X=2 Y=3 for G19: Y=1 Z=2 X=3							
			Measure				ZO calculation			
			Drill-hole	Shaft	Groove	Web	Drill-hole	Shaft	Groo-ve	Web
_NMSP	INTEGER >0	R27	Number of measurements at the same location							
_PRNUM	INTEGER >0	R22	Probe type/ workpiece probe number 3 2 1 		_	_______ 2-digit number
_____________ 1 Monoprobe 0 Multiprobe (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM(2-digit)-1]**)							
_RA	INTEGER	R31								
_RF	REAL	R31								
_SETVAL	REAL	R42/ R32	Setpoint (acc. to drawing)				Setpoint			
_SETV[8]	REAL									
_STA1	REAL 0...360 degrees	R26								
_SZA	REAL	R19	Protection zone in abscissa (only for _MVAR=1xxx)							
_SZO	REAL	R18	Protection zone in ordinate (only for _MVAR=1xxx)							
_TDIF	REAL >0	R37	Dimensions difference check							
_TMV	REAL >0	R34	Offset range with mean value calculation							
_TNAME	STRING[32]		Tool name (alt. to "_TNUM" if tool management active)							
_TNUM	INTEGER >=0	R9	Tool number for automatic tool offset							
_TUL	REAL	R40	Upper tolerance limit (per drawing)							
_TLL	REAL	R41	Lower tolerance limit (per drawing)							
_TSA	REAL >0	R36	Safe area							
_TZL	REAL >=0	R33	Zero offset area							
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)							


840 D
NCU 571


840 D
NCU 572
NCU 573


810 D

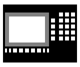

840 Di

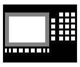
CYCLE 978 **CYCLE 998**			**Workpiece measurement**		
Parameters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible measuring axes abscissa/ordinate** **for G17: X=1 Y=2 Z=3** **for G18: Z=1 X=2 Y=3** **for G19: Y=1 Z=2 X=3**		
			Measure	**ZO calculation**	
			CYCLE978	CYCLE978	CYCLE998
			1 point	1 point	Angle
_CALNUM	INTEGER	R12			
_CORA	REAL 0...359.5	R13	Offset angular position only active if monoprobe		
_CPA	REAL	R20			
_CPO	REAL	R21			
_EVNUM	INTEGER >=0	R11	Empirical value memory number number of data field **GUD5:_EV[_EVNUM-1]** Mean value memory number number of the data field **GUD5:_MV[_EVNUM-1]** Only active if **GUD6:_CHBIT[4]=1**		
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm!		
_ID	REAL	R19			Infeed Offset axis
_INCA	INTEGER	R26			
_K	INTEGER >0	R29	Weighting factor k for mean value calculation		
_KNUM	INTEGER >=0	R10	Normal D number structure 6 5 4 3 2 1 │ │ │ │ │ └ 1-digit D # │ │ │ │ │ │ │ │ └─┴── **0** n. ass. 1) │ │ │ │ │ └────── **0/1** │ │ length offset │ │ in meas. axis │ │ **2** radius comp. │ └────── **0** offset normal │ **1** offset inverted └──────── **0** offset corres. 4th digit **1** offset of L1 **2** offset of L2 **3** offset of L3 **4** Radius comp. 1) If MD 18105 >9<1000, 3-digit D number Surface D number structure 2) 8 7 6 5 4 3 2 1 │ │ │ └─┴─┴─┴─┴ 5-digit D # │ │ │ │ │ │ │ │ │ │ │ └────── **0/1** │ │ length offset │ │ in measuring axis │ │ **2** radius offset │ └──────── **0** offset normal │ **1** offset inverted └──────── **0** offset corres. 4th digit **1** offset of L1 **2** offset of L2 **3** offset of L3 **4** radius comp. 2) If MD 18105 >900 also valid for normal D number structure	without/with autom. offset of the ZO memory **0** without offset **1..99** automatic offset in ZO G54...G57 G505...G599 **1000** automatic offset in basic frame G500	

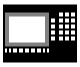
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

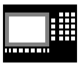
CYCLE 978 CYCLE 998			Workpiece measurement		
Parameters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible measuring axes abscissa/ordinate** **for G17: X=1 Y=2 Z=3** **for G18: Z=1 X=2 Y=3** **for G19: Y=1 Z=2 X=3**		
			Measure	**ZO calculation**	
			CYCLE978	CYCLE978	CYCLE998
			1 point	1 point	Angle
_MA	INTEGER >0	R30	Measuring axis 1..3		Offset axis/measuring axis 102 ...301 | |__ measuring axis |____ offset axis
_MD	INTEGER	R31			
_MVAR	INTEGER >=0	R23	Measurement variant 0	100	105
			1000*	1100*	1105*
_NMSP	INTEGER	R27	Number of measurements at the same location		
_PRNUM	INTEGER >0	R22	Probe type/ workpiece probe number 3 2 1 | |_ |_______ 2-digit number | |____________ 1 Monoprobe 0 Multiprobe (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM(2-digit)-1]**)		
_SETVAL	REAL	R32	Setpoint (acc. to drawing)	Setpoint	Setpoint Approach position
_RA	INTEGER	R31			
_RF	REAL	R31			
_STA1	REAL 0...360 degrees	R24			Setpoint Angle
_SZA	REAL	R19			
_SZO	REAL	R18			
_TDIF	REAL >0	R37	Dimensions difference check		
_TMV	REAL >0	R34	Offset range with mean value calculation only active if **GUD6:_CHBIT[4]=1**		
_TNAME	STRING[]		Tool name (alt. to "_TNUM" if tool management active)		
_TNUM	INTEGER >=0	R9	Tool number for automatic tool offset		
_TUL	REAL	R40	Upper tolerance limit (per drawing)		
_TLL	REAL	R41	Lower tolerance limit (per drawing)		
_TSA	REAL >0	R36	Safe area		
_TZL	REAL >=0	R33	Zero offset area		
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)		

* Difference measurement (not with monoprobe)


840 D
NCU 571


840 D
NCU 572
NCU 573


810 D


840 Di

CYCLE 979			Workpiece measurement							
Para-meters GUD5	Type	Com-parable para-meters 840C	Possible measurements G17: X -Y plane G18: Z -X plane G19: Y -Z plane							
			Measure				ZO calculation			
			Drill-hole	Shaft	Groove	Web	Drill-hole	Shaft	Groove	Web
_CALNUM	INTEGER	R12								
_CORA	REAL 0...359.5	R13	Offset angular position only active if monoprobe							
_CPA	REAL	R20	Center abscissa (with reference to the workpiece zero)							
_CPO	REAL	R21	Center ordinate (with reference to the workpiece zero)							
_EVNUM	INTEGER >=0	R11	Empirical value memory number number of data field **GUD5:_EV[_EVNUM-1]** Mean value memory number number of the data field **GUD5:_MV[_EVNUM-1]** Only active if **GUD6:_CHBIT[4]=1**							
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always 1mm!							
_ID	REAL	R19				Infeed applicate				Infeed applicate
_INCA	REAL	R26	Indexing angle				Indexing angle			
_K	INTEGER >0	R29	Weighting factor k for mean value calculation							
_KNUM	INTEGER >=0	R10	without/with automatic tool offset (D number) **0** without tool offset				without/with autom. offset of the ZO memory **0** without offset			
			Normal D number structure **6 5 4 3 2 1** 1-digit D No. **0** unassigned [1] **0/1** length com-pensation in measuring axis **2** radius compensation **0** normal compensation **1** inverted compensation **0** compens. corr. to 4th digit **1** comp. of L1 **2** comp. of L2 **3** comp. of L3 radius comp. 1) If MD 18105 >9<1000, 3-digit D number		Surface D number structure[2] **8 7 6 5 4 3 2 1** 5-digit D No. **0/1** length comp. in measuring axis **2** radius compensation **0** normal compensation **1** inverted compensation **0** compensation corr. to 4th digit **1** comp. of L1 **2** comp. of L2 **3** comp. of L3 **4** radius comp 2) If MD 18105 >999 also valid for normal D number structure		**1..99** automatic offset in ZO G54...G57 G505...G599 **1000** automatic offset in basic frame G500			

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 979			Workpiece measurement							
Para-meters GUD5	Type	Com-parable para-meters 840C	Possible measurements G17: X -Y plane G18: Z -X plane G19: Y -Z plane							
			Measure				ZO calculation			
			Drill-hole	Shaft	Groove	Web	Drill-hole	Shaft	Groove	Web
_MA	INTEGER	R30								
_MVAR	INTEGER >0	R23	Measurement variant 1	2	3	4	101	102	103	104
_NMSP	INTEGER >0	R27	Number of measurements at the same location							
_PRNUM	**INTEGER >0**	R22	Number of >measuring points/ probe type/ workpiece probe number **4 3 2 1** | | |_|_______ 2-digit number | | | |___________ 1 Monoprobe | 0 Multiprobe |_____________ 0 3 measuring points 1 4 measuring points (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM(2-digit) –1]**							
_RA	INTEGER	R31								
_RF	REAL	R31	Velocity for circular interpol.				Velocity for circular interpol.			
_SETVAL	REAL	R32 R42	Setpoint (acc. to drawing)				Setpoint			
_SETV[8]	REAL									
_STA1	REAL	R24	Initial angle							
_SZA	REAL	R19								
_SZO	REAL	R18								
_TDIF	REAL >0	R37	Dimensions difference check							
_TMV	REAL >0	R34	Offset range with mean value calculation only active if **GUD6:_CHBIT[4]=1**							
_TNAME	STRING[]		Tool name (alt. to "_TNUM" if tool management active)							
_TNUM	INTEGER >=0	R9	Tool number for automatic tool offset							
_TUL	REAL	R40	Upper tolerance limit (per drawing)							
_TLL	REAL	R41	Lower tolerance limit (per drawing)							
_TSA	REAL >0	R36	Safe area							
_TZL	REAL >=0	R33	Zero offset area							
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)							

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 982			Tool measurement of turning, drilling, and milling tools for turning machines		
Parameters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible axes abscissa/ordinate for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2**		
			Calibrate tool probe	**Measure tool**	**Measure tool automatically**
_CALNUM	INTEGER	R12			
_CORA	REAL 0...359.5	R13		Offset angle after reversal when measuring milling tools	
_CPA	REAL	R20			
_CPO	REAL	R21			
_EVNUM	INTEGER >=0	R11		Empirical value memory number number of data field **GUD5:_EV[_EVNUM-1]**	
_FA	REAL >0	R28	Multiplication factor for measurement distance "2a", "a" always being 1mm!		
_ID	REAL	R19			
_INCA	REAL 0...360 degrees	R26			
_K	INTEGER	R29			
_KNUM	INTEGER	R10			
_MA	INTEGER >0	R30	Measuring axis 1..2		
_MD	INTEGER	R31			
_MVAR	INTEGER >=0	R23	Measurement variant		
			0	xxx01	xxx02

_MVAR digits: **5 4 3 2 1**

Digit 1:
- **0** calibrate
- **1** measurement of turning (SL 1-8), milling, and drilling tools measuring axis in _MA
- **2** automatic measurement in the abscissa and ordinate

Digit 2:
- **0** always 0

---

only for milling tools, setting data SD42950=2!

Digit 3:
- **0** measurement/autom. measurement without reversal
- **1** measurement/autom. measurement with reversal

Digit 4:
- **0** measurement: correct length only
- **1** measurement: correct radius only
- **2** measurement: correct length and radius
- **3** autom. measurement: correct length and radius travel round the measurement cube opposite the starting position side

Digit 5:
- **0** axial position of the milling tool/drill (radius in ordinate, for G18: X axis)
- **1** radial position of the milling tool/drill (radius in abscissa, for G18: Z axis)

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

CYCLE 982			Tool measurement of turning, drilling, and milling tools for turning machines		
Parameters GUD5	**Type**	**Com-parable para-meters 840C**	**Possible axes abscissa/ordinate for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2**		
			Calibrate tool probe	**Measure tool**	**Measure tool automatically**
_NMSP	INTEGER >0	R27	Number of measurements at the same location		
_PRNUM	INTEGER >0	R22	Tool probe number (number of the data field assigned to the tool probe **GUD6:_TP[_PRNUM-1]**)		
_RA	INTEGER	R31			
_RF	REAL	R31			
_SETVAL	REAL	R32			
_SETV[8]	REAL				
_STA1	REAL 0...360 degrees	R24		Starting angle when measuring milling tools	
_SZA	REAL	R19			
_SZO	REAL	R18			
_TDIF	REAL >0	R37		Dimensions difference check	
_TMV	REAL	R34			
_TNAME	STRING[]				
_TNUM	INTEGER	R9			
_TUL	REAL	R40			
_TLL	REAL	R41			
_TSA	REAL >0	R36	Safe area		
_TZL	REAL >=0	R33	Zero offset area		
_VMS	REAL >=0	R25	Variable measuring velocity (if 0 150/300mm/min)		

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Result parameters calibration

							CYCLE971	CYCLE972	CYCLE973	CYCLE976
GUD5										
_OVR [0]	REAL									
_OVR [1]	REAL									
_OVR [2]	REAL									
_OVR [3]	REAL									
_OVR [4]	REAL	Actual value	Probe ball diameter							
_OVR [5]	REAL	Difference	Probe ball diameter							
_OVR [6]	REAL									
_OVR [7]	REAL									
_OVR [8]	REAL	Trigger point	Minus	direction	Actual value	Abscissa				
_OVR [9]	REAL	Trigger point	Minus	direction	Difference	Abscissa				
_OVR [10]	REAL	Trigger point	Plus	direction	Actual value	Abscissa				
_OVR [11]	REAL	Trigger point	Plus	direction	Difference	Abscissa				
_OVR [12]	REAL	Trigger point	Minus	direction	Actual value	Ordinate				
_OVR [13]	REAL	Trigger point	Minus	direction	Difference	Ordinate				
_OVR [14]	REAL	Trigger point	Plus	direction	Actual value	Ordinate				
_OVR [15]	REAL	Trigger point	Plus	direction	Difference	Ordinate				
	REAL	Trigger point	Minus	direction	Actual value	Applicate				
_OVR [17]	REAL	Trigger point	Minus	direction	Difference	Applicate				
_OVR [18]	REAL	Trigger point	Plus	direction	Actual value	Applicate				
_OVR [19]	REAL	Trigger point	Plus	direction	Difference	Applicate				
_OVR [20]	REAL	Positional deviation				Abscissa				
_OVR [21]	REAL	Positional deviation				Ordinate				
_OVR [22]	REAL									
_OVR[23]	REAL									
_OVR [24]	REAL									
_OVR[25]	REAL									
_OVR[26]	REAL									
_OVR [27]	REAL	Zero offset area								
_OVR [28]	REAL	Safe area								
_OVR [29]	REAL	Permissible dim. difference								
_OVI [0]	INTEGER									
_OVI[1]	INTEGER									
_OVI [2]	INTEGER	Measuring cycle number								
_OVI [3]	INTEGER	Measurement variant								
_OVI [4]	INTEGER									
_OVI [5]	INTEGER	Probe number								
_OVI [6]	INTEGER									
_OVI [7]	INTEGER									
_OVI [8]	INTEGER									
_OVI [9]	INTEGER	Alarm number								

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

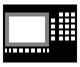
Result parameters measurement (turning machines)

GUD5				CYCLE974	CYCLE994	CYCLE972 CYCLE982
_OVR [0]	REAL	Setpoint		Measuring axis	Diameter/radius	
_OVR [1]	REAL	Setpoint		Abscissa	Abscissa	
_OVR [2]	REAL	Setpoint		Ordinate	Ordinate	
_OVR [3]	REAL	Setpoint				
_OVR [4]	REAL	Actual value		Measuring axis	Diameter/radius	
_OVR [5]	REAL	Actual value			Abscissa	
_OVR [6]	REAL	Actual value			Ordinate	
_OVR [7]	REAL	Actual value				
_OVR [8]	REAL	Tolerance	Upper limit	Measuring axis	Diameter/radius	Actual value L1
_OVR [9]	REAL	Tolerance	Upper limit			Difference L1
_OVR [10]	REAL	Tolerance	Upper limit			Actual value L2
_OVR [11]	REAL	Tolerance	Upper limit			Difference L2
_OVR[12	REAL	Tolerance	Lower limit	Measuring axis	Diameter/radius	Actual value radius only CYCLE982
_OVR [13]	REAL	Tolerance	Lower limit			Difference radius only CYCLE982
_OVR [14]	REAL	Tolerance	Lower limit			
_OVR [15]	REAL	Tolerance	Lower limit			
_OVR [16]	REAL	Difference		Measuring axis	Diameter/radius	
_OVR [17]	REAL	Difference			Abscissa	
_OVR [18]	REAL	Difference			Ordinate	
_OVR [19]	REAL	Difference				
_OVR [20]	REAL	Offset value				
_OVR [21]	REAL					
_OVR [22]	REAL					
_OVR[23]	REAL					
_OVR [24]	REAL					
_OVR[25]	REAL					
_OVR[26]	REAL					
_OVR [27]	REAL	Zero offset area				
_OVR [28]	REAL	Safe area				
_OVR [29]	REAL	Permissible dimen. difference				
_OVR [30]	REAL	Empirical value				
_OVI [0]	INTEGER	D # / ZO #				
_OVI[1]	INTEGER					
_OVI [2]	INTEGER	Measuring cycle number				
_OVI [3]	INTEGER	Measurement variant				
_OVI [4]	INTEGER					
_OVI [5]	INTEGER	Probe number				
_OVI [6]	INTEGER					
_OVI [7]	INTEGER					
_OVI [8]	INTEGER	Tool number				
_OVI [9]	INTEGER	Alarm number				
_OVI[11]	INTEGER	Status offset request				

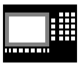
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

For measurement with automatic tool offset only

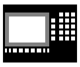
Result parameters measurement (milling and machining centers)							
GUD5				CYCLE977	CYCLE978	CYCLE979	CYCLE998
_OVR [0]	REAL	Setpoint		Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Angle
_OVR [1]	REAL	Setpoint		Abscissa	Abscissa	Abscissa	
_OVR [2]	REAL	Setpoint		Ordinate	Ordinate	Ordinate	
_OVR [3]	REAL	Setpoint			Applicate		
_OVR [4]	REAL	Actual value		Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	
_OVR [5]	REAL	Actual value		Abscissa		Abscissa	
_OVR [6]	REAL	Actual value		Ordinate		Ordinate	
_OVR [7]	REAL	Actual value					
_OVR [8]	REAL	Tolerance	Upper limit	Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Angle
_OVR [9]	REAL	Tolerance	Upper limit	Abscissa		Abscissa	
_OVR [10]	REAL	Tolerance	Upper limit	Ordinate		Ordinate	
_OVR [11]	REAL	Tolerance	Upper limit				
_OVR [12]	REAL	Tolerance	Lower limit	Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	
_OVR [13]	REAL	Tolerance	Lower limit	Abscissa		Abscissa	
_OVR [14]	REAL	Tolerance	Lower limit	Ordinate		Ordinate	
_OVR [15]	REAL	Tolerance	Lower limit				
_OVR [16]	REAL	Difference		Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Angle
_OVR [17]	REAL	Difference		Abscissa		Abscissa	
_OVR [18]	REAL	Difference		Ordinate		Ordinate	
_OVR [19]	REAL	Difference					
_OVR [20]	REAL	Offset value					
_OVR [21]	REAL						
_OVR [22]	REAL						
_OVR[23]	REAL						



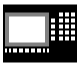
840 D
NCU 571



840 D
NCU 572
NCU 573



810 D



840 Di

_OVR [24]	REAL					
_OVR[25]	REAL		a			
_OVR[26]	REAL					
_OVR [27]	REAL	Zero offset area				
_OVR [28]	REAL	Safe area				
_OVR [29]	REAL	Permissible dimension difference				
_OVR [30]	REAL	Empirical value				
_OVR [31]	REAL	Mean value				
_OVI [0]	INTEGER	D # / ZO #				
_OVI[1]	INTEGER					
_OVI [2]	INTEGER	Measuring cycle number				
_OVI [3]	INTEGER	Measurement variant				
_OVI [4]	INTEGER	Weighting factor				
_OVI [5]	INTEGER	Probe number				
_OVI [6]	INTEGER	Mean value memory number				
_OVI [7]	INTEGER	Empirical value memory number				
_OVI [8]	INTEGER	Tool number				
_OVI [9]	INTEGER	Alarm number				
_OVI[11]	INTEGER	Status offset request				
_OVI12]	INTEGER	Internal error number of the measure function				

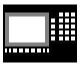
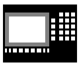
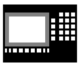
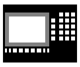
840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

MD number	Identifier	Description	Max. input value	Default value	Value for measuring cycles
NC machine data					
10132	MMC-CMD-TIMEOUT	Monitoring time for MMC command in part program	100	1	3
11420	LEN_PROTOCOL_FILE	File size for log files	100	1	5
13200	MEAS_PROBE_LOW_ACTIV	Switching performance of the probe 0= 0V ➔ 24V; 1= 24V ➔ 0V	TRUE	0	0
18118	MM_NUM_GUD_MODULES	Number of data blocks	9	7	7
18120	MM_NUM_GUD_NAMES_NCK	Number of GUD variables in PLC	400	10	20
18130	MM_NUM_GUD_NAMES_CHAN	Number of GUD variables per channel	200	10	100
18150	MM_GUD_VALUES_MEM	Memory space for the values of the GUD variables	50	12	60
18170	MM_NUM_MAX_FUNC_NAMES	Number of special functions (cycles, DRAM)	plus	40	70
18180	MM_NUM_MAX_FUNC_PARAM	Number of special functions (cycles, DRAM)	plus	300	600
28020	MM_NUM_LUD_NAMES_TOTAL	Number of LUD variables in total (in all program levels)	300	200	200
28082	MM_SYSTEM_FRAME_MASK (measuring cycles SW 6 and higher)	Channel-specific system frames	7FH	21H	21H (Bit0, 5=1)
NC machine data for measurement in JOG (SW 5.3 and higher)					
11602	ASUP_START_MASK	Ignore reasons for stopping ASUB	3	0	1, 3 Bit0=1
11604	ASUP_START_PRIO_LEVEL	Priority for "ASUP_START_MASK" active	64H	0	from 1 to 64H
20110	RESET_MODE_MASK	Define control default setting after power-up and RESET	07FFFH	0	at least 4045H (Bit0, 2, 6, 14=1)
20112	START_MODE_MASK	Define control default setting after part program start	07FFFH	400H	400H (Bit6=0)

Cycle machine data

The measuring cycle data are stored in modules GUD5 and GUD6.

Module	Identifier	Description
General		
GUD6	_CBIT[16]	Central measuring cycle bits
GUD6	_CVAL[4]	Central values
GUD6	_TP[3,10]	Tool probe
GUD6	_WP[3,11]	Workpiece probe
GUD6	_KB[3,7]	Calibration block
GUD6	_CM[8]	Monitoring functions for tool measurement with rotating spindle (rotating tool)
GUD6	_MFS[6]	Feedrates and speeds during measurement with rotating tool
GUD6	_SI[2]	Central measuring cycle strings
Channel-specific		
GUD6	_CHBIT[20]	Channel-specific measuring cycle bits
GUD6	_EVMVNUM[2]	Number of empirical values and mean values
GUD6	_SPEE[4]	Traversing velocities for intermediate positioning
GUD5	_EV[20]	Empirical values
GUD5	_MV[20]	Mean values

			
840 D NCU 571	840 D NCU 572 NCU 573	810 D	840 Di

Cycle machine data for measuring in JOG

The measuring cycle data for measurement in JOG are in modules GUD6 and GUD7

Module	Identifier	Description
Channel-specific		
GUD6	_JM_I[5]	INT value field for JOG measurement
GUD6	_JM_B[7]	Boolean values field for JOG measurement
GUD7	E_MESS_MS_IN	Input workpiece probe
GUD7	E_MESS_MT_IN	Input tool probe
GUD7	E_MESS_D	Internal data item
GUD7	E_MESS_D_M	Measuring path for manual measuring [mm] (in front of and behind the measuring point)
GUD7	E_MESS_D_L	Measuring path for length measurement [mm] (in front of and behind the measuring point) for tool measurement
GUD7	E_MESS_D_R	Measuring path for radius measurement [mm] (in front of and behind the measuring point) for tool measurement
GUD7	E_MESS_FM	Measuring feedrate [mm/min]
GUD7	E_MESS_F	Plane feedrate for collision monitoring [mm/min]
GUD7	E_MESS_FZ	Infeed feedrate for collision monitoring [mm/min]
GUD7	E_MESS_MAX_V	Max. peripheral speed for measuring with rotating spindle [m/min]
GUD7	E_MESS_MAX_S	Max. spindle speed for measuring with rotating spindle [rpm]
GUD7	E_MESS_MAX_F	Max. feedrate for measuring with rotating spindle [mm/min]
GUD7	E_MESS_MIN_F	Min. feedrate for measuring with rotating spindle [mm/min]
GUD7	E_MESS_MIN_D	Measuring accuracy for measuring with rotating spindle [mm/min]
GUD7	E_MESS_MT_TYP[3]	Type tool probe
GUD7	E_MESS_MT_AX[3]	Permissible axis directions for tool probe
GUD7	E_MESS_MT_DL[3]	Diameter of tool measuring probe for length measurement [mm]
GUD7	E_MESS_MT_DR[3]	Diameter of tool measuring probe for radius measurement [mm]
GUD7	E_MESS_MT_DZ[3]	Infeed for measurement tool probe diameter
GUD7	E_MESS_MT_DIR[3]	Approach direction in the plane tool probe
GUD7	E_MESS[3]	Internal data item
GUD7	E_MEAS	Internal data item

Cycle machine data for logging

The cycle data for logging are in module GUD6

Module	Identifier	Description
General		
GUD6	_PROTNAME[2]	String field for log header (32 characters)
GUD6	_HEADLINE[10]	String field for log header (80 characters)
GUD6	_PROTFORM[6]	Int field for formatting for log
GUD6	_PROTSYM[2]	Char field for separator in the log
GUD6	_PROTVAL[13]	Strings for log content (80 characters)
GUD6	_PMI[4]	Int field for internal flags for logging
GUD6	_SP_B[20]	Int field for variable column widths
GUD6	_TXT[100]	String field for formatted strings (12 characters)
GUD6	_DIGIT	Integer number of decimal places

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Central values					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_CVAL	**Number of elements**			
GUD 6	_CVAL[0]	Number of tool probes			3
GUD 6	_CVAL[1]	Number of workpiece probes			3
GUD 6	_CVAL[2]	Number of the gauging block			3
GUD 6	_CVAL[3]	Currently not assigned			
	_TP	**Tool probe**			
	Assignment for milling				
GUD 6	_TP[x,0]	Trigger point in minus direction, abscissa			0
GUD 6	_TP[x,1]	Trigger point in plus direction, abscissa			0
GUD 6	_TP[x,2]	Trigger point in minus direction, ordinate			0
GUD 6	_TP[x,3]	Trigger point in plus direction, ordinate			0
GUD 6	_TP[x,4]	Trigger point in minus direction, applicate			0
GUD 6	_TP[x,5]	Trigger point in plus direction, applicate			0
GUD 6	_TP[x,6]	Edge length/disk diameter			0
GUD 6	_TP[x,7]	Assigned internally			133
GUD 6	_TP[x,8]	Probe type 0: cube 101: disk in G17 201: disk in G18 301: disk in G19			0
GUD 6	_TP[x,9]	Distance between upper edge of tool probe and lower edge of tool			2
	Assignment for turning				
GUD 6	_TP[x,0]	Trigger point in minus direction, abscissa			0
GUD 6	_TP[x,1]	Trigger point in plus direction, abscissa			0
GUD 6	_TP[x,2]	Trigger point in minus direction, ordinate			0
GUD 6	_TP[x,3]	Trigger point in plus direction, ordinate			0
GUD 6	_TP[x,4]	No meaning			0
	to				
GUD 6	_TP[x,9]	No meaning			0
	_WP	**Workpiece probe**			
GUD 6	_WP[x,0]	Ball diameter			6
GUD 6	_WP[x,1]	Trigger point in minus direction of abscissa			3
GUD 6	_WP[x,2]	Trigger point in plus direction of abscissa			-3
GUD 6	_WP[x,3]	Trigger point in minus direction of ordinate			3
GUD 6	_WP[x,4]	Trigger point in plus direction of ordinate			-3
GUD 6	_WP[x,5]	Trigger point in minus direction of applicate			3
GUD 6	_WP[x,6]	Trigger point in plus direction of applicate			-3
GUD 6	_WP[x,7]	Positional deviation abscissa			0
GUD 6	_WP[x,8]	Positional deviation ordinate			0
GUD 6	_WP[x,9]	Internal value			0
GUD 6	_WP[x,10]	Internal value			0

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840 Di

Central values					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_KB	**Calibration block**			
GUD 6	_KB[x,0]	Groove edge in plus direction, ordinate			0
GUD 6	_KB[x,1]	Groove edge in minus direction, ordinate			0
GUD 6	_KB[x,2]	Groove base in abscissa			0
GUD 6	_KB[x,3]	Groove edge in plus direction, abscissa			0
GUD 6	_KB[x,4]	Groove edge in minus direction, abscissa			0
GUD 6	_KB[x,5]	Upper edge groove in ordinate			0
GUD 6	_KB[x,6]	Groove base in ordinate			0
	_CM	**Monitoring functions _CBIT[12] = 0**			
GUD 6	_CM[x,0]	Max. permissible peripheral speed [m/min]/[feet/min]			60
GUD 6	_CM[x,1]	Max. permissible speed [rpm]			2000
GUD 6	_CM[x,2]	Minimum feedrate for probing [mm/min]			1
GUD 6	_CM[x,3]	Required measuring accuracy [mm]			0,005
GUD 6	_CM[x,4]	Max. permissible feedrate for probing			20
GUD 6	_CM[x,5]	Direction of spindle rotation			4
GUD 6	_CM[x,6]	Feed factor 1			10
GUD 6	_CM[x,7]	Feed factor 2			0
	_MFS	**Speed and feedrate _CBIT[12] = 1**			
GUD 6	_MFS[x,0]	Speed 1st probing			0
GUD 6	_MFS[x,1]	Feed 1st probing			0
GUD 6	_MFS[x,2]	Speed 2nd probing			0
GUD 6	_MFS[x,3]	Feed 2nd probing			0
GUD 6	_MFS[x,4]	Speed 3rd probing			0
GUD 6	_MFS[x,5]	Feed 3rd probing			0
Central value for logging					
GUD 6	**_PROTFORM**	Int field for formatting for log			
GUD 6	_PROTFORM[0]	Number of line per page			60
GUD 6	_PROTFORM[1]	Number of characters per line			80
GUD 6	_PROTFORM[2]	First page number			1
GUD 6	_PROTFORM[3]	Number of header lines			5
GUD 6	_PROTFORM[4]	Number of value lines in the log			1
GUD 6	_PROTFORM[5]	Number of characters per column			12
GUD6	**_PROTSYM**	Separator in the protocol			
GUD6	_PROTSYM[0]	Separators between the values in the log			";"
GUD6	_PROTSYM[1]	Special characters for identification when tolerance limits are exceeded			"**;#**"
GUD6	**_PMI**	Int field for internal flags for logging			
GUD6	_PMI[0]	Current line number			0
GUD6	_PMI[1]	Flag for interim output of log header 1: output log header			0
GUD6	_PMI[2]	Current page number			0
GUD6	_PMI[3]	Number of log files			0
GUD6	**_SP_B**	Int field for variable column widths			
GUD6	_SP_B[0...19]	Internal flag			0
GUD6	_DIGIT	Integer number of decimal places			3

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Central bits					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_CBIT	**Central bits**			
GUD 6	_CBIT[0]	Measurement repetition after violation of dimensional difference and safe area			0
GUD 6	_CBIT[1]	M0 with measurement repetition			0
GUD 6	_CBIT[2]	No M0 for alarm "oversize", "undersize", "permissible dimensional difference exceeded"			0
GUD 6	_CBIT[3]	1 = actual values metric 0 = actual values inch			1
GUD 6	_CBIT[4...7]	currently not assigned			0
GUD 6	_CBIT[8]	Mono probe position offset			0
GUD 6	_CBIT[9]	Assigned internally			0
GUD 6	_CBIT[10]	Log destination			0
GUD 6	_CBIT[11]	Log header			0
GUD 6	_CBIT[12]	0: feedrate and speed calculated 1: set by user			0
GUD 6	_CBIT[13]	Reset of _TP[], _WP[], _KB[], _EV[], and _MV[]			0
GUD 6	_CBIT[14]	0: probe length referred to tip equator 1: probe length referred to total length			0
GUD 6	_CBIT[15]	0: no effect 1: calculated probe type is entered in the geo memory (radius)			0

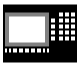
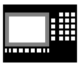
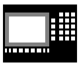
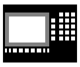
Central strings					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_SI	**Central strings**			
GUD 6	_SI[0]	Currently not assigned			0
GUD 6	_SI[1]	Software version			4
Central strings for logging					
	_PROTNAME (32 chars)				
GUD 6	_PROTNAME [0]	Name of the main program to log from (for log header)			
GUD 6	_PROTNAME [1]	Name of the log file to be created			
GUD 6	**_HEADLINE (80 chars)**	Strings for log header (80 chars)			
GUD 6	_HEADLINE[0....9]	The user can enter customized texts in these strings; they are included in the log			
GUD6	**_PROTVAL (80 chars)**	Strings for log content			
GUD6	_PROTVAL[0]	Content of the header line (line 9)			
GUD6	_PROTVAL[0]	Content of the header line (line 10)			
GUD6	_PROTVAL[2...5]	Specification of the values to be logged in successive lines			
GUD6	_TXT[100]	String field for formatted strings (12 characters)			

Overview of measuring cycle parameters

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Channel-oriented values					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_EVMVNUM	**Number of empirical values and mean values**			
GUD 6	_EVMVNUM[0]	Number of empirical values			20
GUD 6	_EVMVNUM[1]	Number of mean values			20
	_SPEED	**Traversing velocities for intermediate positioning**			
GUD 6	_SPEED[0]	Rapid traverse in % (only active with collision monitoring switched off)	100		50
GUD 6	_SPEED[1]	Positioning velocity in the plane with collision monitoring active			1000
GUD 6	_SPEED[2]	Positioning velocity applicate			1000
GUD 6	_SPEED[3]	Fast measuring feed			900
	_EV	**Empirical values**			
GUD 5	_EV[x]	Empirical value			0
	_MV	**Mean values**			
GUD 5	_MV[x]	Mean value			0

Channel-specific values (for measurement in JOG)					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_JM_I				
GUD 6	**_JM_I** [0]	Workpiece probe number specified 0: specified by _JM_I[1] 1: specified by tool parameters (ShopMill)			0
GUD 6	**_JM_I** [1]	Probe number for workpiece measurement (_PRNUM) only if _JM_I[0]=0			1
GUD 6	**_JM_I** [2]	Probe number for tool measurem. (_PRNUM)			1
GUD 6	**_JM_I** [3]	Working plane 17: G17 18: G18 19: G19 Every other value for working plane is defined in the machine data.			17
GUD 6	**_JM_I** [4]	Definition of the active ZO for measurement 0: G500 1: G54 ... 4: G57 5...: G505... 100: active ZO is defined in machine data			0
GUD7	E_MESS_MS_IN	Input workpiece probe			0
GUD7	E_MESS_MT_IN	Input tool probe			1
GUD7	E_MESS_D	Internal data item			5
GUD7	E_MESS_D_M	Measuring path for manual measuring [mm] (in front of and behind the measuring point)			50
GUD7	E_MESS_D_L	Measuring path for length measurement [mm] (in front of and behind the measuring point) for tool measurement			2
GUD7	E_MESS_D_R	Measuring path for radius measurement [mm] (in front of and behind the measuring point) for tool measurement			1

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Channel-specific values (for measurement in JOG)					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
GUD7	E_MESS_FM	Measuring feedrate [mm/min]			300
GUD7	E_MESS_F	Plane feedrate for collision monitoring [mm/min]			2000
GUD7	E_MESS_FZ	Infeed feedrate for collision monitoring [mm/min]			2000
GUD7	E_MESS_MAX_V	Max. peripheral speed for measuring with rotating spindle [m/min]			100
GUD7	E_MESS_MAX_S	Max. spindle speed for measuring with rotating spindle [rpm]			1000
GUD7	E_MESS_MAX_F	Max. feedrate for measuring with rotating spindle [mm/min]			20
GUD7	E_MESS_MIN_F	Min. feedrate for measuring with rotating spindle [mm/min]			1
GUD7	E_MESS_MIN_D	Measuring accuracy for measuring with rotating spindle [mm/min]			0.01
GUD7	E_MESS_MT_TYP[3]	Type tool probe			0
GUD7	E_MESS_MT_AX[3]	Permissible axis directions for tool probe			133
GUD7	E_MESS_MT_DL[3] 1)	Diameter of tool measuring probe for length measurement [mm]			0
GUD7	E_MESS_MT_DR[3] 1)	Diameter of tool measuring probe for radius measurement [mm]			0
GUD7	E_MESS_MT_DZ[3]	Infeed for measurement tool probe diameter			2
GUD7	E_MESS_MT_DIR[3]	Approach direction in the plane tool probe			-1
GUD7	E_MESS[3]	Internal data item			
GUD7	E_MEAS	Internal data item			

1) During installation value input is mandatory here!

Channel-oriented bits					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_CHBIT	**Channel bits**			
GUD 6	_CHBIT[0]	Measurement input 1 for workpiece measurement			0
GUD 6	_CHBIT[1]	Measurement input 2 for tool measurement			1
GUD 6	_CHBIT[2]	Collision monitoring			1
GUD 6	_CHBIT[3]	Tool offset mode for tool measurement 0: offset in geometry, wear is reset			0
GUD 6	_CHBIT[4]	Without mean value memory			0
GUD 6	_CHBIT[5]	Reverse EV inclusion			0
GUD 6	_CHBIT[6]	Tool offset mode for workpiece measurement with automatic tool offset 0: offset in wear			0
GUD 6	_CHBIT[7]	Measured value offset for CYCLE994 by trigger values (measuring cycles SW 5.4 and higher)			0

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840 Di

GUD 6	_CHBIT[10]	Display of measurement result screen			0
GUD 6	_CHBIT[11]	Acknowledgment measurement result screen with NC start			0
GUD 6	_CHBIT[12]	Currently not assigned			0
GUD 6	_CHBIT[13]	Coupling spindle position with coordinate rotation in the plane			0
GUD 6	_CHBIT[14]	Adapt spindle positioning			0
GUD 6	_CHBIT[15]	0: max. 5 measurement processes 1: only 1 measurement process			0
GUD 6	_CHBIT[16]	Retraction from measuring point in rapid traverse (_CHBIT[2]=1)			0
GUD 6	_CHBIT[17]	Measuring velocity _SPEED[3] and _VMS			0
GUD 6	_CHBIT[18]	Measurement result display is retained until the next measuring cycle is called in the display.			0

Channel-specific bits (for measurement in JOG, SW 5.3 and higher)					
Module	**Identifier**	**Description**	**Max. input value**	**Default value**	**Value for measuring cycles**
	_JM_B				
GUD 6	**_JM_B**[0]	Tool offset mode for tool measurement 0: offset in geometry, wear is reset			0
GUD 6	**_JM_B** [1]	Number of measurement attempts			0
GUD 6	**_JM_B**[2]	Retraction from measuring point in rapid traverse			0
GUD 6	**_JM_B** [3]	Fast measuring feed			0
GUD 6	**_JM_B** [4]	Currently not assigned			0
GUD 6	**_JM_B** [5]	Currently not assigned			0
GUD 6	**_JM_B** [6]	Internal data item			0

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Result parameters measurement									
GUD5				CYCLE974	CYCLE977	CYCLE978	CYCLE979	CYCLE994	CYCLE998
_OVR [0]	REAL	Setpoint		Measuring axis	Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Diameter/ Radius	Angle
_OVR [1]	REAL	Setpoint		Abscissa	Abscissa	Abscissa	Abscissa	Abscissa	
_OVR [2]	REAL	Setpoint		Ordinate	Ordinate	Ordinate	Ordinate	Ordinate	
_OVR [3]	REAL	Setpoint		Applicate		Applicate			
_OVR [4]	REAL	Actual value		Measuring axis	Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Diameter/ Radius	Angle
_OVR [5]	REAL	Actual value		Abscissa	Abscissa		Abscissa	Abscissa	
_OVR [6]	REAL	Actual value			Ordinate		Ordinate	Ordinate	
_OVR [7]	REAL	Actual value							
_OVR [8]	REAL	Tolerance	Upper limit	Measuring axis	Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Diameter/ Radius	Angle
_OVR [9]	REAL	Tolerance	Upper limit		Abscissa		Abscissa	Abscissa	
_OVR [10]	REAL	Tolerance	Upper limit		Ordinate		Ordinate	Ordinate	
_OVR [11]	REAL	Tolerance	Upper limit						
_OVR [12]	REAL	Tolerance	Lower limit	Measuring axis	Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Diameter/ Radius	
_OVR [13]	REAL	Tolerance	Lower limit		Abscissa		Abscissa	Abscissa	
_OVR [14]	REAL	Tolerance	Lower limit		Ordinate		Ordinate	Ordinate	
_OVR [15]	REAL	Tolerance	Lower limit						
_OVR [16]	REAL	Difference		Measuring axis	Drill-hole Shaft Groove Web	Measuring axis	Drill-hole Shaft Groove Web	Diameter/ Radius	Angle
_OVR [17]	REAL	Difference			Abscissa		Abscissa	Abscissa	
_OVR [18]	REAL	Difference			Ordinate		Ordinate	Ordinate	
_OVR [19]	REAL	Difference							
_OVR [20]	REAL	Offset value							
_OVR [21]	REAL								
_OVR [22]	REAL								
_OVR[23]	REAL								
_OVR [24]	REAL								
_OVR[25]	REAL								
_OVR[26]	REAL								
_OVR [27]	REAL	Zero offset area							
_OVR [28]	REAL	Safe area							
_OVR [29]	REAL	Permissible dimension difference							
_OVR [30]	REAL	Empirical value							
_OVR [31]	REAL	Mean value							

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

_OVI [0]	INTEGER	D # / ZO #						
_OVI[1]	INTEGER							
_OVI [2]	INTEGER	Measuring cycle number						
_OVI [3]	INTEGER	Measurement variant						
_OVI [4]	INTEGER	Weighting factor						
_OVI [5]	INTEGER	Probe number						
_OVI [6]	INTEGER	Mean value memory number						
_OVI [7]	INTEGER	Empirical value memory number						
_OVI [8]	INTEGER	Tool number						
_OVI [9]	INTEGER	Alarm number						

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

B Abbreviations

CNC	Computerized Numerical Control
CPU	Central Processing Unit
DIN	Deutsche Industrie Norm (German standard)
DOS	Disk Operating System
DRF	Differential Resolver Function
FM-NC	Function Module - Numerical Control
GUD	Global User Data
I/O	Input/output
LUD	Local User Data
MCS	Machine Coordinate System
MD	Machine Data
MMC	Man-Machine Communication: User interface for NC for operating, programming, and simulation
MS	Microsoft (software manufacturer)
NC	Numerical Control
NCK	Numerical Control Kernel: NC kernel with block preparation, traversing range, etc.
NCU	Numerical Control Unit: Hardware unit of the NCK
PCIN	Name of the software for data exchange with the control
PG	Programming device
PLC	Programmable Logic Control

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840 Di

SR	Subroutine
SW	Software (version)
TO	Tool Offset
TOA	Tool Offset Active: Identification (file type) for tool offsets
UI	User Interface
RS-232-C (V.24)	Serial interface (defines exchange line between DTE and DCE)
WCS	Workpiece Coordinate System
ZO	Zero offset

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

C Terms

Important terms are listed below in alphabetical order. Cross-references to other entries in this glossary are indicated by the symbol ->.

A

Applicable probe types

In order to measure tool and workpiece dimensions, a touch-trigger probe is required that supplies a constant signal (rather than a pulse) when deflected.
The probe type is defined in the measuring cycles in a parameter.
Probes are therefore classified in three groups according to the number of directions in which they can be deflected.

- Multidirectional
- Bidirectional
- Monodirectional

B

Blank measurement

The blank measurement ascertains the position, deviation, and zero offset of the workpiece in the result of a -> workpiece measurement.

C

Calibration

During calibration, the trigger points of the probe are ascertained and stored in the measuring cycles data in the GUD6 module.

Calibration tool

Is a special tool (usually a cylindrical stylus), whose dimensions are known and that is used for precisely determining the distances between the machine zero and the probe trigger point (of the workpiece probe).

Collision monitoring

In the context of measuring cycles, this is a function that monitors all intermediate positions generated within the measuring cycle for the switching signal of the probe. When the probe switches, motion is stopped immediately and an alarm message is output.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

D

Data modules for measuring cycles

Data modules GUD5.DEF and GUD6.DEF contain data required for configuration and execution of the measuring cycles.
These blocks must be loaded into the control during start-up. They must then be adapted according to the characteristics of the relevant machine by the machine manufacturer.
They are stored in the nonvolatile storage area of the control such that their setting values remain stored even when the control is switched off and on.

Delete distance-to-go

If a measuring point is to be approached, a traverse command is transmitted to the position control loop and the probe is moved towards the measuring point. A point behind the expected measuring point is defined as setpoint position. As soon as the probe makes contact, the actual axis value at the time the switching position is reached is measured and the drive is stopped. The remaining "distance-to-go" is deleted.

Differential measurement

Differential measurement means that the measuring point is measured twice, the first time at the probe position reached and the second time with a spindle reversal of 180° (rotation of probe through 180°).

Dimension difference check

Is a tolerance window. On reaching a limit (_TDIF) the tool will probably be worn and have to be replaced. The dimension difference check has no effect on generation of the compensation value.

Display of measuring results

Measuring results can be displayed automatically while a measuring cycle is running.
Activation of this function depends on the configuration of the measuring cycle interface in the MMC and the settings in the measuring cycle data.

E

Empirical value

The empirical values are used to suppress constant dimensional deviations that are not subject to a trend.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

G...J

Inprocess measurement

This method processes the probe signal directly in the NC.

K

L

Logging of measurement results

Measurement results can be optionally be logged in a file located in the part program memory as from SW4. The log can be output from the control either via RS-232-C or on a diskette.

Lower tolerance limit

When measuring a dimensional deviation as the lower tolerance limit (_TLL) ranging between "2/3 tolerance of workpiece" and "Dimensional difference control", this is regarded 100% as tool compensation. The previous average value is erased.
AUTOMATIC operation is interrupted when the tolerance limit of the workpiece is exceeded.
"Undersize" is displayed to the operator depending on the tolerance zone position. Machining can be continued by means of NC start.

M

Mean value

The mean value calculation takes account of the trend of the dimensional deviations of a machining series.
The weighting factor k from which the mean value is derived is selectable.
Mean value calculation alone is not enough to ensure constant machining quality. The measured dimensional deviation can be corrected for constant deviations without a trend by an -> empirical value.

Measure tool

To perform tool measurement, the changed tool is moved up to the probe which is either permanently fixed or swiveled into the working range. The automatically derived tool geometry is entered in the relevant tool offset data record.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Measurement accuracy
The measurement accuracy which can be obtained is thus dependent on the following factors:
- Repeat accuracy of the machine
- Repeat accuracy of the probe
- Resolution of the measuring system

The repeat accuracy of the 840D and FM-NC controls for "on-the-fly measurement" is ±1 μm.

Measurement at any angle
A measurement variant used to measure a drill-hole, shaft, groove, or web at random angles. The measurement path is traveled at a certain set angle.

Measurement path multiplication
The path increment a is normally 1 mm, but can be increased with parameter _FA when measuring cycles are called. -> Measuring path multiplication factor

Measurement variant
The measurement variant of each measuring cycle is defined in parameter _MVAR.
The parameter can have certain integer values for each measuring cycle, which are checked for validity within the cycle.

Measuring path multiplication factor
This parameter (_FA) is used to change the path increment a, which is normally 1 mm, when the measuring cycles are called.

Measuring velocity
The measuring speed can be freely selected by means of parameter _VMS.
The maximum measuring speed must be selected such that safe deceleration within the probe deflecting path is ensured.

Monodirectional probe
This type can only be used for workpiece measurement on milling machines and machining centers with slight limitations.

Multidirectional probe
With this type, measuring cycles for workpiece measurement can be used without limitation.

Multiple measurement at the same location
Parameter _NMSP can be used to determine the number of measurements at the same location.
The setpoint/actual value difference D is determined arithmetically.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840 Di

N

O

Offset angle position
If a -> monoprobe is used, the position of the probe can also be corrected for machine-specific reasons using the parameter _CORA.

P

Paraxial measurement
A measurement variant used for paraxial measurement of a workpieces, such as a drill-hole, shaft, rectangle, etc. The measuring path is traveled paraxially.

Positional deviation
The positional deviation describes the difference between the spindle center and the probe tip center ascertained by calibration. It is compensated for by the measuring cycles.

Probe ball diameter
The diameter of the probe tip. It is ascertained during calibration and stored in the measuring cycle data.

Probe type
In order to measure tool and workpiece dimensions, a touch-trigger probe is required that supplies a constant signal (rather than a pulse) when deflected.
Probes are therefore classified in three groups according to the number of directions in which they can be deflected.

- Multidirectional
- Bidirectional
- Monodirectional

Q

R

Reference groove
A groove located in the working area (permanent feature of the machine) whose precise position is known and that can be used to calibrate workpiece probes.

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840 Di

S

Safe area

The safe area _TSA does not affect the offset value; it is used for diagnosis.
If this limit is reached, there is a defect in the probe or the set position is incorrect.

Setpoint

In the measuring procedure "inprocess measurement", a position is specified as the -> setpoint value for the cycle at which the signal of the touch-trigger probe is expected.

T

Tool name

If tool management is active, the name of the tool can be entered in parameter _TNAME as an alternative to the -> tool number.
The tool number is derived from it within the cycle and entered in _TNUM.

Tool number

The parameter _TNUM contains the tool number of the tool to be automatically offset after workpiece measurement.

Trigger point

The trigger points of the probe are ascertained during calibration and stored in the GUD6 module for the axis direction.

U

Upper tolerance limit

When measuring a dimensional deviation as the upper tolerance limit (_TUL) ranging between "2/3 tolerance of workpiece" and "Dimensional difference control", this is regarded 100% as tool compensation. The previous average value is erased.
AUTOMATIC operation is interrupted when the tolerance limit of the workpiece is exceeded.
"Oversize" is displayed to the operator depending on the tolerance zone position. Machining can be continued by means of NC start.

V

Variable measuring velocity

The measuring speed can be freely selected by means of _VMS.
The maximum measuring speed must be selected such that safe deceleration within the probe deflecting path is ensured. -> Measuring velocity

840 D
NCU 571

840 D
NCU 572
NCU 573

810 D

840 Di

W

Weighting factor for mean value derivation

The weighting factor k can be applied to allow different weighting to be given to an individual measurement.
A new measurement result thus has only a limited effect on the new tool offset as a function of _K.

Workpiece measurement

For workpiece measurement, a measuring probe is moved up to the clamped workpiece in the same way as a tool. The flexibility of the measuring cycles makes it possible to perform nearly all measurements which may need to be taken on a milling machine.

X

Y

Z

Zero offset area

This tolerance range (lower limit _TZL) corresponds to the amount of maximum accidental dimensional deviations. It has to be determined for each machine.

ZO calculation

In the result of a measurement, the actual-setpoint value difference is stored in the data set of any settable zero offset.

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Notes

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

D References

General Documentation

/BU/ SINUMERIK 840D/840Di/810D/802S, C, D
Ordering Information
Catalog NC 60
Order No.: E86060-K4460-A101-A9-7600

/ST7/ **SIMATIC**
SIMATIC S7 Programmable Logic Controllers
Catalog ST 70
Order No.: E86060-K4670-A111-A3

/Z/ SINUMERIK, SIROTEC, SIMODRIVE
Accessories and Equipment for Special-Purpose Machines
Catalog NC Z
Order No.: E86060-K4490-A001-A8-7600

Electronic Documentation

/CD1/ The SINUMERIK System (11.02 Edition)
DOC ON CD
(includes all SINUMERIK 840D/840Di/810D/802 and SIMODRIVE publications)
Order No.: 6FC5 298-6CA00-0BG3

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

User Documentation

/AUK/ SINUMERIK 840D/810D
AutoTurn Short Operating Guide (09.01 Edition)
Order No.: 6FC5 298-4AA30-0BP3

/AUP/ SINUMERIK 840D/810D
AutoTurn Graphic Programming System (02.02 Edition)
Operator's Guide
Programming/Setup
Order No.: 6FC5 298-4AA40-0BP3

/BA/ SINUMERIK 840D/810D
Operator's Guide MMC (10.00 Edition)
Order No.: 6FC5 298-6AA00-0BP0

/BAD/ SINUMERIK 840D/840Di/810D
Operator's Guide HMI Advanced (11.02 Edition)
Order No.: 6FC5 298-6AF00-0BP2

/BEM/ SINUMERIK 840D/810D
Operator's Guide HMI Embedded (11.02 Edition)
Order No.: 6FC5 298-6AC00-0BP2

/BAH/ SINUMERIK 840D/840Di/810D
Operator's Guide HT 6 (06.02 Edition)
Order No.: 6FC5 298-0AD60-0BP2

/BAK/ SINUMERIK 840D/840Di/810D
Short Operating Guide (02.01 Edition)
Order No.: 6FC5 298-6AA10-0BP0

/BAM/ SINUMERIK 810D/840D
Operator's Guide ManualTurn (08.02 Edition)
Order No.: 6FC5 298-6AD00-0BP0

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/BAS/ SINUMERIK 840D/810D
Operator's Guide ShopMill (09.02 Edition)
Order No.: 6FC5 298-6AD10-0BP1

/BAT/ SINUMERIK 840D/810D
Operator's Guide ShopTurn (10.02 Edition)
Order No.: 6FC5 298-6AD50-0BP2

/BAP/ SINUMERIK 840D/840Di/810D
Operator's Guide (04.00 Edition)
Order No.: 6FC5 298-5AD20-0BP1

/BNM/ SINUMERIK 840D/840Di/810D
User's Guide Measuring Cycles (11.02 Edition)
Order No.: 6FC5 298-6AA70-0BP2

/DA/ SINUMERIK 840D/840Di/810D
Diagnostics Guide (11.02 Edition)
Order No.: 6FC5 298-6AA20-0BP3

/KAM/ SINUMERIK 840D/810D
Short Guide ManualTurn (04.01 Edition)
Order No.: 6FC5 298-5AD40-0BP0

/KAS/ SINUMERIK 840D/810D
Short Guide ShopMill (04.01 Edition)
Order No.: 6FC5 298-5AD30-0BP0

/PG/ SINUMERIK 840D/840Di/810D
Programming Guide Fundamentals (11.02 Edition)
Order No.: 6FC5 298-6AB00-0BP2

/PGA/ SINUMERIK 840D/840Di/810D
Programming Guide Advanced (11.02 Edition)
Order No.: 6FC5 298-6AB10-0BP2

/PGK/ SINUMERIK 840D/840Di/810D
Short Guide Programming (02.01 Edition)
Order No.: 6FC5 298-6AB30-0BP1

/PGM/ SINUMERIK 840D/840Di/810D
Programming Guide ISO Milling (11.02 Edition)
Order No.: 6FC5 298-6AC20-0BP2

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/PGT/ SINUMERIK 840D/840Di/810D
Programming Guide ISO Turning (11.02 Edition)
Order No.: 6FC5 298-6AC10-0BP2

/PGZ/ SINUMERIK 840D/840Di/810D
Programming Guide Cycles (11.02 Edition)
Order No.: 6FC5 298-6AB40-0BP2

/PI/ **PCIN 4.4**
Software for Data Transfer to/from MMC Module
Order No.: 6FX2 060-4AA00-4XB0 (Eng., Fr., Ger.)
Order from: WK Fürth

/SYI/ SINUMERIK 840Di
System Overview (02.01 Edition)
Order No.: 6FC5 298-6AE40-0BP0

Manufacturer/Service Documentation

a) Lists

/LIS/ SINUMERIK 840D/840Di/810D/
SIMODRIVE 611D
Lists (11.02 Edition)
Order No.: 6FC5 297-6AB70-0BP3

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

b) Hardware

/BH/ SINUMERIK 840D/840Di/810D
Operator Components Manual (Hardware) (11.02 Edition)
Order No.: 6FC5 297-6AA50-0BP2

/BHA/ SIMODRIVE **Sensor**
Absolute Position Sensor with Profibus-DP
User's Guide (Hardware) (02.99 Edition)
Order No.: 6SN1 197-0AB10-0YP1

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE
EMC Installation Guide (06.99 Edition)
Planning Guide (Hardware)
Order No.: 6FC5 297-0AD30-0BP1

/PHC/ SINUMERIK 810D
Configuring Manual (Hardware) (03.02 Edition)
Order No.: 6FC5 297-6AD10-0BP0

/PHD/ SINUMERIK 840D
Configuring Manual NCU 561.2-573.2 (Hardware) (10.02 Edition)
Order No.: 6FC5 297-6AC10-0BP2

/PHF/ SINUMERIK FM-NC
Configuring Manual NCU 570 (HW) (04.96 Edition)
Order No.: 6FC5 297-3AC00-0BP0

/PMH/ SIMODRIVE **Sensor**
Measuring System for Main Spindle Drives
Configuring/Installation Guide, SIMAG-H (Hardware) (05.99 Edition)
Order No.: 6SN1197-0AB30-0BP0

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

c) Software

/FB1/ SINUMERIK 840D/840Di/810D
Description of Functions, Basic Machine (Part 1) (11.02 Edition)
(the various sections are listed below)
Order No.: 6FC5 297-6AC20-0BP2

A2 Various Interface Signals
A3 Axis Monitoring, Protection Zones
B1 Continuous Path Mode, Exact Stop and Look Ahead
B2 Acceleration
D1 Diagnostic Tools
D2 Interactive Programming
F1 Travel to Fixed Stop
G2 Velocities, Setpoint/Actual-Value Systems, Closed-Loop Control
H2 Output of Auxiliary Functions to PLC
K1 Mode Group, Channels, Program Operation Mode
K2 Axes, Coordinate Systems, Frames
Actual-Value System for Workpiece, External Zero Offset
K4 Communication
N2 EMERGENCY STOP
P1 Transverse Axes
P3 Basic PLC Program
R1 Reference Point Approach
S1 Spindles
V1 Feeds
W1 Tool Compensation

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/FB2/

SINUMERIK 840D/840Di/810D(CCU2)
Description of Functions, Extended Functions (Part 2) (11.02 Edition)
including FM-NC: Turning, Stepper Motor
(the various sections are listed below)
Order No.: 6FC5 297-6AC30-0BP2

A4 Digital and Analog NCK I/Os
B3 Several Operator Panels and NCUs
B4 Operation via PC/PG
F3 Remote Diagnostics
H1 JOG with/without Handwheel
K3 Compensations
K5 Mode Groups, Channels, Axis Replacement
L1 FM-NC Local Bus
M1 Kinematic Transformation
M5 Measurements
N3 Software Cams, Position Switching Signals
N4 Punching and Nibbling
P2 Positioning Axes
P5 Oscillation
R2 Rotary Axes
S3 Synchronous Spindles
S5 Synchronized Actions (up to and including SW 3, then /FBSY/)
S6 Stepper Motor Control
S7 Memory Configuration
T1 Indexing Axes
W3 Tool Change
W4 Grinding

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/FB3/ SINUMERIK 840D/840Di/810D(CCU2)
Description of Functions, Special Functions (Part 3) (11.02 Edition)
(the various sections are listed below)
Order No.: 6FC5 297-6AC80-0BP2

F2 3-Axis to 5-Axis Transformation
G1 Gantry Axes
G3 Cycle Times
K6 Contour Tunnel Monitoring
M3 Coupled Motion and Leading Value Coupling
S8 Constant Workpiece Speed for Centerless Grinding
T3 Tangential Control
TE0 Installation und Archiving of Compile Cycles
TE1 Clearance Control
TE2 Analog Axis
TE3 Master-Slave for drives
TE4 Transformation Package Handling
TE5 Setpoint Exchange
TE6 MCS Coupling
TE7 Retrace Support
TE8 Path-Synchronous Switch Signal
V2 Preprocessing
W5 3D Tool Radius Compensation

/FBA/ SIMODRIVE 611D/SINUMERIK 840D/810D
Description of Functions, Drive Functions (11.02 Edition)
(the various sections are listed below)
Order No.: 6SN1 197-0AA80-0BP9

DB1 Operational Messages/Alarm Reactions
DD1 Diagnostic Functions
DD2 Speed Control Loop
DE1 Extended Drive Functions
DF1 Enable Commands
DG1 Encoder Parameterization
DM1 Calculation of Motor/Power Section Parameters and Controller Data
DS1 Current Control Loop
DÜ1 Monitors/Limitations

/FBAN/ SINUMERIK 840D/SIMODRIVE 611 DIGITAL
Description of Functions **ANA MODULE** (02.00 Edition)
Order No.: 6SN1 197-0AB80-0BP0

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/FBD/
SINUMERIK 840D
Description of Functions **Digitizing** (07.99 Edition)
Order No.: 6FC5 297-4AC50-0BP0

DI1 Start-up
DI2 Scanning with Tactile Sensors (scancad scan)
DI3 Scanning with Lasers (scancad laser)
DI4 Milling Program Generation (scancad mill)

/FBDN/
CAM Integration DNC NT-2000
Description of Functions (01.02 Edition)
System for NC Data Management and Data Distribution
Order No.: 6FC5 297-5AE50-0BP2

/FBDT/
SINUMERIK 840D/840Di/810D
IT Solutions
NC Data Transfer (SinDNC)
Description of Functions (09.01 Edition)
Order No.: 6FC5 297-1AE70-0BP1

/FBFA/
SINUMERIK 840D/840Di/810D
Description of Functions (11.02 Edition)
ISO Dialects for SINUMERIK
Order No.: 6FC5 297-6AE10-0BP2

/FBFE/
SINUMERIK 840D/810D
Description of Functions **Remote Diagnosis** (11.02 Edition)
Order No.: 6FC5 297-0AF00-0BP2

/FBH/
SINUMERIK 840D/810D
HMI Configuring Package (11.02 Edition)
Order No.: (supplied with the software)
Part 1 User's Guide
Part 2 Description of Functions

/FBHLA/
SINUMERIK 840D/SIMODRIVE 611 digital
Description of Functions
HLA Module (04.00 Edition)
Order No.: 6SN1 197-0AB60-0BP2

/FBMA/
SINUMERIK 840D/810D
Description of Functions **ManualTurn** (08.02 Edition)
Order No.: 6FC5 297-6AD50-0BP0

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/FBO/
SINUMERIK 840D/810D
Description of Functions
Configuring OP 030 Operator Interface (09.01 Edition)
(the individual sections are listed below)
Order No.: 6FC5 297-6AC40-0BP0

BA Operator's Guide
EU Development Environment (Configuring Package)
PSE Introduction to Configuring of Operator Interface
IK Screen Kit: Software Update and Configuration

PS Online only: Configuring Syntax (Configuring Package)

/FBP/
SINUMERIK 840D
Description of Functions **C-PLC Programming** (03.96 Edition)
Order No.: 6FC5 297-3AB60-0BP0

/FBR/
SINUMERIK 840D/810D
IT Solutions
Description of Functions **SINCOM Computer Link** (09.01 Edition)
Order No.: 6FC5 297-6AD60-0BP0

NFL Host Computer Interface
NPL PLC/NCK Interface

/FBSI/
SINUMERIK 840D/SIMODRIVE
Description of Functions (09.02 Edition)
SINUMERIK Safety Integrated
Order No.: 6FC5 297-6AB80-0BP1

/FBSP/
SINUMERIK 840D/810D
Description of Functions **ShopMill** (09.02 Edition)
Order No.: 6FC5 297-6AD80-0BP1

/FBST/
SIMATIC
FM STEPDRIVE/SIMOSTEP
Description of Functions (01.01 Edition)
Order No.: 6SN1 197-0AA70-0YP4

/FBSY/
SINUMERIK 840D/810D
Description of Functions **Synchronized Actions** (10.02 Edition)
for wood, glass, ceramics and presses
Order No.: 6FC5 297-6AD40-0BP2

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/FBT/ SINUMERIK 840D/810D
Description of Functions **ShopTurn** (10.02 Edition)
Order No.: 6FC5 297-6AD70-0BP2

/FBTC/ SINUMERIK 840D/810D
IT Solutions
SINUMERIK Tool Data Communication SinTDC (01.02 Edition)
Description of Functions
Order No.: 6FC5 297-5AF30-0BP0

/FBTD/ SINUMERIK 840D/810D
Description of Functions
Tool Information System (SinTDI) with Online Help (02.01 Edition)
Order No.: 6FC5 297-6AE00-0BP0

/FBU/ **SIMODRIVE 611 universal**
Description of Functions (08.02 Edition)
Closed-Loop Control Component for Speed Control and Positioning
Order No.: 6SN1 197-0AB20-0BP6

/FBW/ SINUMERIK 840D/810D
Description of Functions **Tool Management** (10.02 Edition)
Order No.: 6FC5 297-6AC60-0BP1

/FBWI/ SINUMERIK 840D/840Di/810D
Description of Functions **WinTPM** (02.02 Edition)
Order No.: The document is an integral part of the software

/HBA/ SINUMERIK 840D/840Di/810D
Manual @Event (01.02 Edition)
Order No.: 6AU1900-0CL20-0AA0

/HBI/ SINUMERIK 840Di
Manual (09.02 Edition)
Order No.: 6FC5 297-6AE60-0BP0

/INC/ SINUMERIK 840D/840Di/810D
Commissioning Tool **SINUMERIK SinuCOM NC** (02.02 Edition)
Order No.: (an integral part of the Online Help for the start-up tool)

/PFK/ **SIMODRIVE**
Planning Guide **1FT5/1FT6/1FK6 Motors** (12.01 Edition)
AC Servo Motors for Feed and Main Spindle Drives
Order No.: 6SN1 197-0AB20-0BP0

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/PJE/
SINUMERIK 840D/810D
Configuring Package HMI Embedded (08.01 Edition)
Description of Functions: Software Update, Configuration Installation
Order No.: 6FC5 297-6EA10-0BP0
(the document PS Configuring Syntax is supplied with the software and available as a pdf file)

/PJFE/
SIMODRIVE
Planning Guide (09.01 Edition)
Built-In Synchronous Motors 1FE1
AC Motors for Main Spindle Drives
Order No.: 6SN1 197-0AC00-0BP1

/PJLM/
SIMODRIVE
Planning Guide **Linear Motors 1FN1, 1FN3** (11.01 Edition)
ALL General Information about Linear Motors
1FN1 1FN1 AC Linear Motor
1FN3 1FN3 AC Linear Motor
CON Connections
Order No.: 6SN1 197-0AB70-0BP2

/PJM/
SIMODRIVE
Planning Guide **Motors** (11.00 Edition)
AC Motors for Feed and Main Spindle Drives
Order No.: 6SN1 197-0AA20-0BP5

/PJU/
SIMODRIVE 611
Planning Guide **Inverters** (08.02 Edition)
Order No.: 6SN1 197-0AA00-0BP6

/PMS/
SIMODRIVE
Planning Guide **ECO Motor Spindle** (04.02 Edition)
for Main Spindle Drives
Order No.: 6SN1 197-0AD04-0BP0

/POS1/
SIMODRIVE POSMO A
User's Guide (08.02 Edition)
Distributed Positioning Motor on PROFIBUS DP
Order No.: 6SN2 197-0AA00-0BP3

/POS2/
SIMODRIVE POSMO A
Installation Instructions (enclosed with POSMO A) (12.98 Edition)
Order No.: 462 008 0815 00

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/POS3/ **SIMODRIVE POSMO SI/CD/CA**
Distributed Servo Drive Systems, User's Guide (08.02 Edition)
Order No.: 6SN2 197-0AA20-0BP3

/PPH/ **SIMODRIVE**
Planning Guide **1PH2/1PH4/1PH7 Motors** (12.01 Edition)
AC Asynchronous Induction for Main Spindle Drives
Order No.: 6SN1 197-0AC60-0BP0

/PPM/ **SIMODRIVE**
Planning Guide **Hollow-Shaft Motors** (10.01 Edition)
for Main Spindle Drives
1PM4 and 1PM6
Order No.: 6SN1 197-0AD03-0BP0

/S7H/ **SIMATIC S7-300**
Manual: Assembly, CPU Data (Hardware) (10.98 Edition)
Reference Manual: Module Data
Order No.: 6ES7 398-8AA03-8AA0

/S7HT/ **SIMATIC S7-300**
Manual STEP7, Fundamentals, V.3.1 (03.97 Edition)
Order No.: 6ES7 810-4CA02-8AA0

/S7HR/ **SIMATIC S7-300**
Manual (03.97 Edition)
Manual STEP7, Reference Manuals, V.3.1
Order No.: 6ES7 810-4CA02-8AR0

/S7S/ **SIMATIC S7-300**
FM 353 Positioning Module for Stepper Drive (04.97 Edition)
Order together with configuring package

/S7L/ **SIMATIC S7-300**
FM 354 Positioning Module for Servo Drive (04.97 Edition)
Order together with configuring package

/S7M/ **SIMATIC S7-300**
FM 357 Multimodule for Servo and Stepper Drives (10.99 Edition)
Order together with configuring package

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

/SP/ **SIMODRIVE 611-A/611-D**
SimoPro 3.1
Program for Configuring Machine-Tool Drives
Order No.: 6SC6 111-6PC00-0AA❑,
Order from: WK Fürth

d) Installation and Start-up

/IAA/ **SIMODRIVE 611A**
Installation and Start-Up Guide (10.00 Edition)
(incl. description of SIMODRIVE 611D start-up software)
Order No.: 6SN 1197-0AA60-0BP6

/IAC/ SINUMERIK 810D
Installation and Start-Up Guide (03.02 Edition)
(incl. description of SIMODRIVE 611D start-up software)
Order No.: 6FC5 297-6AD20-0BP0

/IAD/ SINUMERIK 840D/SIMODRIVE 611D
Installation and Start-Up Guide (11.02 Edition)
(incl. description of SIMODRIVE 611D start-up software)
Order No.: 6FC5 297-6AB10-0BP2

/IAM/ SINUMERIK 840D/840Di/810D
HMI/MMC Installation and Start-Up Guide (11.02 Edition)
Order No.: 6FC5 297-6AE20-0BP2

AE1	Updates/Supplements
BE1	Expanding the Operator Interface
HE1	Online Help
IM2	Starting up HMI Embedded
IM4	Starting up HMI Advanced
TX1	Creating Foreign Language Texts

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

E Index

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

F Identifiers

List of input/output variables of the measuring cycles

Name	*Stands for*	*Explanation*
_CALNUM	**Cal**ibration groove **num**ber	
_CBIT[16]	**C**entral **Bit**s	Field for NCK-global bits
_CHBIT[16]	**Ch**annel **Bit**s	Field for channel-specific bits
_CM[8]		Field: Monitoring functions for tool measurement with rotating spindle with 8 elements each
_CORA	**Cor**rection **a**ngle position	Offset angle position
_CPA	**C**enter **p**oint **a**bscissa	
_CPO	**C**enter **p**oint **o**rdinate	
_CVAL[4]		Field: Number of elements with e elements each
_DIGIT		Number of decimal places
_EV[20]		20 empirical value memories
_EVMVNUM[2]		Number of empirical values and mean values
_EVNUM		Number of empirical value memory
_FA	Factor for multipl. of measurem. path	Measuring path
_HEADLINE[10]		10 strings for protocol headers
_ID	Infeed in applicate	Incremental infeed depth/offset
_INCA	**Ind**exing **a**ngle	
_K	Weighting factor for averaging	
_KB[3,7]		Field: Gauging block data with 7 elements each
_KNUM		Offset number
_MA	Number of **m**easuring **a**xis	
_MD	**M**easuring **d**irection	
_MFS[]		Field: Feedrates and spindle speeds for tool measurement with rotating spindle with 6 elements each
_MV[20]		20 mean value memory
_MVAR	**M**easuring **var**iant	
_NMSP	**N**umber of **m**easurements at **s**ame **sp**ot	
_OVI[11]		Field: Output values INT
_OVR[32]		Field: Output values REAL
_PRNUM	**Pr**obe type and probe **num**ber	

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

_PROTFORM[6]		Log formatting
_PROTNAME[2]		Name of log file
_PROTSYM[2]		Separator in the log
_PROTVAL[11]		Log header line
_RA	Number of **r**otary **a**xis	
_RF	Feedrate for circular interpolation	Feedrate in circular-path programming
_SETVAL	**Set**point **val**ue	
_SETV[3]		Measure setpoint values on rectangle
_SI[2]	**S**ystem **i**nformation	
_SPEED[3]		Field: Feedrate values
_STA1	**St**arting **a**ngle	
_SZA	**S**afety **z**one on workpiece **a**bscissa	Protection zone in abscissa
_SZO	**S**afety **z**one on workpiece **o**rdinate	Protection zone in ordinate
_TDIF	**T**olerance dimensional **dif**ference check	
_TLL	**T**olerance **l**ower **l**imit	
_TMV		Mean value generation with compensation
_TNAME	**T**ool **name**	Tool name for use in tool management
_TNUM	**T num**ber for automatic tool offset	
_TP[3,10]		Field: Tool probe data with 6 elements each
_TSA	**T**olerance **s**afe **a**rea	
_TUL	**T**olerance **u**pper **l**imit	
_TZL	**T**olerance **z**ero offset range	Zero offset
_VMS	**V**ariable **meas**uring velocity	
_WP[3,11]		Field: Workpiece probe data with 9 elements each

■

840 D NCU 571 | 840 D NCU 572 NCU 573 | 810 D | 840 Di

Notes

<table>
<tr><td>To
SIEMENS AG

A&D MC BMS
P.O. Box 3180
D-91050 Erlangen, Germany
Phone: ++49-(0)180-5050-222 [Hotline]
Fax: ++49-(0)9131-98-2176 [Documentation]
Email: motioncontrol.docu@erlf.siemens.de</td><td>Suggestions

Corrections<hr>For Publication/Manual:

SINUMERIK 840D/840Di/810D

Measuring Cycles

User Documentation</td></tr>
<tr><td>From:

Name<hr>Company/Department
Address:<hr>Zip Code: Town:<hr>Phone: /<hr>Fax: /</td><td>User's Guide

Order No.: 6FC5298-6AA70-0BP2
Edition: 11.02<hr>Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.</td></tr>
</table>

Suggestions and/or corrections

Overview of SINUMERIK 840D/840Di/810D Documentation (11.2002)

General Documentation

- *SINUMERIK* 840D/810D — Brochure
- *SINUMERIK* 840D/840Di/810D/ — Catalog Ordering Info. NC 60 ***)**
- *SINUMERIK SIROTEC SIMODRIVE* Accessories — Catalog Accessories NC-Z

User Documentation

- *SINUMERIK* 840D/810D/FM-NC — **AutoTurn**
  – Short Guide
  – Programming/Setup
- *SINUMERIK* 840D/840Di/810D — Operator's Guide **– HT 6**
- *SINUMERIK* 840D/840Di/810D — **Diagnostics Guide *)**
- *SINUMERIK* 840D/840Di/810D — **Operator's Guide *)**
  – Short Guide
  – HMI Embedded
  – HMI Advanced

User Documentation

- *SINUMERIK* 840D/840Di/810D — **Program. Guide**
  – Short Guide
  – Fundamentals ***)**
  – Advanced ***)**
  – Cycles
  – Measuring Cycles
  – ISO Turning/Milling
- *SINUMERIK* 840D/810D — Operator's Guide
  – **ManualTurn**
  – Short Guide ManualTurn
  – **ShopMill**
  – Short Guide ShopMill
  – **ShopTurn**
  – Short Guide ShopTurn
- *SINUMERIK* 840Di — **System Overview**

Manufacturer/Service Documentation

- *SINUMERIK* — **Configuring** (HW) ***)**
  – 810D
  – 840D
- *SINUMERIK* 840D/840Di/810D — **Operator Components** (HW) ***)**
- *SINUMERIK* 840D/810D — Description of Functions
  – ManualTurn
  – ShopMill
  – ShopTurn
- *SINUMERIK* 840D/840Di/810D — Description of Functions **Synchronized Actions**

Manufacturer/Service Documentation

- *SINUMERIK SIMODRIVE* 611D 840D/810D — Description of Functions **Drive Functions *)**
- *SINUMERIK* 840D/840Di/810D — **Description of Functions**
  – Basic Machine ***)**
  – Extended Functions
  – Special Functions
- *SINUMERIK* 840D/840Di/810D — Description of Functions **Tool Management**
- *SINUMERIK* 840D/810D — **Configuring Kit** HMI Embedded
- *SINUMERIK* 840D/810D — Description of Functions Operator Interface **OP 030**
- *SINUMERIK* 840D/810D — **IT Solutions**
  – Computer Link
  – Tool Data Information System
  – NC Data Management
  – NC Data Transfer
  – Tool Data Communication

Manufacturer/Service Documentation

- *SINUMERIK SIMODRIVE* — Description of Functions SINUMERIK **Safety Integrated**
- *SINUMERIK* 840D — Description of Functions **Digitizing**
- *SINUMERIK SIMODRIVE* — **Installation & Start-Up Guide *)**
  – 810D
  – 840D/611D
  – HMI
- *SINUMERIK SIMODRIVE* 840D/840Di 810D 611D — **Lists *)**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions **Linear Motor**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions
  – Hydraulics Module
  – Analog Module
- *SINUMERIK SIMODRIVE SIROTEC* — **EMC** Guidelines

Electronic Documentation

- *SINUMERIK SIMODRIVE* 840D/840Di/810D 611, Motors — **DOC ON CD *)** The SINUMERIK System

Manufacturer/Service Documentation

- *SINUMERIK* 840D/840Di/810D — Description of Functions **ISO Dialects** for SINUMERIK
- *SINUMERIK* 840Di — **Manual** (HW + Installation and Start-Up)
- *SINUMERIK* 840D/810D — Description of Functions **Remote Diagnosis**
- *SINUMERIK* 840D/840Di/810D — **Manual @ Event**

***)** These documents are a minimum requirement

Siemens AG
Automation & Drives
Motion Control Systems
P.O. Box 3180, D-91050 Erlangen
Germany

www.ad.siemens.de

Order No.: 6FC5298-6AA70-0BP2

Printed in Germany